Introduction to Ansys SpaceClaim

# Module 05: SpaceClaim to Workbench

Release 2022 R1

\\Ansys

Learning Outcomes

- How to assign material properties in Ansys SpaceClaim

- What is the procedure to not export a body from SpaceClaim to Workbench

- Defining named selections and parameters in Ansys SpaceClaim

- How to transfer a model from SpaceClaim to Workbench

- How to manage Shared Topology in Ansys SpaceClaim

Overall Process
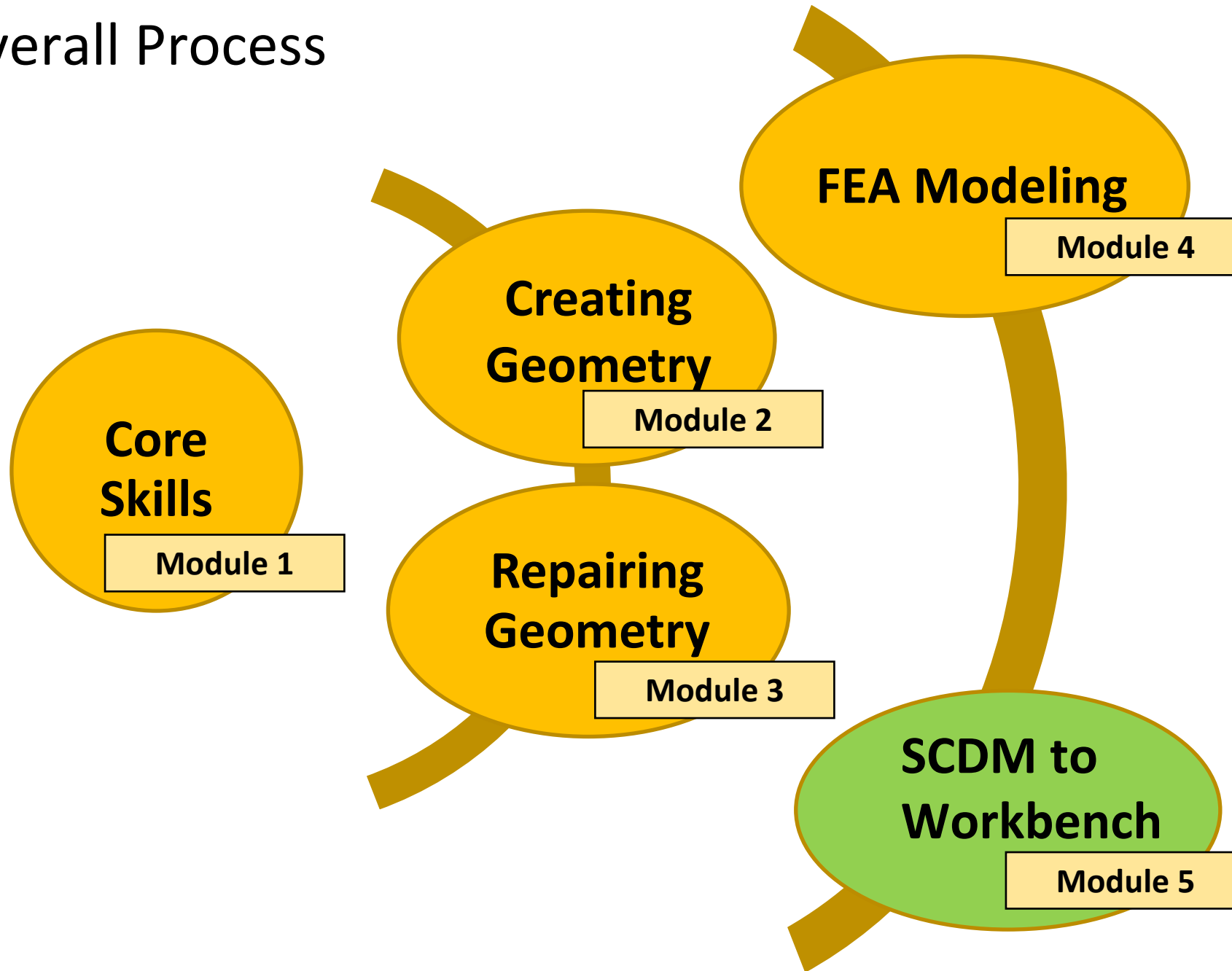
Defining Material

- 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 SCDM to Workbench:
  - Density, Elastic Modulus, Poisson's ratio
  - 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)

Suppress/Activate for Physics

- Suppressed objects are not transferred down Stream to Ansys for analysis

  - Applies to Solid/Surface bodies, components and Beams

  - Applies to multiple selections; objects that do not apply, like Curves, are ignored
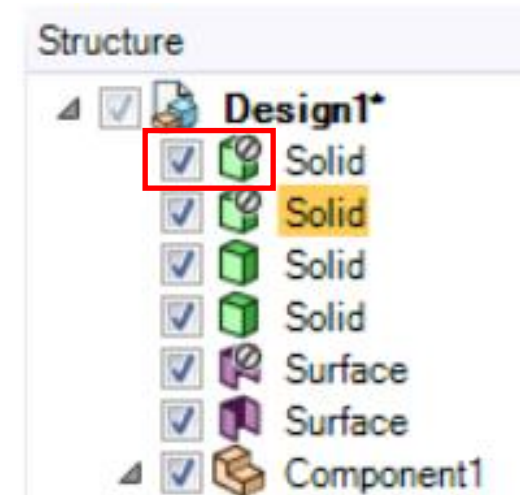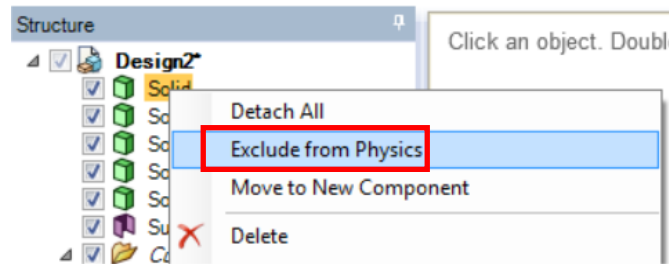
- Objects that are excluded have a universal suppressed symbol

