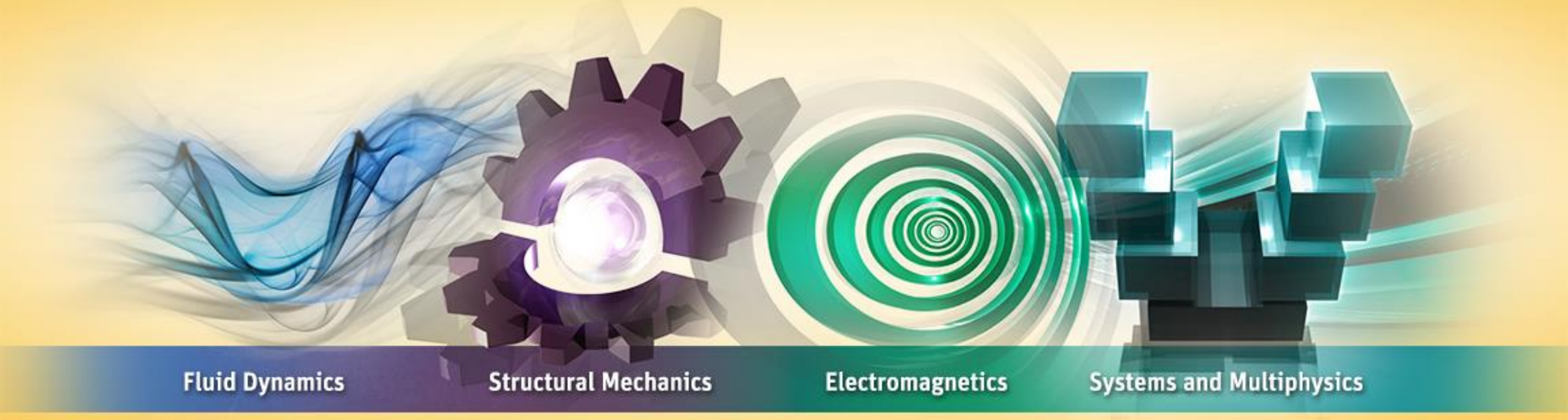
Module 4: Preparing for Analysis



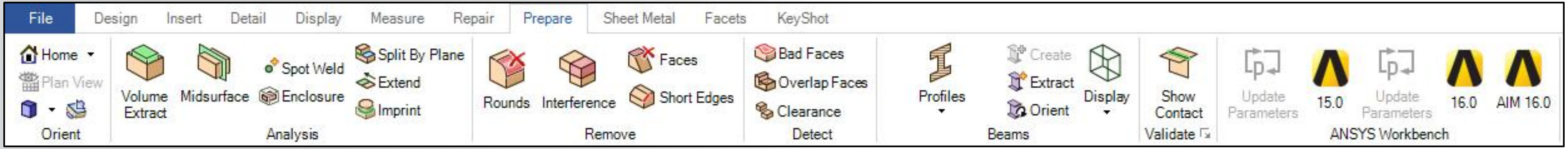
Introduction to ANSYS SpaceClaim Direct Modeler

Realize Your Product Promise®

Overview

In this module we will learn about:

- 3D advanced features
  - Spot welds, Midsurface, Beam extraction
  - Flow volume extraction, Enclosure
- Share topology
  - Show contact
- Material assignment
- Named Selections and Parameters
- Volume decomposition
- Transferring SpaceClaim Model to Workbench



Preprocessing Workflow

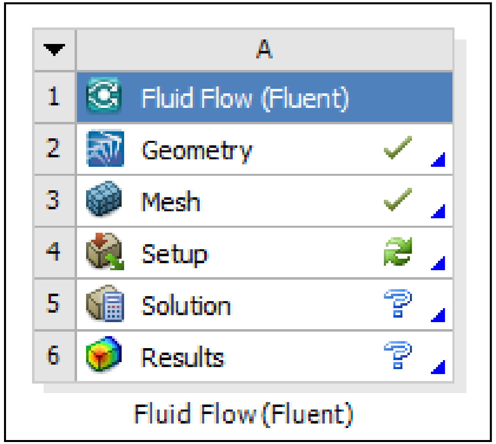
Import/ Geometry Creation
- Sketches and Planes
- 3D Operations
- Geometry Import

Geometry Modifications
- 3D Operations
- Geometry Cleanup and Repair
- Preparation for Analysis

Meshing

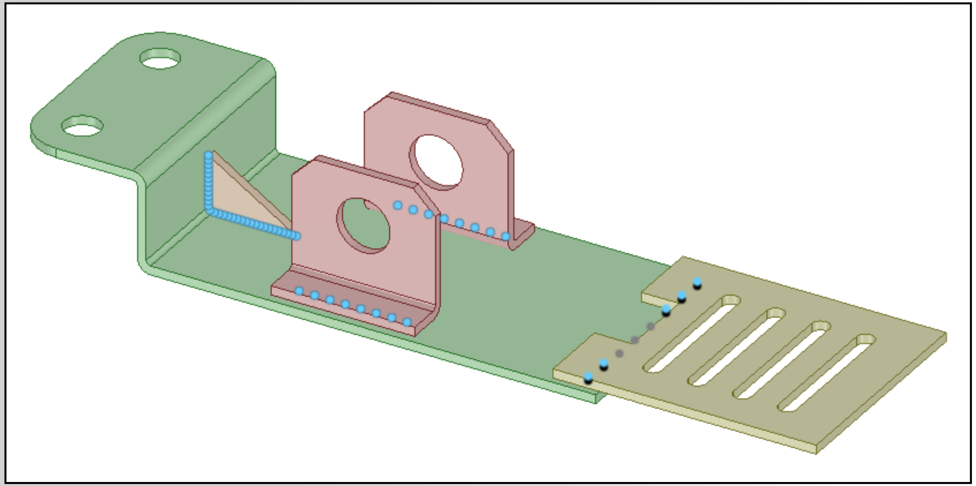
Solver

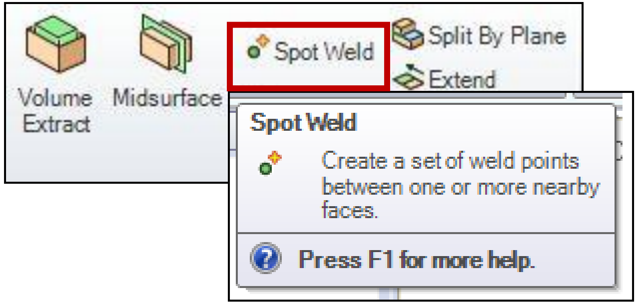
ANSYS SpaceClaim Direct Modeler



Spot Weld

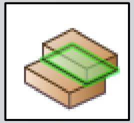
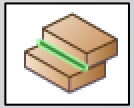
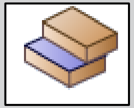
- **Spot Weld:**
  - **Spot weld tool is located on Prepare ribbon**
  - **Spot welds are applied to the edges of a face. These will represent spots that share a common node between the two parts**
  - **The property will automatically pass over as an additional connection between the parts to Workbench**

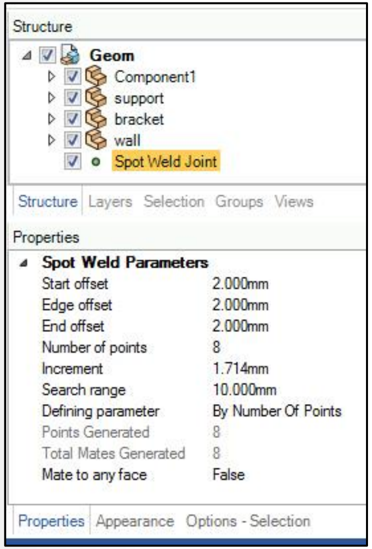


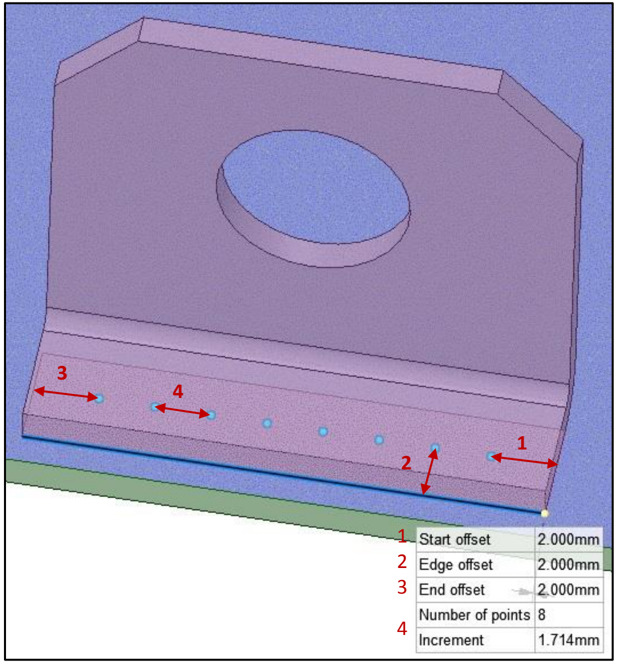


Spot Weld

- **Spot Weld Tool Guide and options**

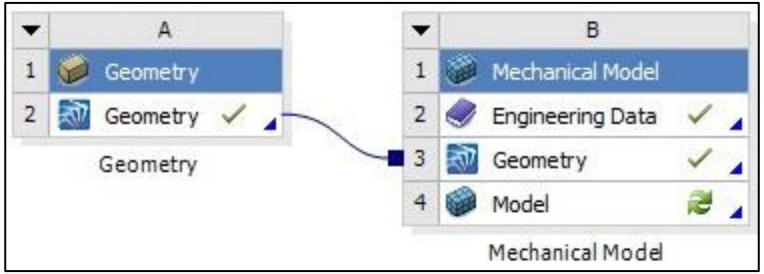
	Use the Select Base Faces tool guide to select the face or faces on which the weld points will be defined. You should select a single face or a chain of tangent faces.
	Use the Select Guiding Edges tool guide to define the edge along which the weld points will be defined.
	Use the Select Mating Faces 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 Complete tool guide completes the spot weld definition.