- To re-include an excluded object or objects, use the 'Activate for Physics' option

/Ansys
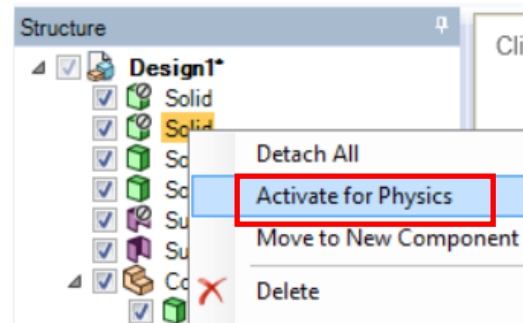
Parameters and Named Selections
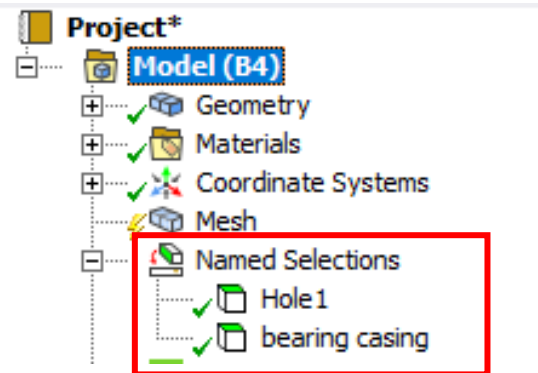
- Parameters and Named Selections can be transferred to Workbench by creating Groups in SpaceClaim

- Driving Dimensions can also be created
  - Need to use the Pull or Move menu or the dimension
  - Make dimension appearing on the display window.

- Parameters can also be edited and modified within Workbench which will modify the geometry in SpaceClaim

Parameters (1)

- Parameters are used for parametric study in WB. To define driving parameters in SpaceClaim:
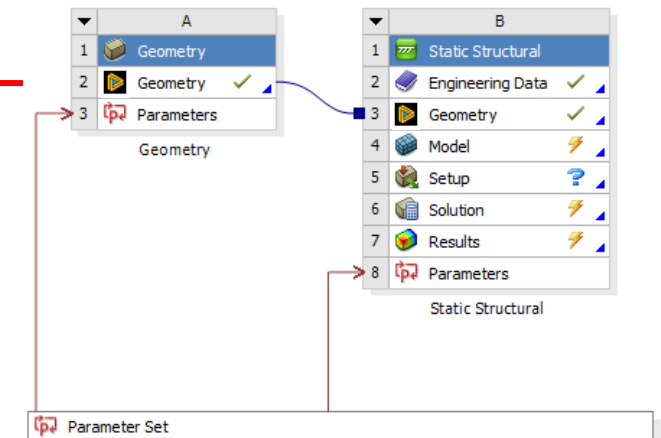
  - You should be in Pull or Move mode or use the dimension
    - Pull mode provides access to change in dimension (ex. Hole radius)
    - Move mode provides access to change in location and orientation of features
  - Select the feature to parameterize (The feature should show dimension). You can also 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
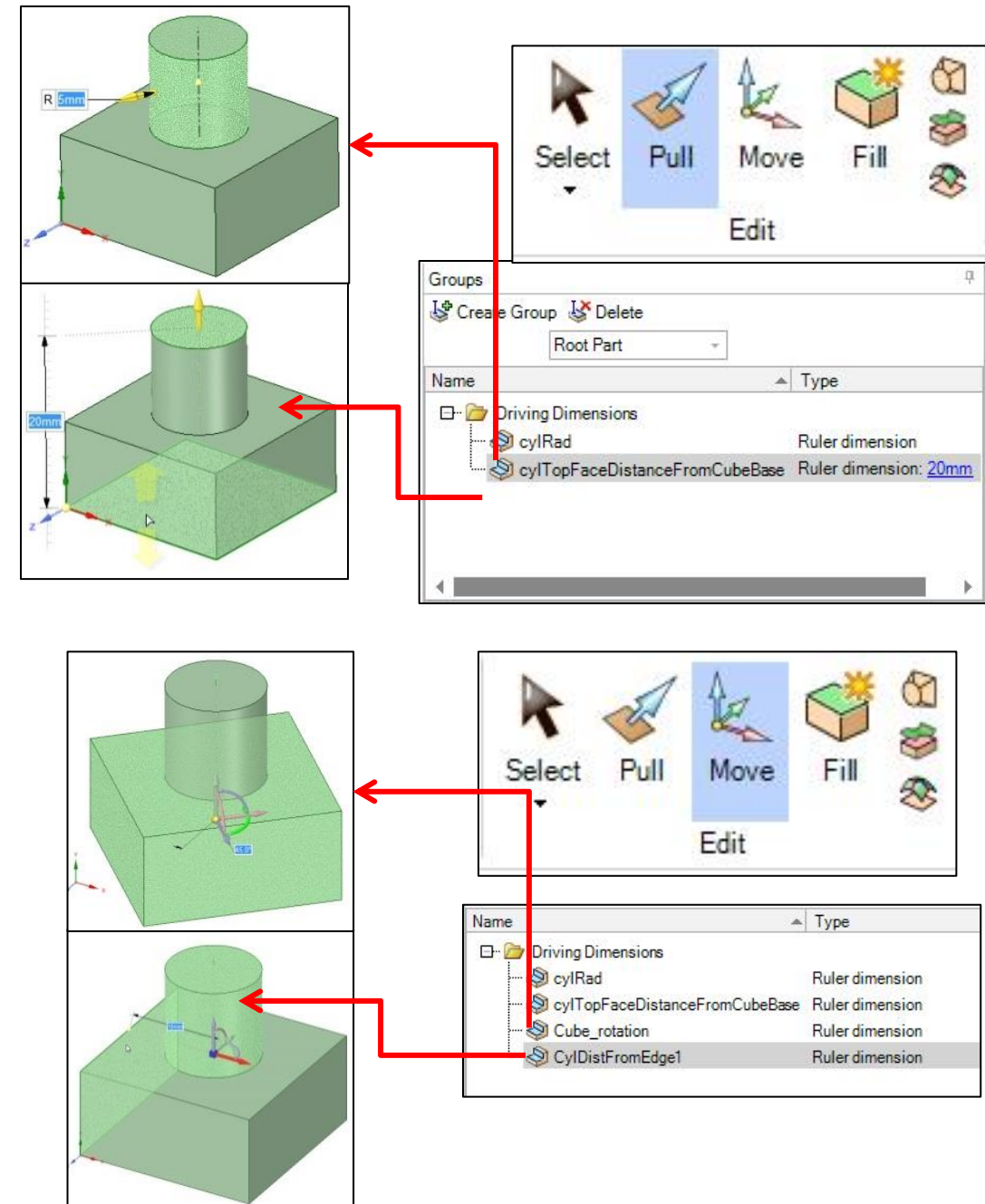
Parameters (2)

- Another way of creating parameters is via the Block Recording through the 'P' button

- It's best to consider that group parameters are intended for direct modeling (where you can have Workbench drive the changes and thus the Workbench parameter manager is best to drive)

- When you need order/history (like in scripting) feature tracking has advantages and the block recording parameters work nicely here and do pass to Workbench
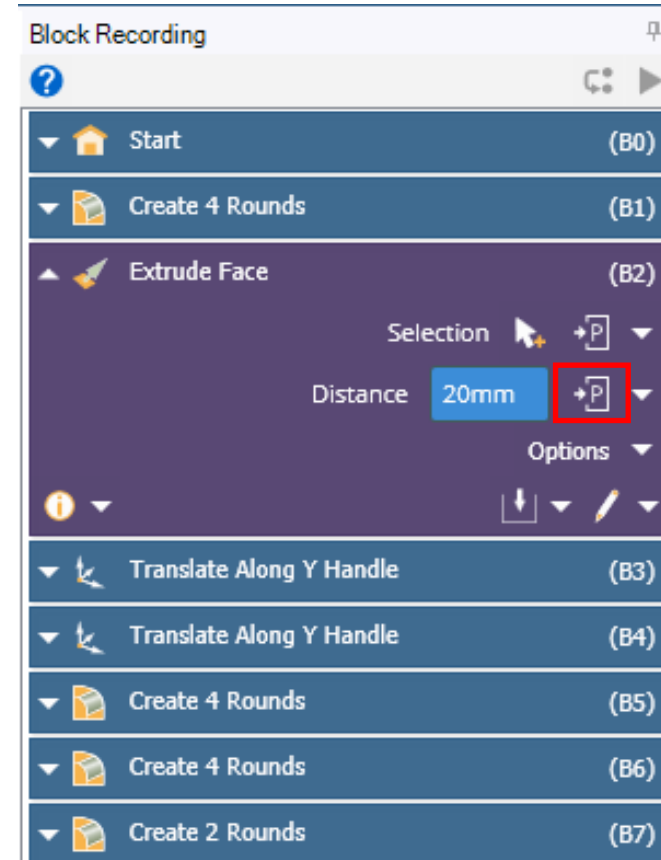
Named Selections

- 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 edges, 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(s), face(s) or body(ies))
  - 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.
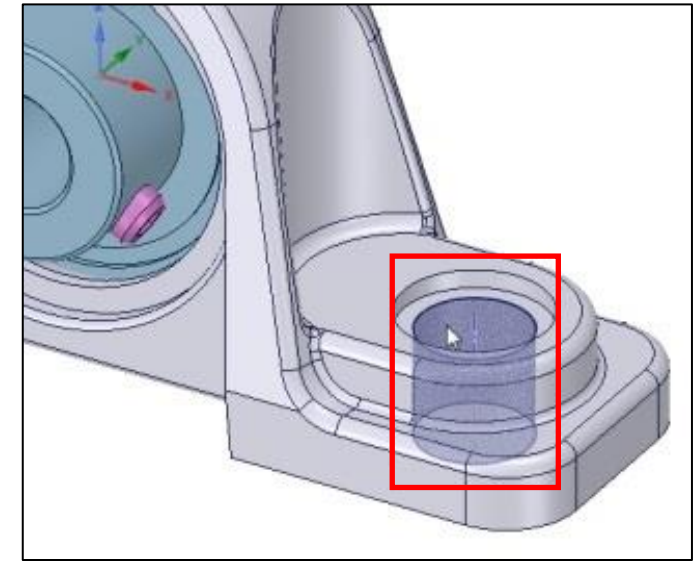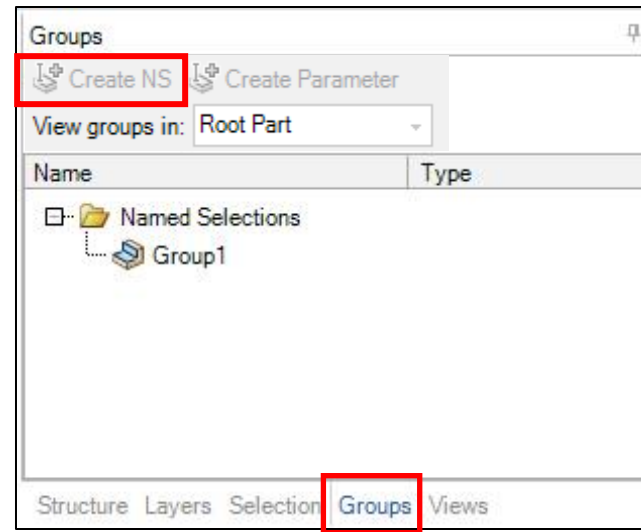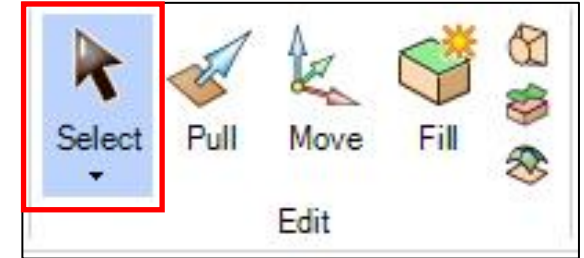
Detect overlapping Named Selections