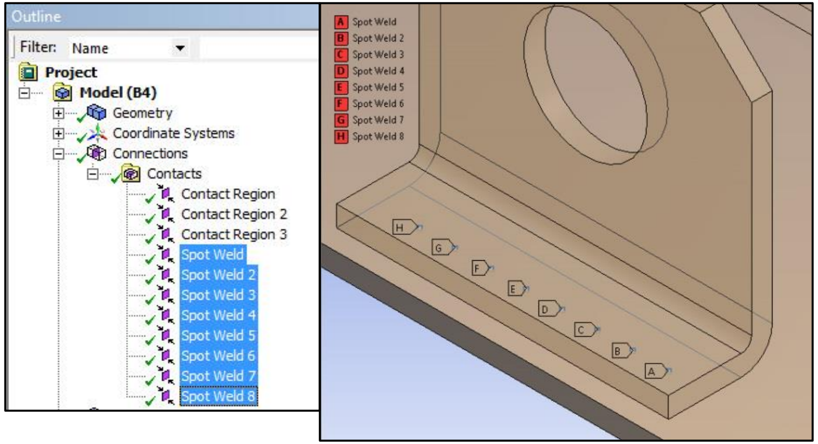




Spot Weld

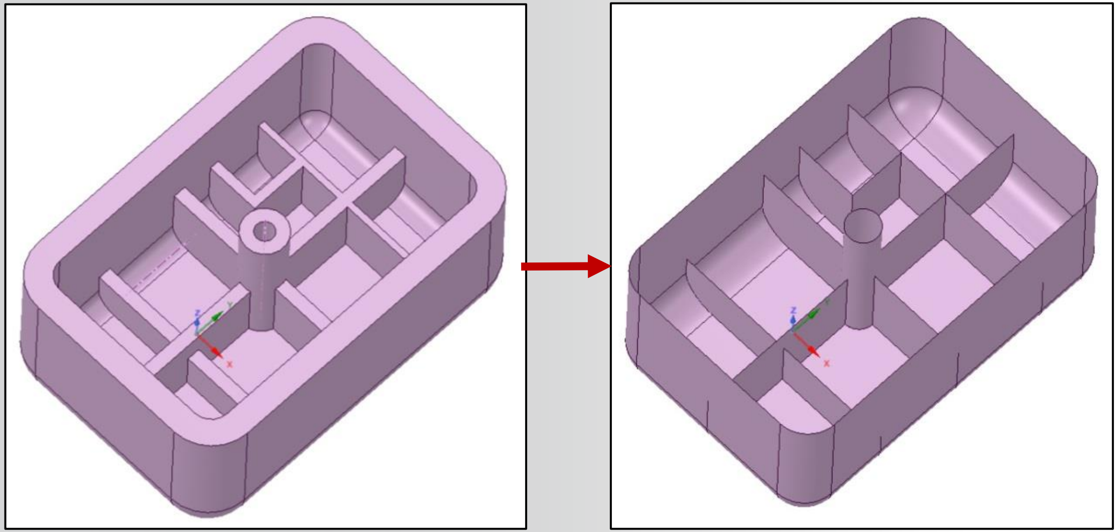
- **Transfer to Workbench**
  - **Expanding the structure tree shows all of the spot welds under the connections. Each pair of points will create its own spot weld.**
  - **Note: These are some limitations when sending spot welds to Workbench**
    - **Only points with mates can be used for simulation**
    - **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.**

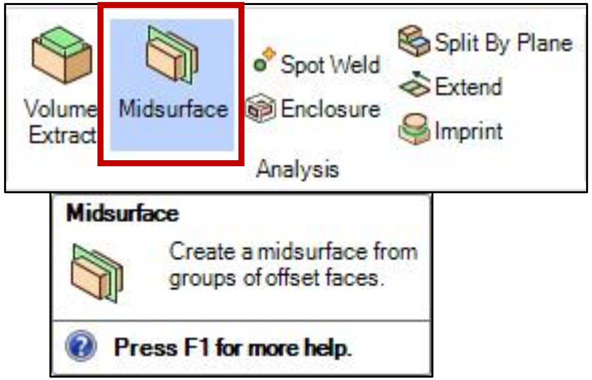




Midsurface

- **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**

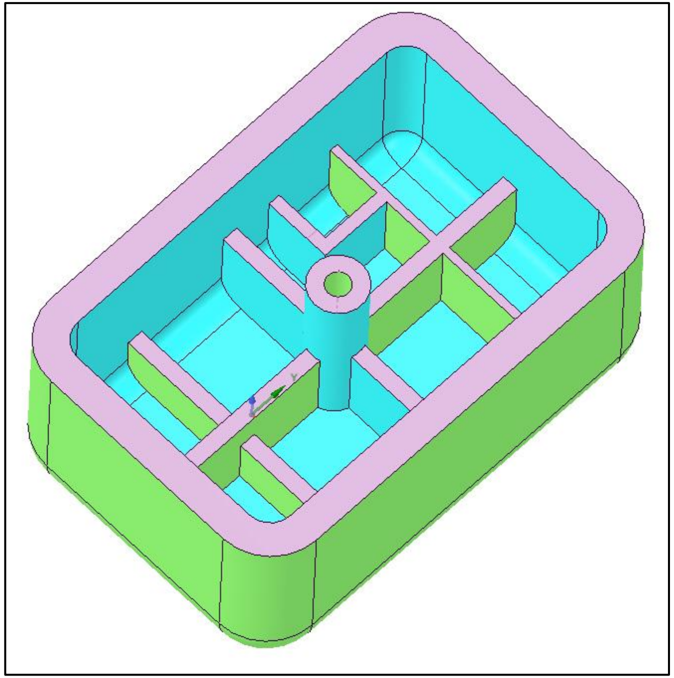




Midsurface

- **Midsurface Tool Guide**

	The Select Faces 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 Add/Remove Faces tool guide allows you to select additional faces to offset or remove detected face pairs from the selection.
	The Swap Sides 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 Complete tool guide creates the midsurface faces.

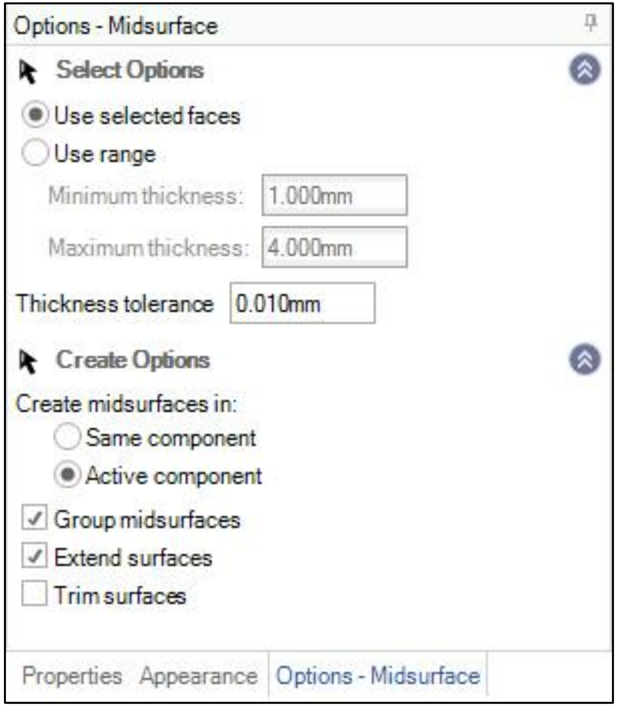


- **Note: Selected face pairs are highlighted in blue and green color where blue color indicates the positive normal side of the midsurface**

Midsurface

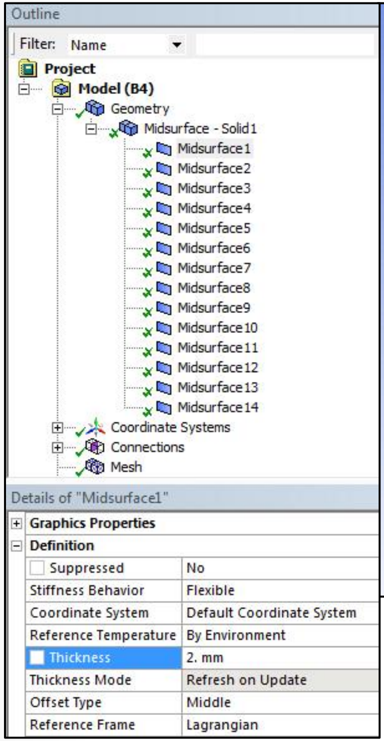
- **Midsurface Options**

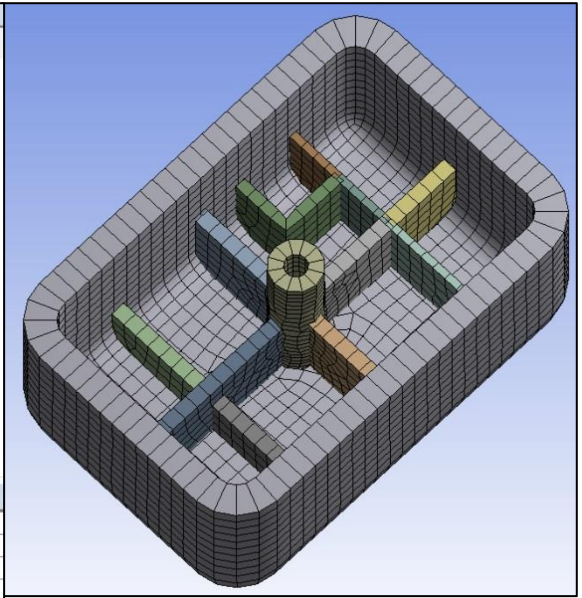
Use selected faces	Select this option to create midsurfaces for only the faces you select.
Use range	Select this option to create midsurfaces on all faces in the specified thickness range.
Thickness tolerance	Change the value of this option to detect offset spline faces with an offset value within the tolerance amount.
Create midsurfaces in	Select Same component to create the midsurfaces in the same component as the part you selected for midsurfacing. Select Active component to create the midsurfaces in the active component.
Group midsurfaces	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).
Extend Surfaces	Automatically extend surfaces (On by default)
Trim Surfaces	Automatically trim the midsurface to the extent of the original body