- Detect Overlapping Selections
  - Goes through NS from top to bottom
  - Compares each NS to every lower NS
  - If an overlap exists, it creates a new NS with the overlapping selection set

- Repair Overlapping Selections
  - Goes through NS from top to bottom
  - If a face exits in a lower NS, it is removed from that NS
 - If no faces exist in a NS, it is removed
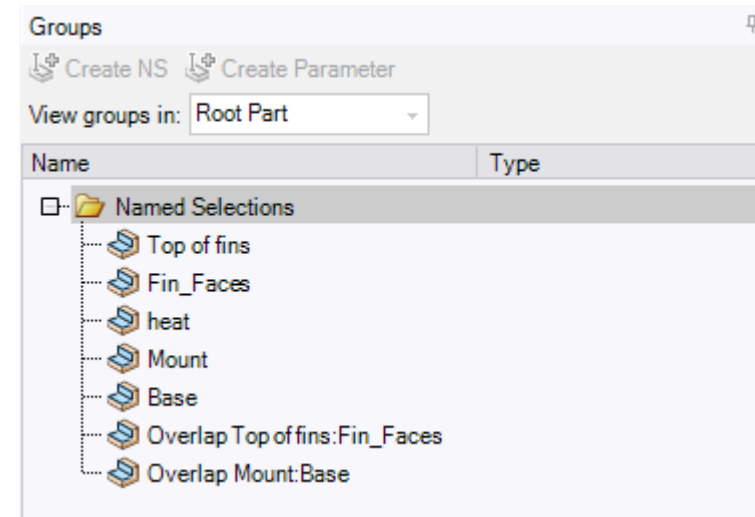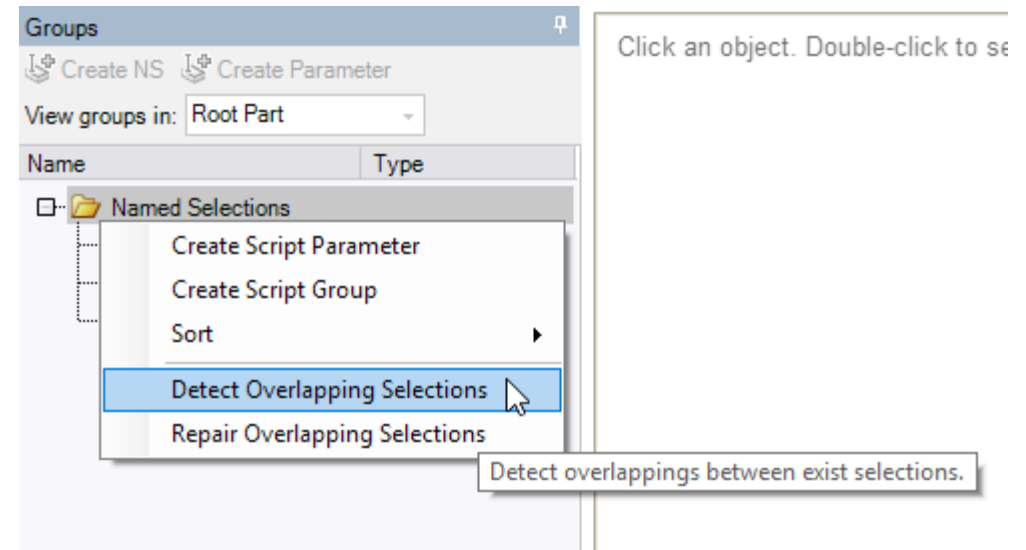
SCDM → Workbench (1)

- ## Transfer to Workbench

  - Driving Dimensions and Named Selections can be renamed (RMB click) in SpaceClaim

  - Make sure Named Selections is activated into WB options (if SpaceClaim is defined as an external CAD):
    SpaceClaim Options / File Options / Workbench

  - Filtering prefixes:
    - Delete the prefixes, or
    - Rename required Named Selections in SpaceClaim with the corresponding prefix
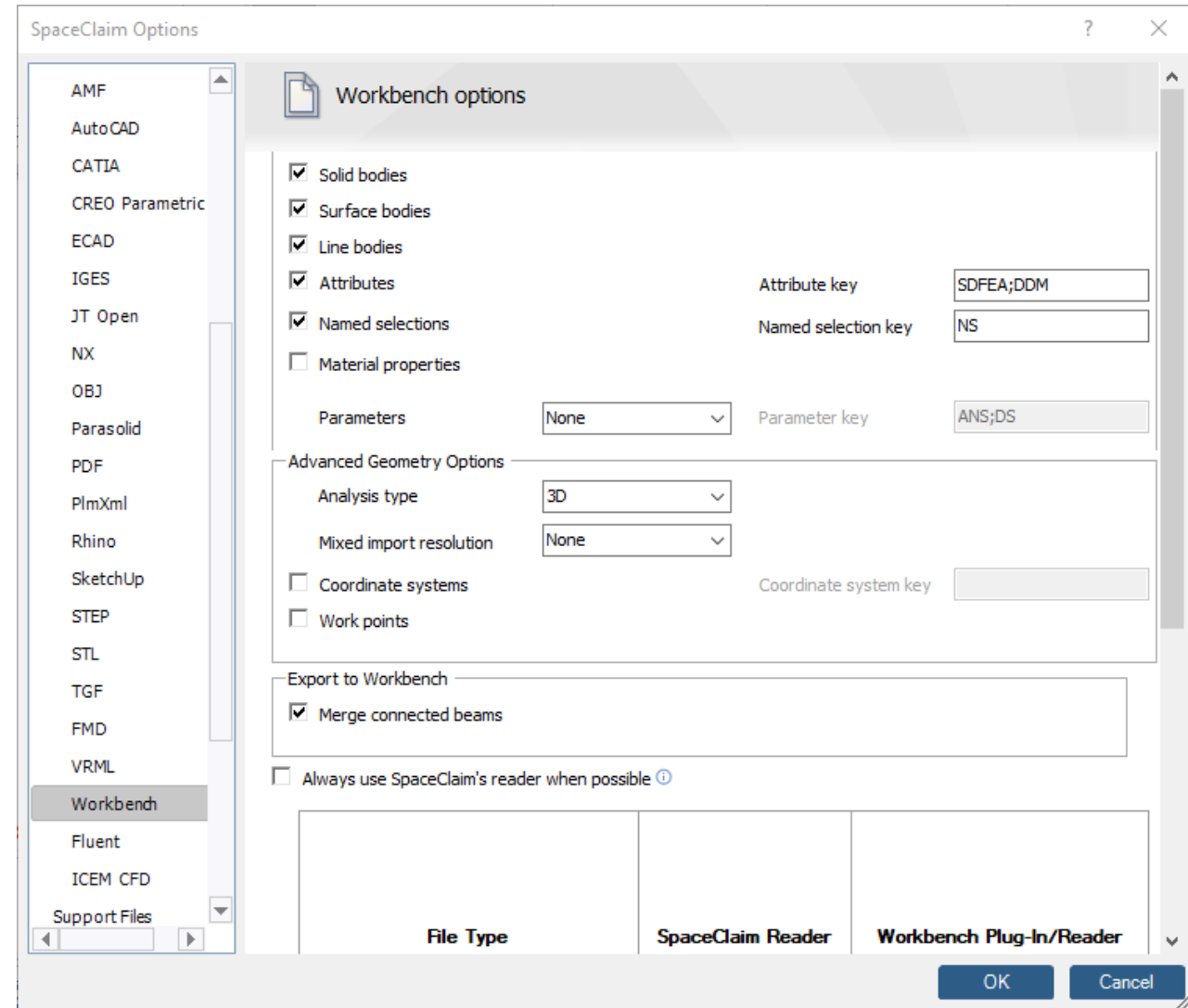
SCDM → Workbench (2)

- Geometry from SpaceClaim can be sent into Workbench using the following methods
  - Workbench→ Workbench 2021 R1 (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 Geometry…, then open the saved Spaceclaim file, or
      - Import Geometry to open the saved SpaceClaim file
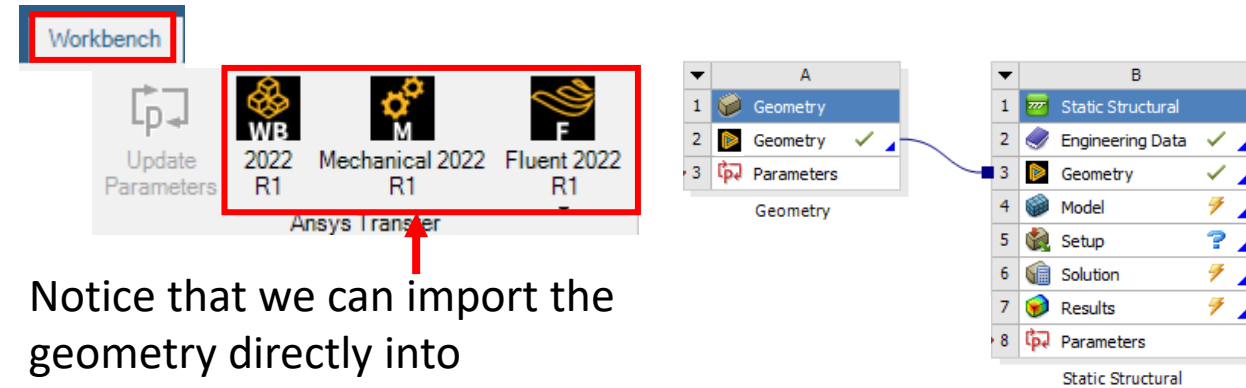
- Notice that we can import the geometry directly into Mechanical or Fluent.

- For Mechanical for instance, a 'Mechanical Model' box will be created in the Workbench project schematic.
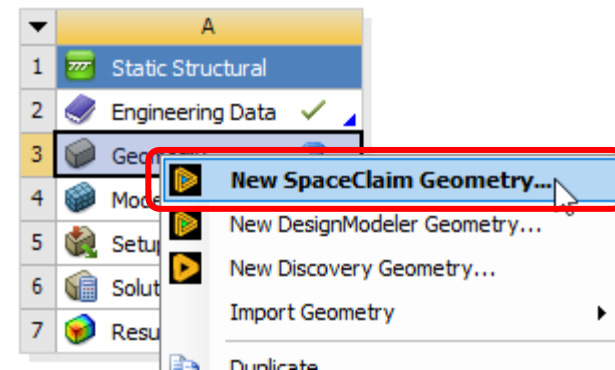
Shared Topology (1)
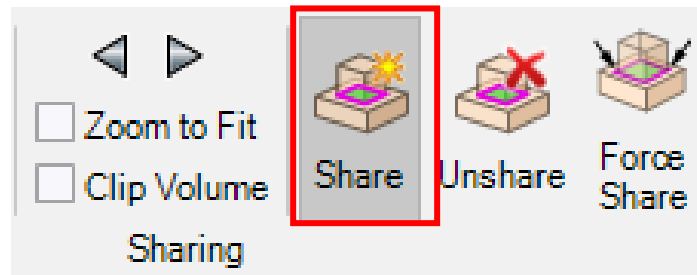
- Shared Topology
  - Since R18.1 Shared topology has been moved into a set of tools in the Workbench tab