Midsurface

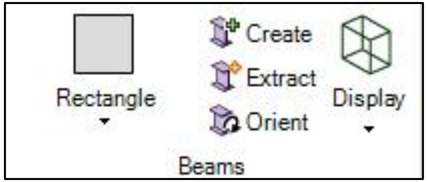
- **Transfer to Workbench**
  - **Only Visible bodies in SpaceClaim are transferred to Workbench.**
  - **Thickness of the midsurfaces created in SpaceClaim is also automatically transferred to Workbench on import.**

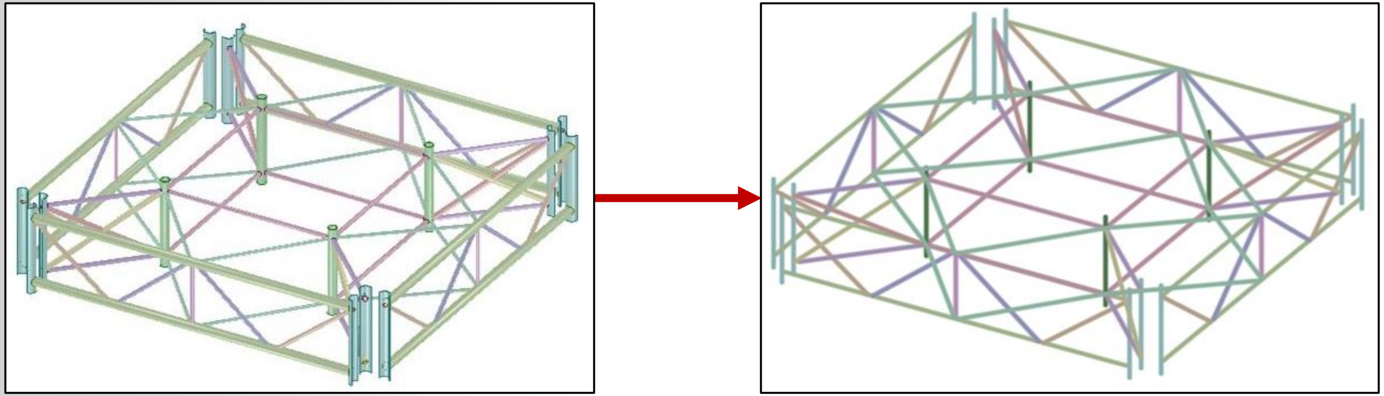




Beams

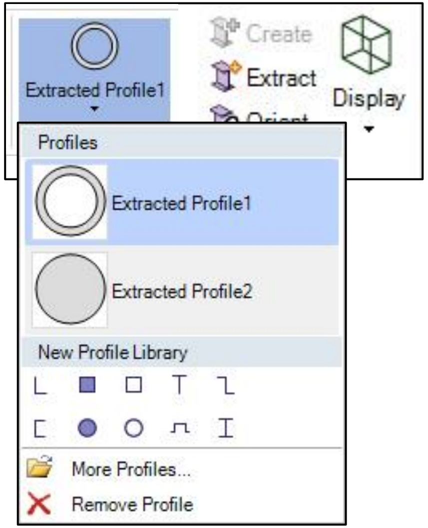
- **Beams Group**
  - **The Beams Group on the Prepare ribbon has the following tools which can be used for creation, extraction and modification of the beams**
    - **Profiles**
    - **Create**
    - **Extract**
    - **Orient**
    - **Display**

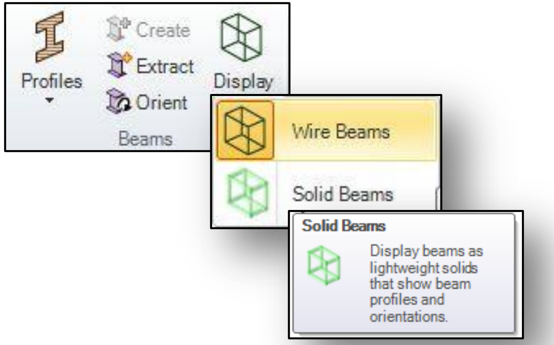




Beams

- **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.

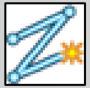
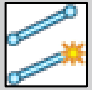


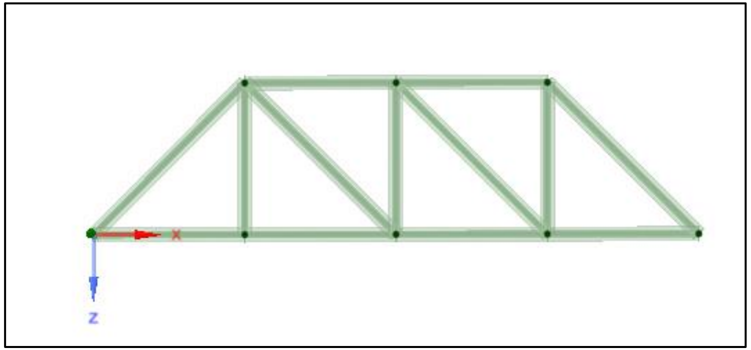


Beams

- **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**

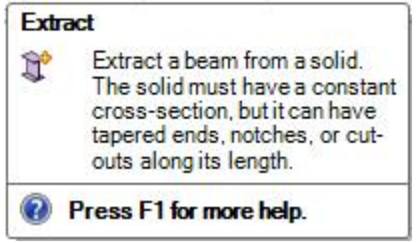
- **Tool Guide**

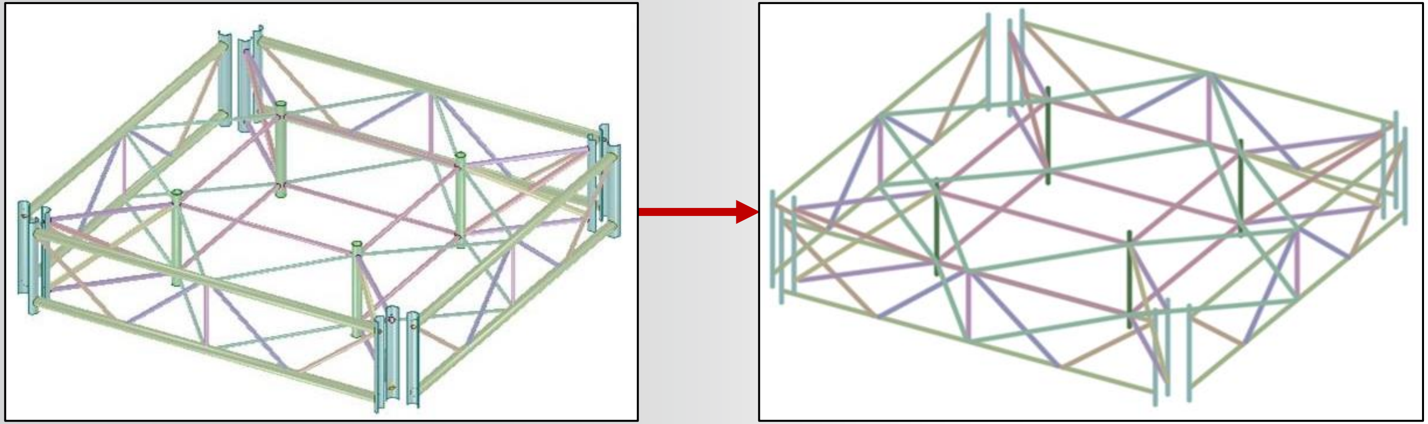
	Use the Select Point Chain tool guide to create a beam along an edge or a series of points that you select.
	Use the Select Point Pairs tool guide to create a beam between two points.