  - SharePrep:
    - Will upgrade legacy scdoc files from the previous shared topology to the new one
    - Resolve intersections among bodies before applying shared topology

Note: SharePrep is the only choice initially in the group if a model has been loaded in which shared topology was declared in the assembly structure, but which has not been upgraded to the new shared topology tool
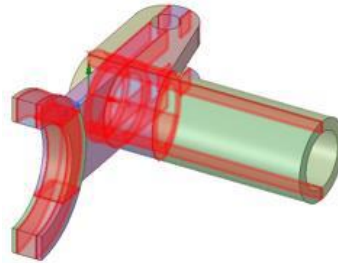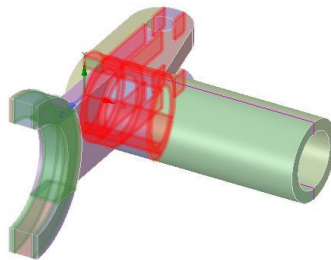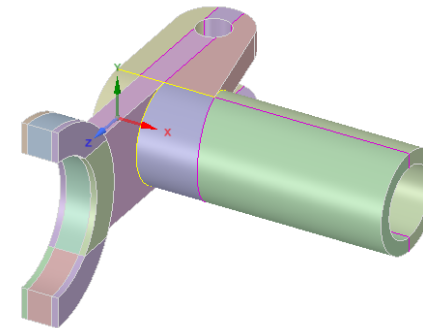
Shared Topology (2)

- Share:

  - Will identify and share topology among the bodies in the model

    - *It will search the visible bodies for sharing opportunities by imprinting the bodies with each others. Potential topologies are shown in red*

    - *You can choose topologies to exclude by clicking the Exclude Problem tool guide, which will then appear in green*

    - *Remaining candidates are shared after selecting the Complete button*
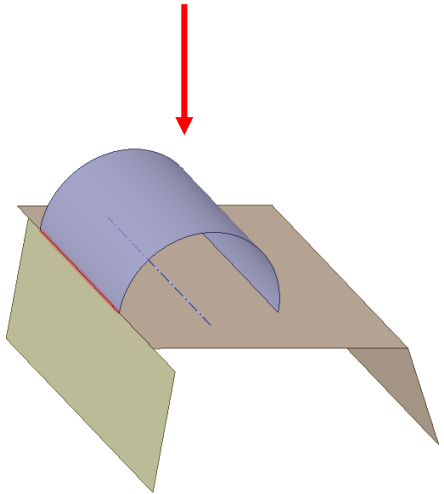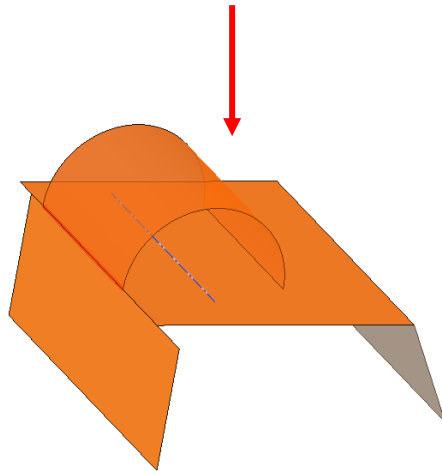
Shared Topology (3)

- ## Manual Selection:
  - You can manually select groups of topologies for sharing
    - Faces and edges may be chosen, control select additional entities if more than 2 need to be shared
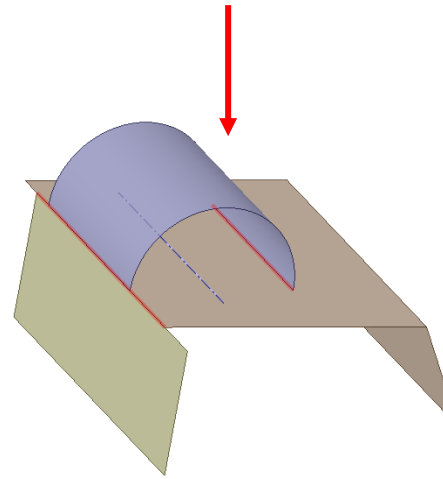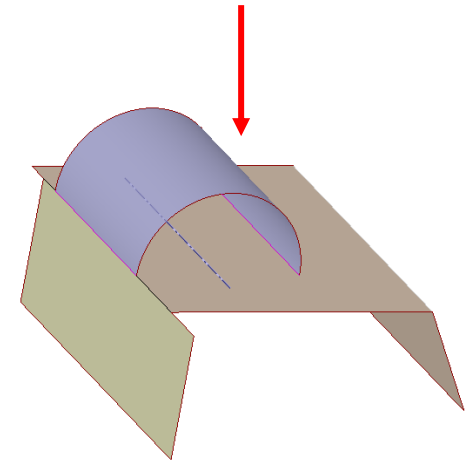
In the Share tool, select the edge

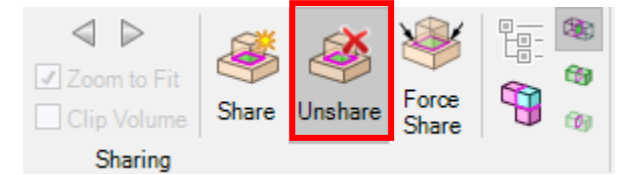**Hold Ctrl and select the surface body (ies)**

After selections are complete, the possible connections are highlighted

Clicking Complete will do the connections
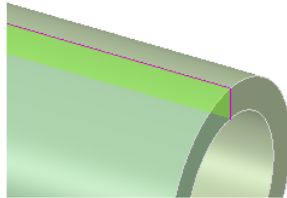
Shared Topology (4)

- Unshare:

  - You can select the entities in the graphics area which are to be unshared. The selection filter will allow selection of entity already shared.