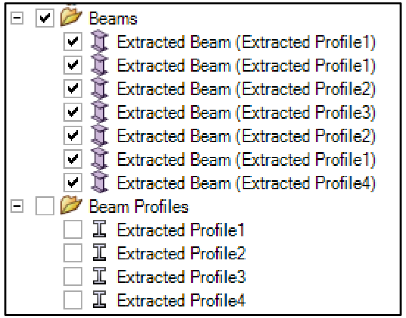


Beams

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

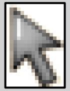


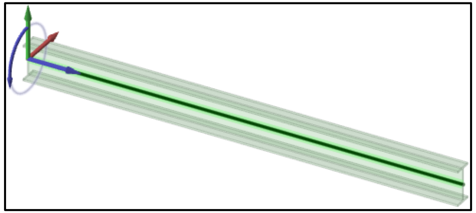




Beams

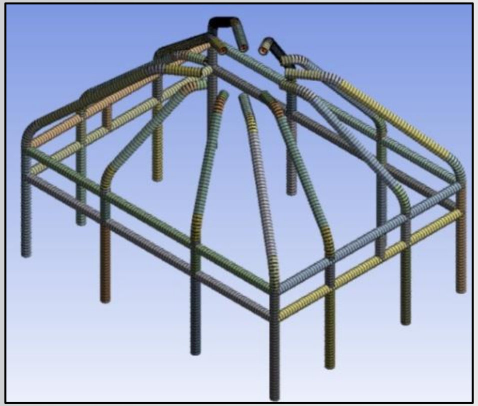
- **Orient**
  - **This tool is used to change the direction of a beam, rotate it around its anchor point, and offset it from its anchor point.**
  - **Orient tool works similar to move tool for beam profiles**
  - **Tool Guide for Orient tool**

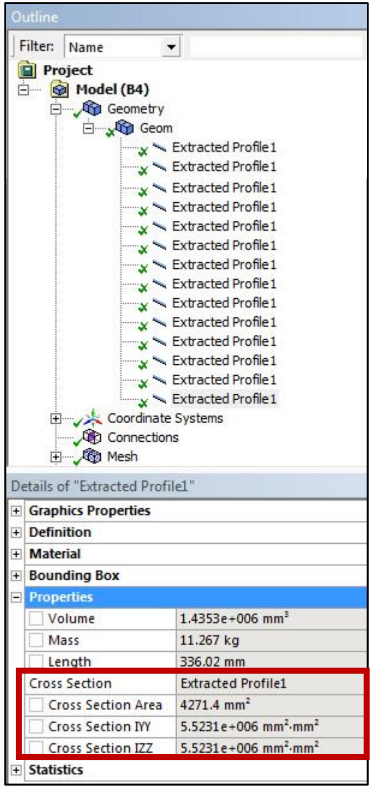
	The Select tool guide is active by default. This tool guide allows you to select the beam object that you want to reorient.
	The Orient to Object tool guide allows you to select a face, edge, or axis and orient the beam in that direction.



ANSYS Beams

- **Transfer to Workbench**
  - **Line bodies should be turned ON 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**



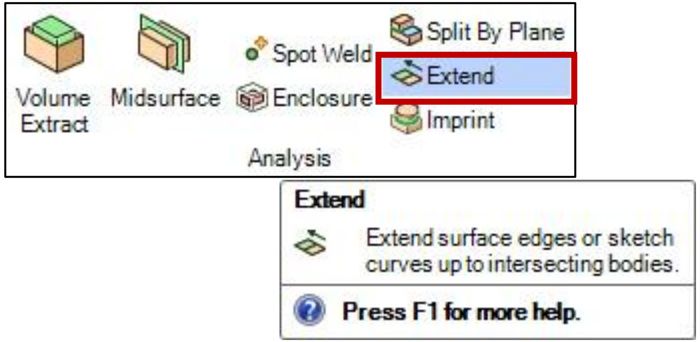


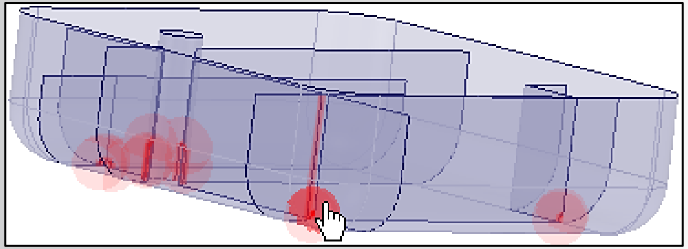
Properties of Schematic A2: Geometry

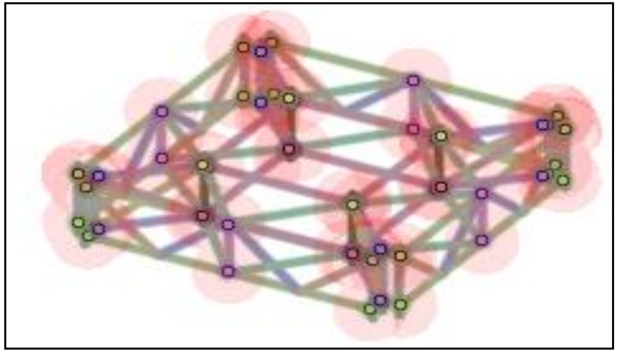
	A	B
1	Property	Value
2	+ General	
5	+ Notes	
7	+ Used Licenses	
9	+ Geometry Source	
12	− Basic Geometry Options	
13	Solid Bodies	☑
14	Surface Bodies	☑
15	Line Bodies	☑
16	Parameters	☑
17	Parameter Key	DS
18	Attributes	☐
19	Named Selections	☐
20	Material Properties	☐
21	+ Advanced Geometry Options	
33	+ SpaceClaim Geometry Options	

Extend

- **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



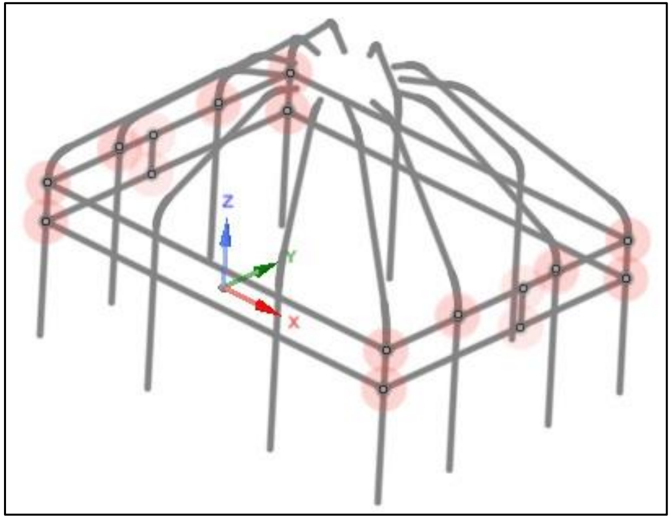




Extend

- **Tool Guide**

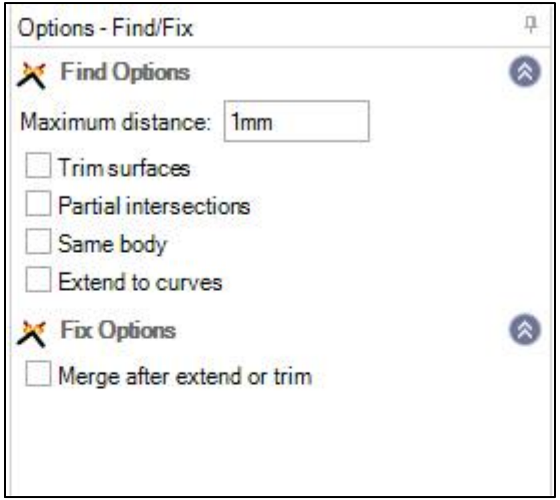
	The Select Problem tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
	The Select Geometry tool guide allows you to select faces that were not automatically found. Hold Ctrl to select multiple objects or box select in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



Extend

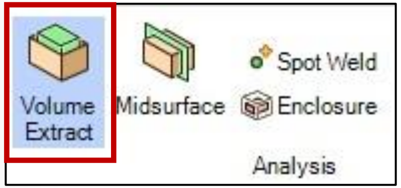
- **Options**

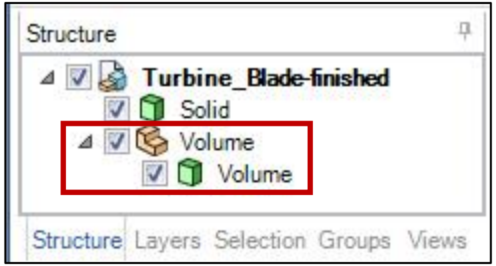
Maximum distance	The maximum distance between parts. The tool will search for adjacent faces again if you change this value.
Trim surfaces	Controls whether or not surfaces may be trimmed in addition to extended.
Partial intersections	Controls whether or not faces that partially intersect are detected.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

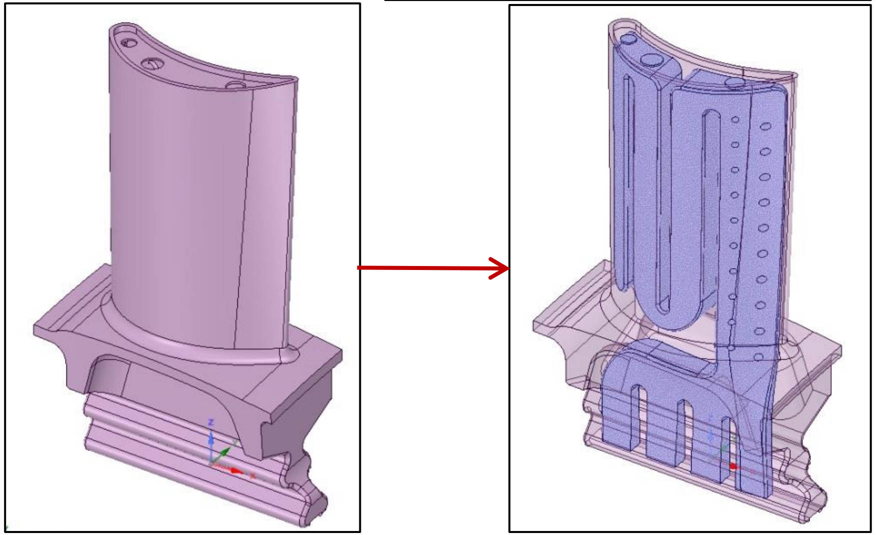


Volume Extraction

- **Volume Extraction**
  - **This tool is used to create a fluid volume based on the volume enclosed by a single solid body or set of solid bodies (including meshes)**
  - **A solid named *Volume* is created in the Structure tree Display in the Structure panel that shows you each of the objects in your design. You can expand or collapse the nodes of the tree to view the objects. You can rename objects, create, modify, replace, and delete objects, as well as work with components and the bodies used to generate the volume are temporarily transparent when the volume is created.**

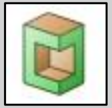
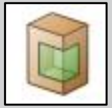






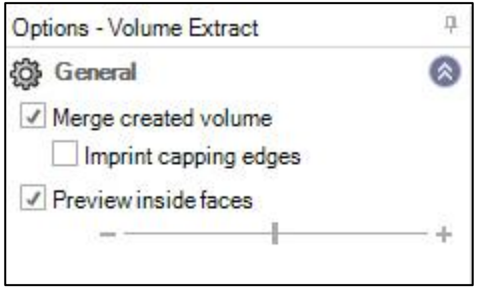
Volume Extraction

- **Tool Guide**

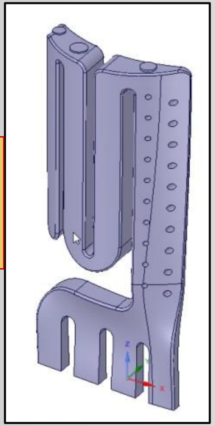
	The Select Edges tool guide selects edge loops that enclose a region. Edges are used to stop the propagation of face selection, emanating face-wise outward, starting at the selected seeds.
	The Select Faces tool guide activates by default when you select the Volume Extract tool, and selects faces whose edges seal an enclosed region. This is a shortcut for selecting all the edges that are detected in a face. You can Ctrl+ Select multiple seed faces, and then choose to click on a different tool guide.
	The Select Cap Faces tool guide selects optional capping faces. This is important when an internal edge loop is either not simply fillable, or when you want some non-standard fill geometry to be created.
	The Select Seed Face tool guide selects a face that lies within the volume you want to enclose. If this is not chosen, then SpaceClaim chooses an arbitrary face to start from, and test whether any bounded volumes are created. If they are not, another face will automatically be selected and the algorithm will re-start. Select a face here to save this iteration time.
	The Complete tool guide creates the volume solid based on the edges and seed face you select.

Volume Extraction

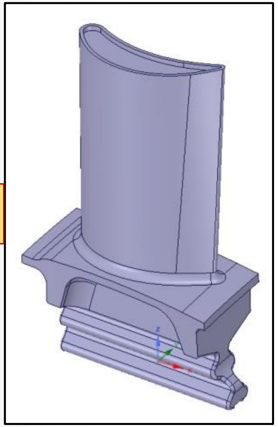
- **Options**
  - **Merge Create Volumes: Merge the Created Volume into outer shell to effectively fill in the inner cavity**
    - **Imprint Capping Edges: Leave the capping faces intact to see the location of inlet and outlets**
  - **Preview inside faces: Use the scroll bar to preview selection of inner faces and detect leaks**



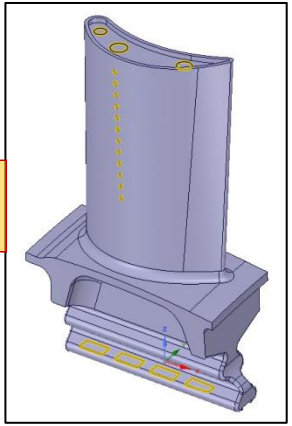
No merge, only Fluid volume from Cavity is extracted



Merge on

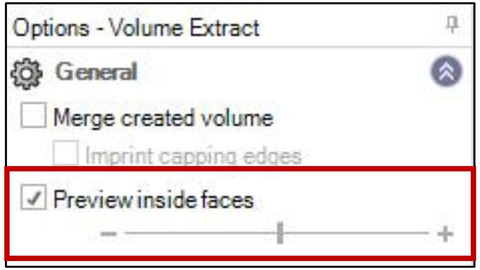


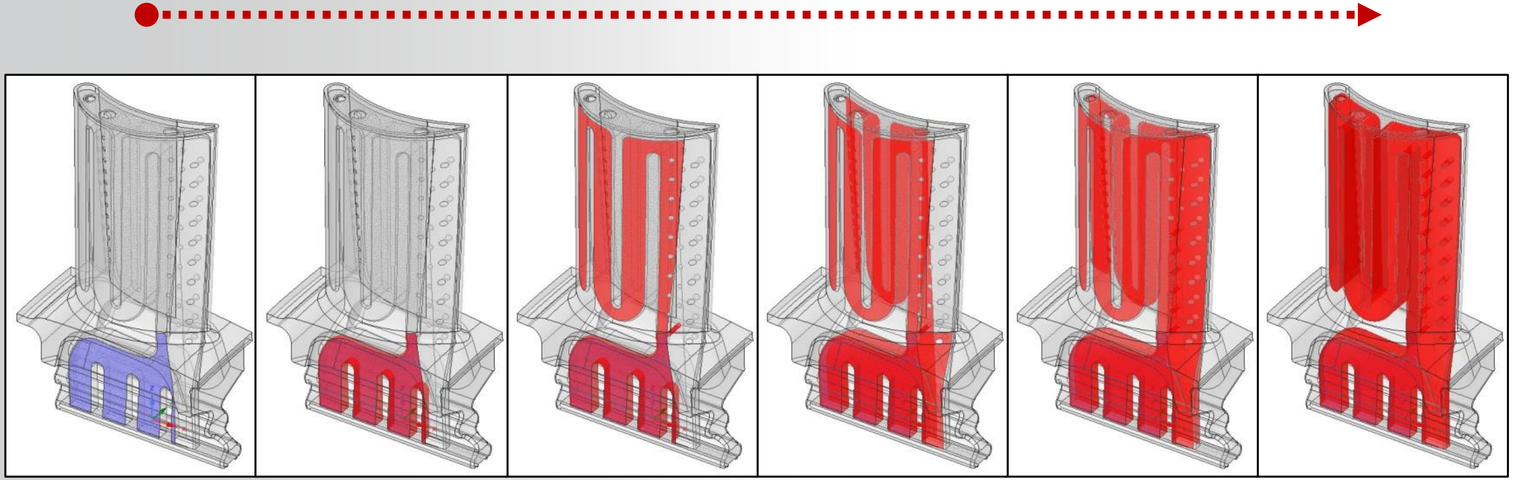
Merge and Imprints



Volume Extraction

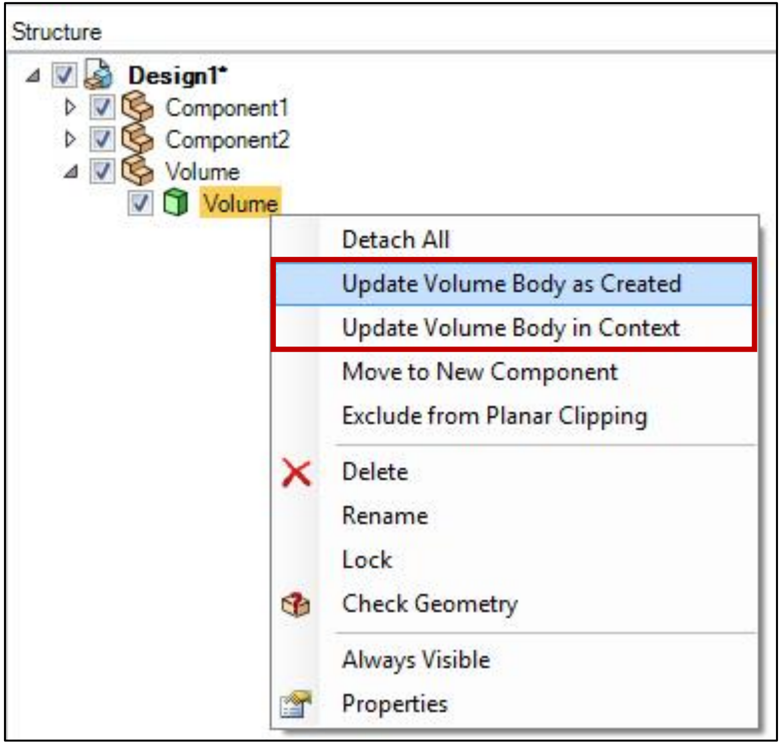
- **Options: Preview inside faces**
  - **After proper selection of Faces/edges defining inlets and outlets of the cavity and seed faces, the Preview inside faces tool gets activated.**
  - **You can slide the slider of the tool to see “wetted” faces and check if there are any leaks in the selected volumes**