  - The Handle Children checkbox may be used to easily unshare any lower topological entities also involved in sharing

  - Example:
    - **Initial Model**
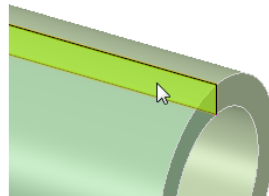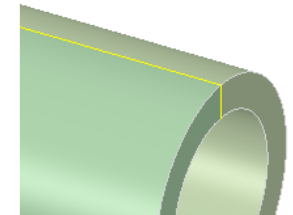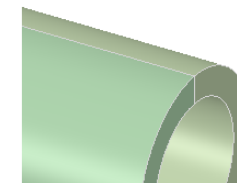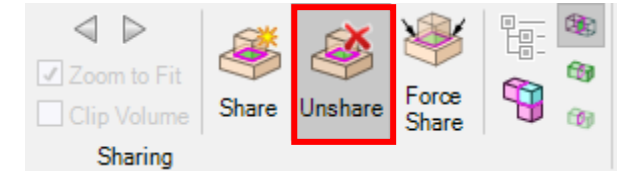
    - **Selection**

    - **Result with <u>Handle Children off</u>. Only the face is unshared, while the boundary edges remain connected**

   - **Result with <u>Handle Children on</u>. The face and its boundary edges are unshared**
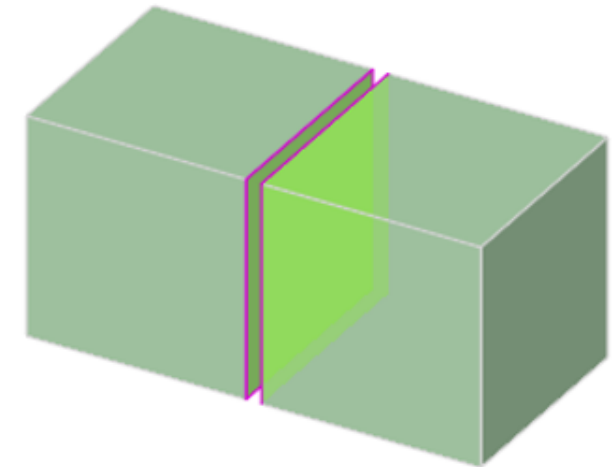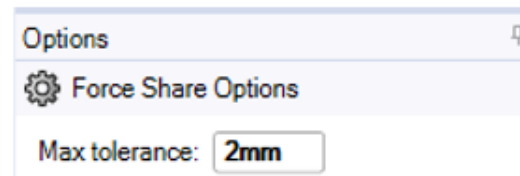
Shared Topology (5)

- The Unshare tool will behave differently depending on the type of topology selected:

  - Faces – Will remove any topology sharing from the face. If the handle children checkbox is checked, any shared edges belonging to the face will also be unshared.

  - Edges – Will remove any topology sharing from the edge and faces adjacent to the edge, since face sharing requires that all its edges also be shared. If the handle children checkbox is checked, any shared vertices belonging to the edge will also be unshared.

  - Vertices – When selecting a vertex, Ansys SpaceClaim will also highlight the edge associated with that vertex. Clicking that vertex will remove only the vertex associated with that edge from the set. Any other vertices shared at that intersection will remain. To remove all vertex sharing at a location, use box select around the vertex.

- To unshare all entities from the visible bodies, box select everything.

Shared Topology (6)
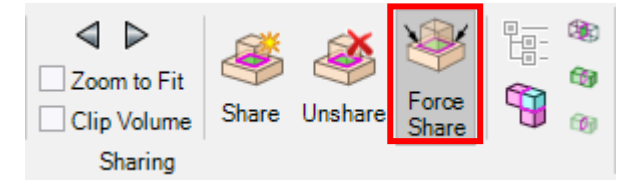
- The Force Share tool allows you to force a connection between Faces, Edges, and Beams. When using the tool, you can continue selecting Face, Edge, or Face-Edge pairs to share.
  - Click the Force Share tool. The tool automatically finds faces and edges within the tolerance limit and highlights them in the design area, as shown in the image above.
  - Select the faces and edges you want to be shared:
  - (Optional) Use the Navigation tools to view each sharing candidate one at a time before you force sharing.
  - Click the Complete tool guide to force sharing on the selected candidate(s).
- Beams are supported in Force Share.
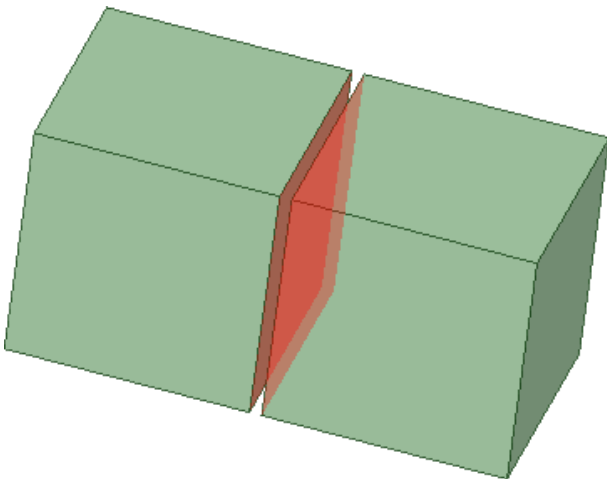
Shared Topology (7)

- **View Assembly Structure:**
  - Allows you to see how the assembly will appear when transferred downstream to Ansys Workbench
  - Bodies sharing topology must belong to the same part to be meshed conformally
  - When bodies share topology, Ansys SpaceClaim will group them by their nearest common parent component. If no common parent component among the shared bodies exists, then the entire assembly will be transferred as a single multi-body part.