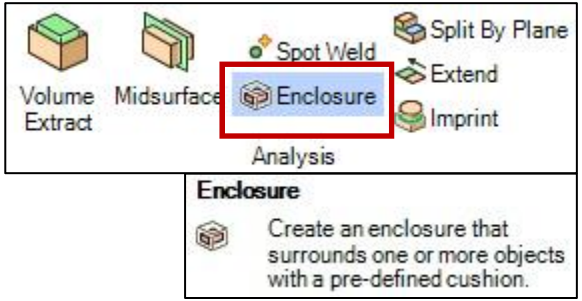
Volume Extraction

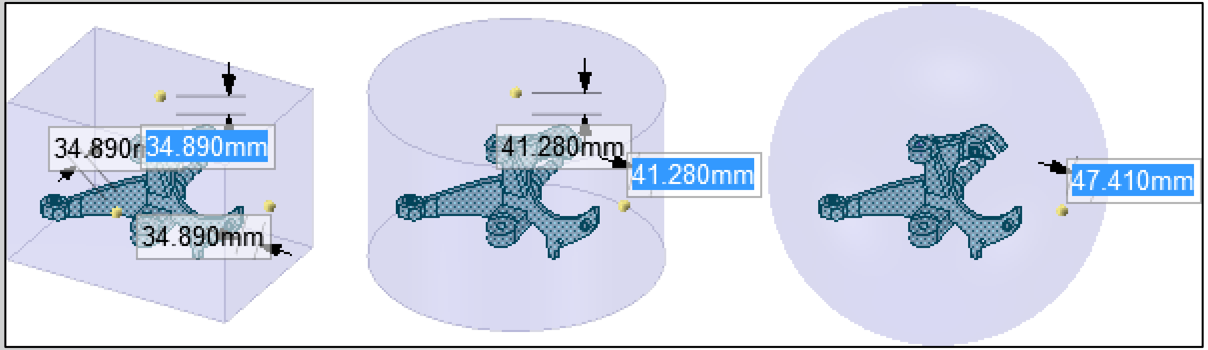
- Updating extracted volume after modifications to Parent solid(s)
  - Right-click the volume part (named Volume by default) in the Structure tree. Select one of the following commands:
    - Update Volume Body as Created updates the volume body based on objects that were visible when the volume was created (regardless of their current visibility). Use this option for simple regeneration.
    - Update Volume Body in Context updates the volume body based on objects that are currently visible in the design area. Use this option to remove bodies from the volume calculation.
  - Note: If parent solids are parametric, and they are modified by updating parameters from Workbench, then you must manually update the fluid volume in SpaceClaim



Enclosure

- **Enclosure**
  - **In SpaceClaim, Enclosure tool is available in Analysis group on Prepare tab**
  - **An enclosure is a solid around a body or bodies that has a cushion around the enclosed solid(s). The enclosure can be a box, cylinder, or sphere, as shown below.**
  - **Enclosures are used by analysis tools to simulate fluid surrounding a solid object**
    - e.g. External aero simulation





Enclosure

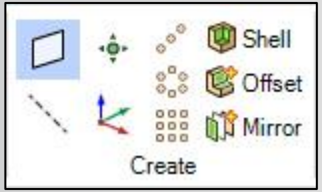
- **Tool guides and Options**

	Use the Select Bodies tool guide to select the bodies that will be enclosed.
	Use the Set Orientation tool guide to change the orientation of the enclosure relative to your design or the axis of a coordinate system.
	Use the Custom Shape tool guide to select a solid to use as the custom shape when you set the enclosure type to **Custom** in the Options panel.
	Click the Complete tool guide when you are finished.

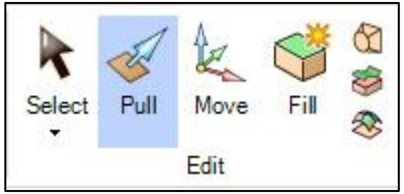
Default cushion	This is a percentage of the minimum enclosure size, and determines the distance between the enclosed object(s) and the closest point of the enclosure to the objects. You can change adjust the distances by typing in the fields in the Design window.
Enclosure type	Select an enclosure shape from the list. If you select Custom shape, you must use the Custom Shape tool guide to select a solid to use as the enclosure shape.
Symmetric dimensions	Forces the dimensions to remain symmetric. Deselect this option if you want to enter values for dimensions and you don't want the opposite dimensions changed.

Volume Decomposition

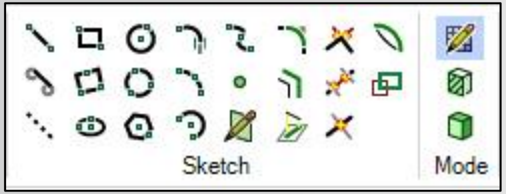
- **Volume decomposition is needed to get the desired mesh type (for ex: hexahedrons) or improve the quality of mesh**
- **Following tools from Design ribbon and their combination can be used to decompose the large volume into small volumes**
  - **Plane: to create cutting planes**



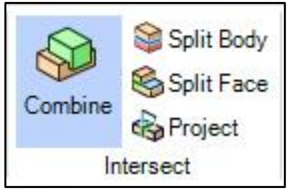
  - **Pull: to Extrude or revolve cutting edges into cutting surfaces**

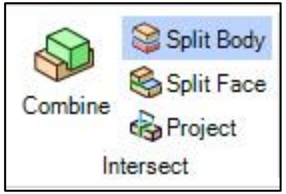


  - **Sketch: to draw cutting edges**



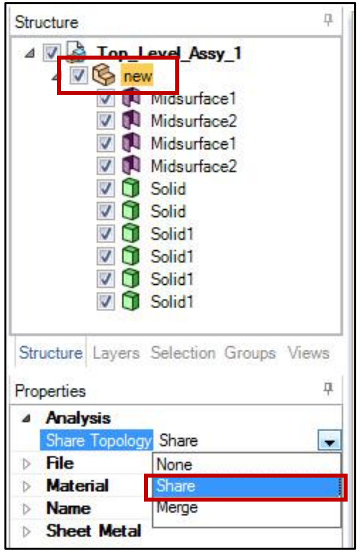
  - **Combine and Split Body: to cut Volumes**

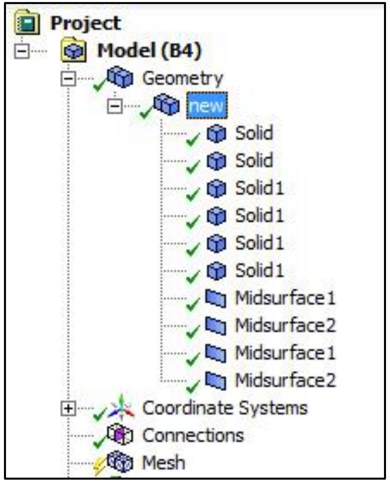


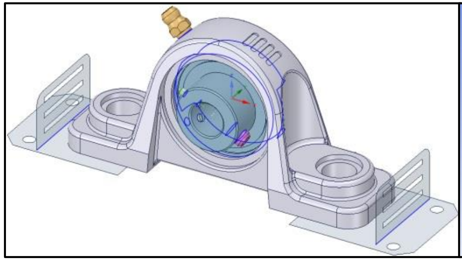


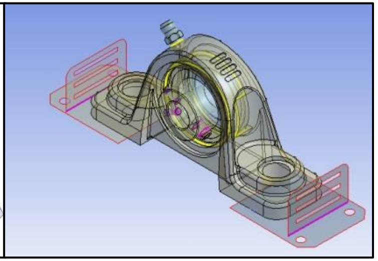
Shared Topology

- **Shared Topology**
  - SpaceClaim can share topology (face, edge, and vertex connections) between bodies and surfaces in contact that are transferred to Workbench
  - Shared topology is the only way to achieve a conformal mesh where bodies meet
  - Shared topology also applies to volume and surface bodies that are enclosed within other volume or surface bodies. This situation is common in analyses involving fluid flow
  - Shared Topology is passed from SpaceClaim to Workbench when the shared topology property for the parent Component is set to 'Share'
  - Solids or Surfaces under the same component will be treated as a multi-body part and will share topology when the property is set



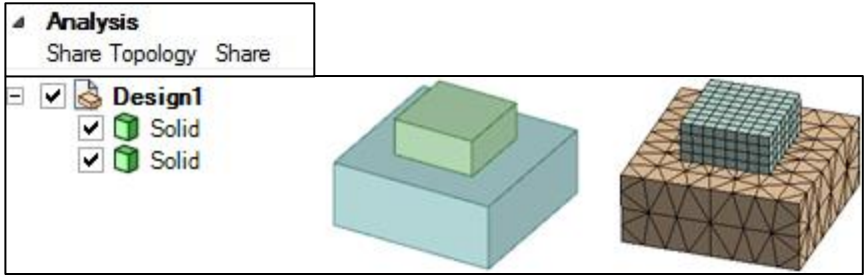


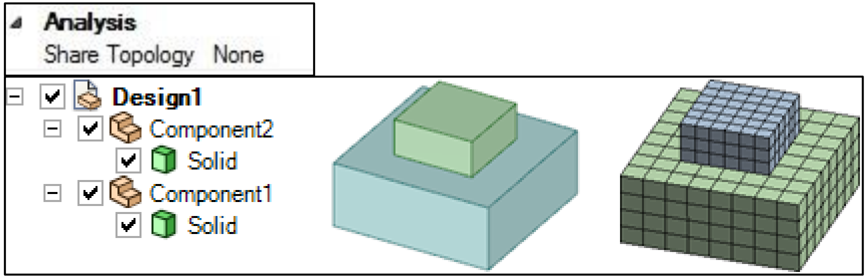


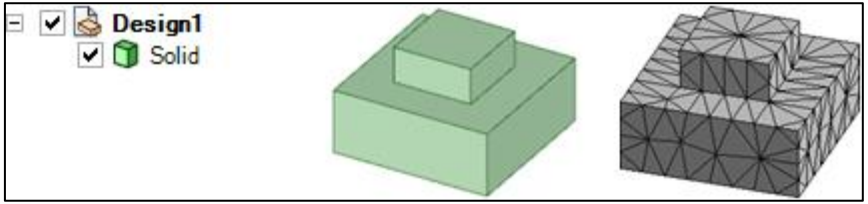


Shared Topology

- **Shared Topology Examples**
  - Mesher meshes two blocks with shared topology. The blue block will gain a new imprinted face which will be shared between the green and blue blocks.
  - Mesher creates separate meshes for two parts because they are in different components and the root part has Shared Topology set to None.
  - The mesh for shared topology is not the same as merged geometry. Here the boxes are merged, and you can see that the mesh is different than it is for two bodies with shared topology.
  - Note: If a solid body is taken out of parent component which is set to shared topology, it will have no imprints and no shared faces/edges with the previous parent part