- **Example:**
  - For this model, bodies B and C shared topology with each other and bodies E and F share topology with each other
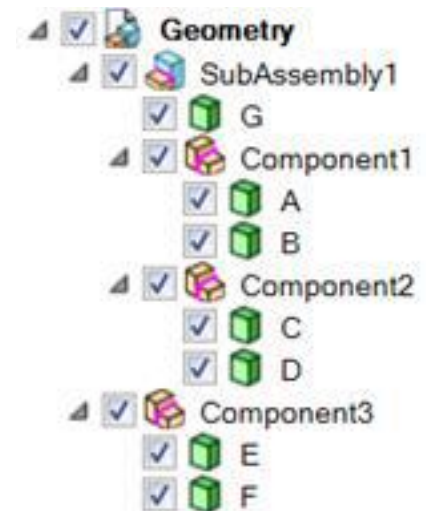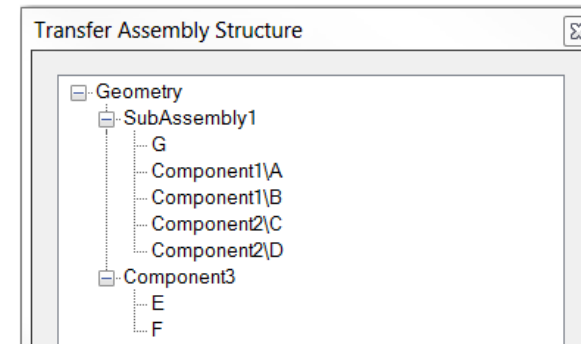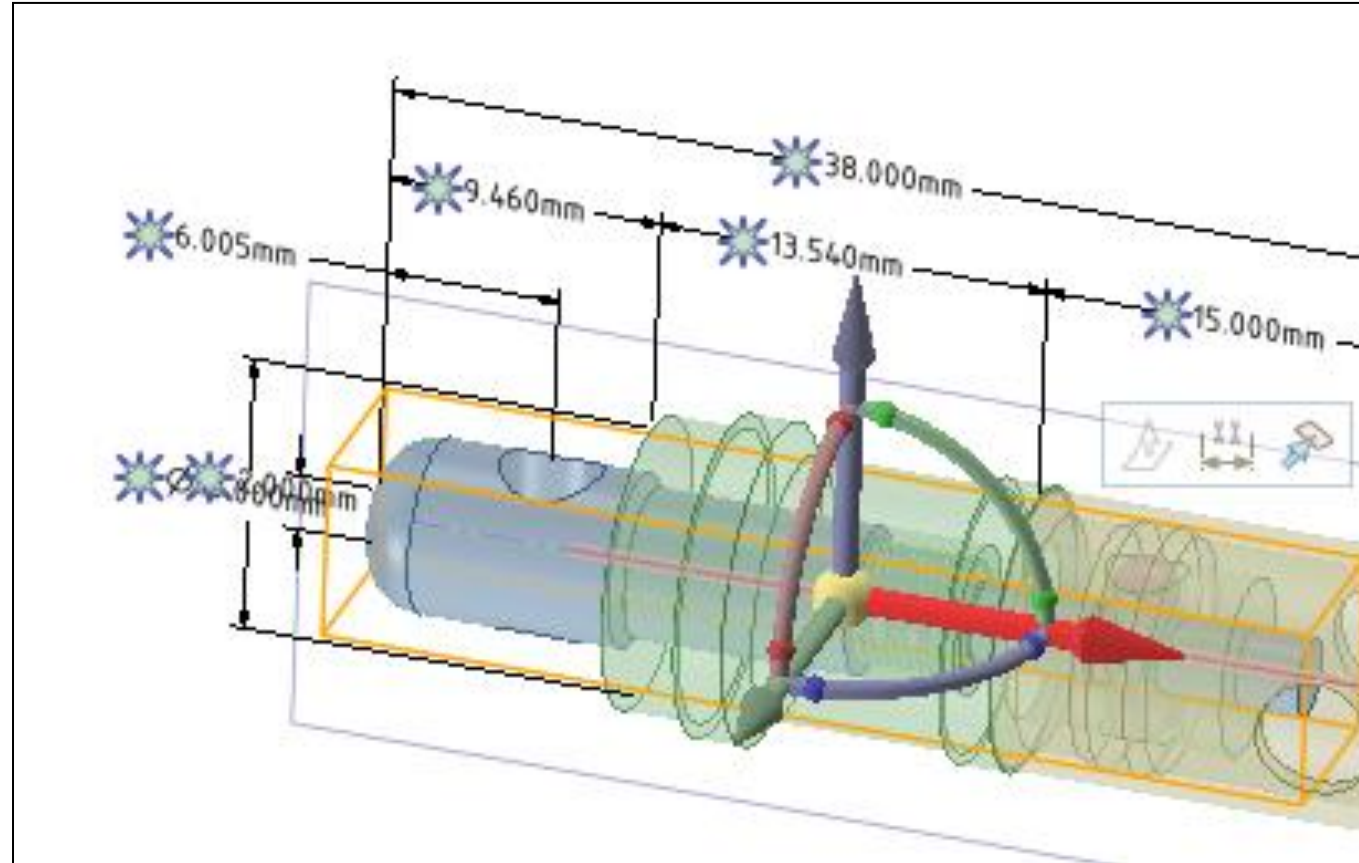
  - The View Assembly Structure shows this assembly

  - Because bodies B and C share topology and they do not belong to the same component, they must be grouped into the same part at their nearest common parent, which is SubAssembly1 with bodies A, D and G. Bodies E and F are already in the same component, so they would transfer as a multi-body part

Summary

- SpaceClaim allows you to define a material properties and import them to Workbench

- We can create Named Selections and Parameters in SpaceClaim using the groups tab

- Paramters can also be created vis the Block Recording

- Shared Topology can be done automatically or manually in SpaceClaim; many features like Share, Unshare and Force Share can be used for this purpose

Workshop 5.1 Parameters

End of presentation