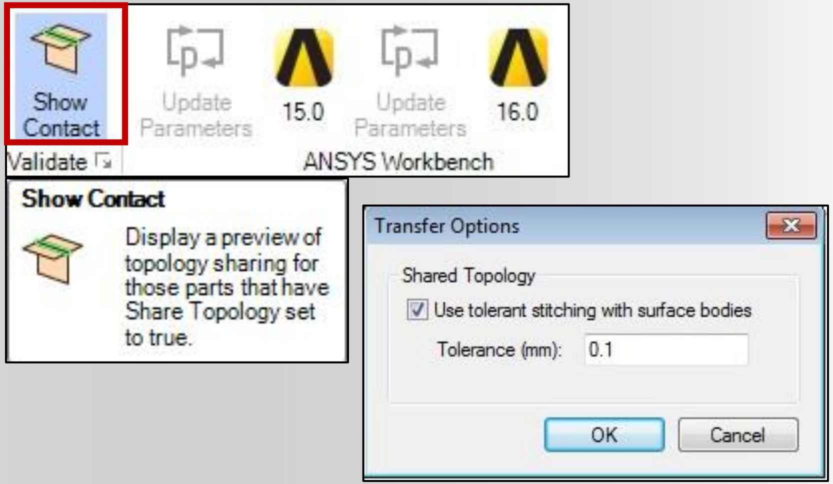


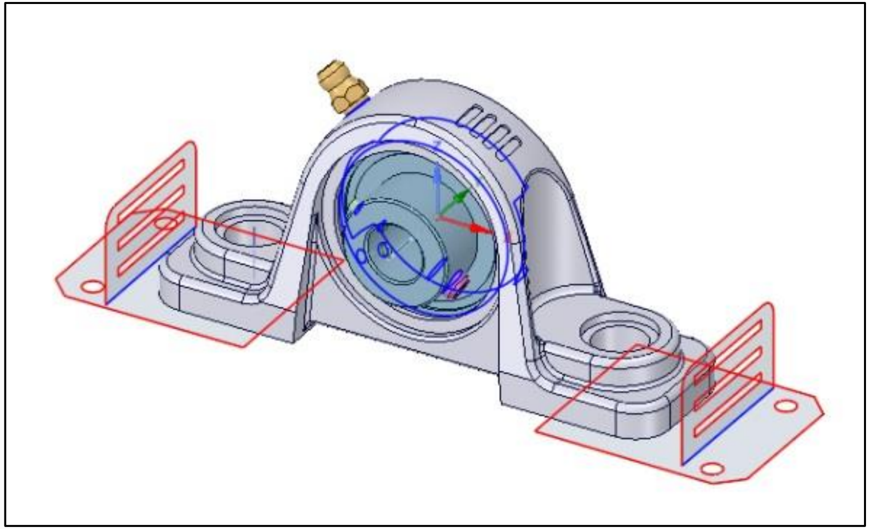




Show Contact

- **Show contact**
  - **This tool in the CAE group on the Prepare tab displays a preview of topology sharing. ‘Show Contact’ lets you see – before sending to Workbench – exactly what topology would be shared.**
  - **Allows you to modify/correct the geometry so that it is transferred to Workbench as intended**

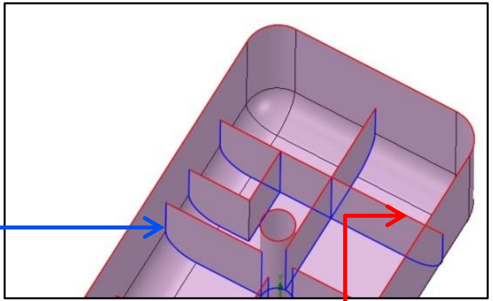


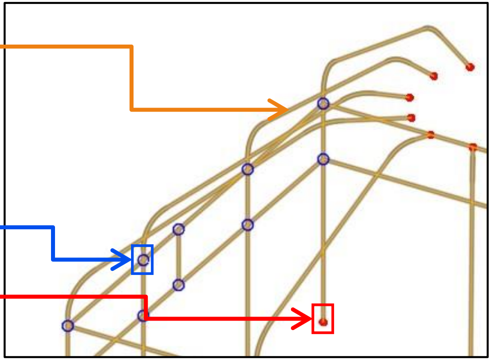


Show Contact

- **Show Contact - Options**

Show Contact settings in the Options Panel	Display color	Description
Edge Options		Edge options control the display of connected and disconnected paths (i.e. edges, sketch curves, Beam paths).
Show Edge Junctions	**Blue**	Shared edges of solids or surfaces and Beams
Show Laminar Edges	**Red**	Edges of surface bodies that are not connected to anything
Show Free Beams	**Orange**	Beams that are not entirely connected (i.e. coincident to an edge or embedded within a face).
Vertex Options		Vertex options control the display of connected endpoints (Beam endpoints and intersections).
Show Beam Junctions	**Blue**	Shared points on Beams (i.e. shared endpoints or interior intersections).
Show Beam Ends	**Red**	Non-shared endpoints of Beams.

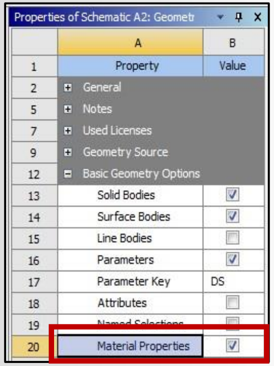


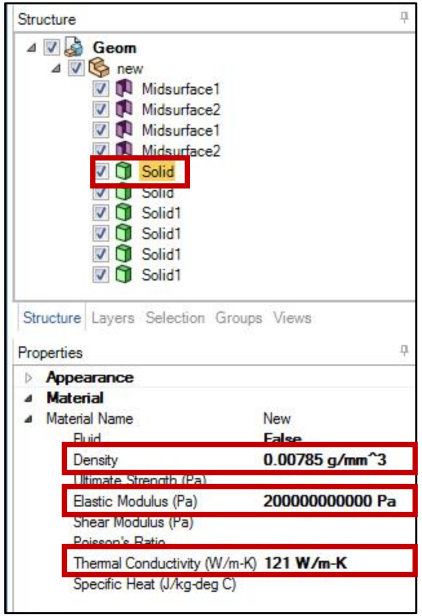


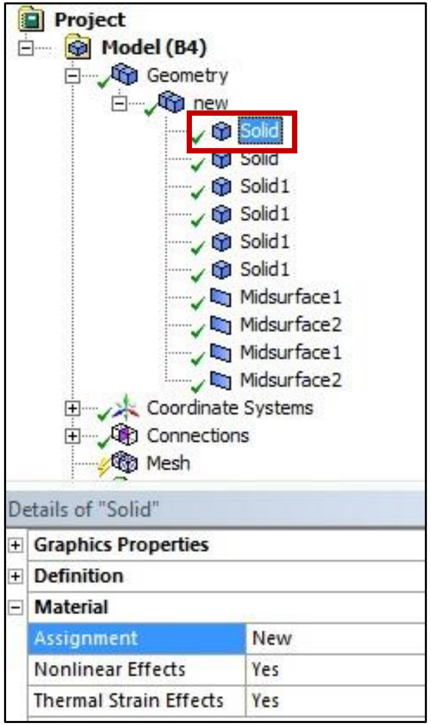
Defining Material

- Material properties can be passed to Workbench as long as the property option is set.

- Following properties must be defined for the material to transfer from SpaceClaim to Workbench:
  – Density, Elastic Modulus, Thermal Conductivity
  – Material properties option should be checked in properties of SpaceClaim geometry in workbench project schematic





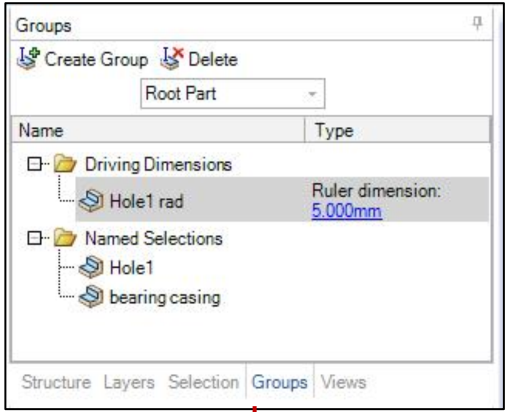


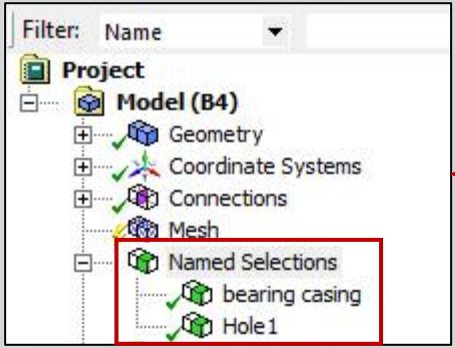
- Material can be defined as new or simply picked from the library

- Note: Even if the material is picked from the library, if it doesn't have one of the properties mentioned above it will not be transferred to Workbench (In such cases standard ANSYS material will be applied i.e. structural steel)

Parameters and Named Selections

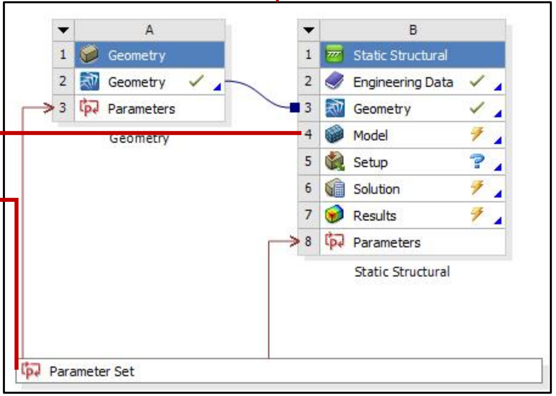
- **Parameters and Named Selections can be transferred to Workbench by creating Groups in SpaceClaim**
- **Driving Dimensions can also be created**
  - **Need to use the Pull or Move menu**
  - **Make dimension appearing on the display window.**
- **Parameters can also be edited and modified within Workbench which will modify the geometry in SpaceClaim**





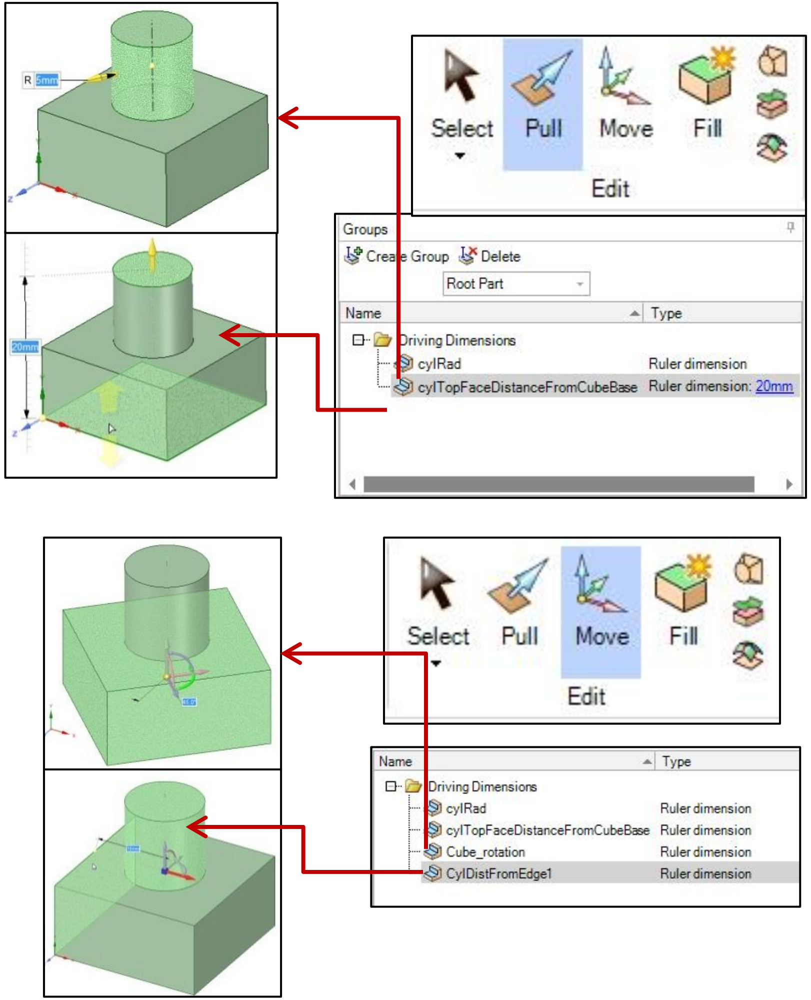
Outline of All Parameters

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	Geometry (A1)			
4	P1	Hole1 rad	5	
*	New input parameter	New name	New expression	
6	Output Parameters			
*	New output parameter		New expression	
8	Charts			



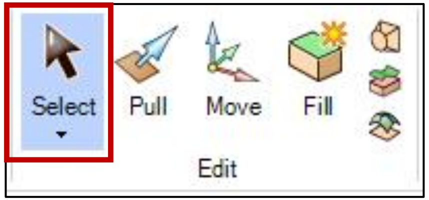
Parameters

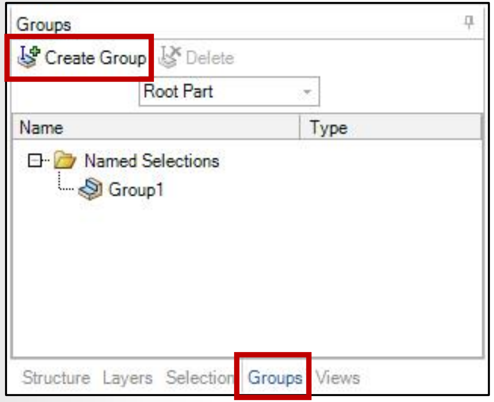
- **Parameters are used for parametric study in WB. To define driving parameters in SpaceClaim:**
  - **You should be in Pull or Move mode**
    - **Pull mode provides access to change in dimension (ex. Hole radius)**
    - **Move mode provides access to change in location and orientation of features**
  - **Select the feature to parameterize (The feature should show dimension). You can also use ruler for dimensions from reference object.**
  - **In the groups tab click on Create Group.**
  - **After this, a parameter group should be created under Driving Dimension Folder for the said dimension.**
  - **Default name (Group#) will be given to Parameter. You can then rename by right clicking on it.**
- **Note: Keep parametric bodies in separate components in order to avoid merging them after parameter update**

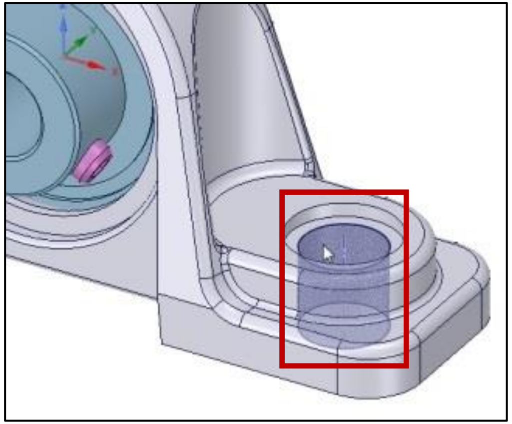


Named Selections

- Named selections are defined to create a base object for mesh methods, or to identify them for applying boundary conditions in Workbench
- Named Selections for edge, faces and bodies are transferred to Workbench
- Procedure to define Named Selection is as follows:
  - Select mode should be active
  - Select required entity/entities (Edge, face or body)
  - In the groups tab click on Create Group
  - Default name (Group#) will be given to Named selection. You can rename by right clicking on it.

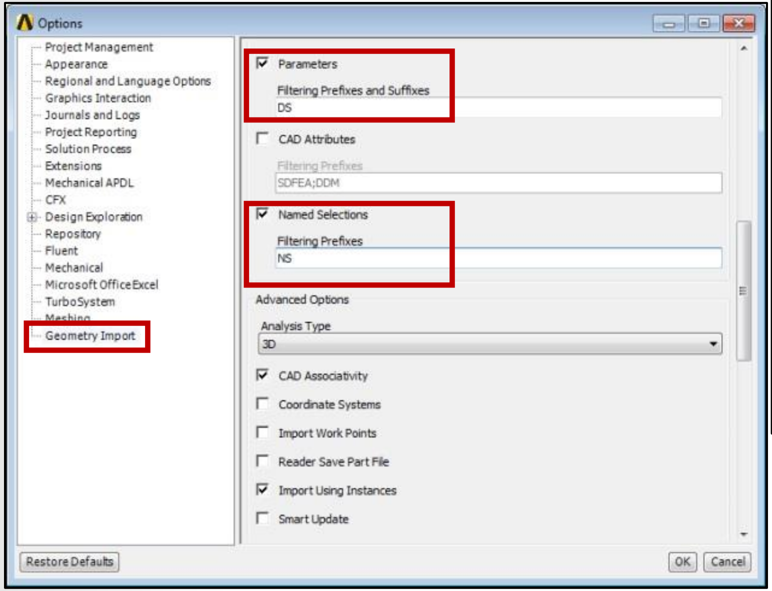


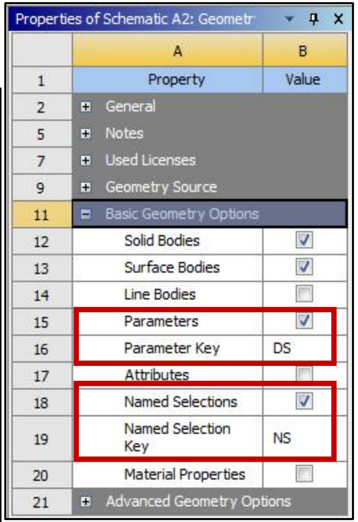




SCDM → Workbench

- **Transfer to Workbench**
  - **Driving Dimensions and Named Selections can be renamed (RMB click) in SpaceClaim**
  - **Make sure Named Selections and Parameters are activated into WB project page**
    - **Tools→ Options → Geometry Import and/or**
    - **Properties for geometry cell**
  - **Filtering prefixes:**
    - **Delete the NS and DS prefixes, or**
    - **Rename required Driving dimensions and Named selections in SpaceClaim with prefix DS and NS respectively**





SCDM → Workbench

- Geometry from SpaceClaim can be sent into Workbench using the following methods
  – Prepare tab → Workbench 16 (Direct link)
    - Opens a Workbench page
    - In the project schematic, link to the appropriate Analysis system by dragging and dropping the system from the toolbox on the SpaceClaim cell

OR

  – Open a Workbench & drag and drop desired analysis system in project schematic
    - Right click on geometry cell
    - Again two options are available
      – **New SpaceClaim Direct modeler**…, then open the saved Spaceclaim file, or
      – **Import Geometry** to open the saved Spaceclaim file