Introduction to Ansys SpaceClaim

# Workshop 5.1: Parameters

Release 2022 R1

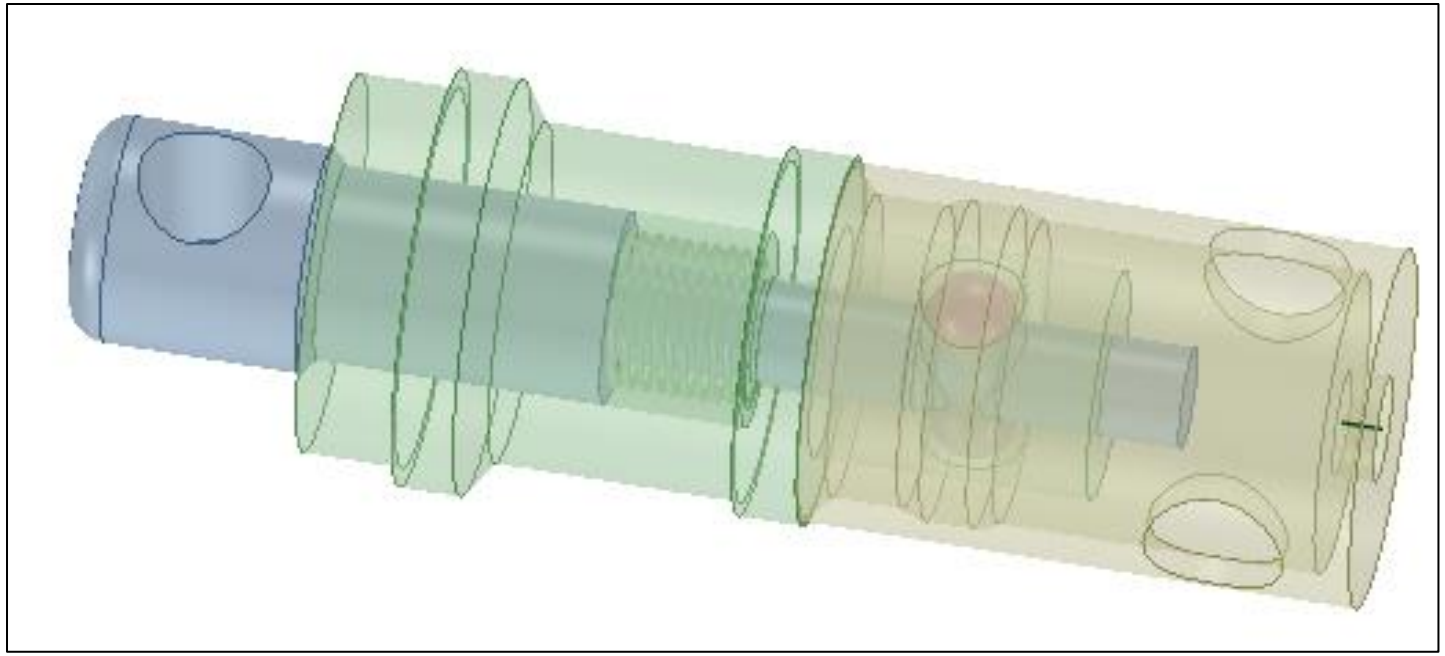
Please note:
- These training materials were developed and tested in Ansys Release 2022 R1. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.
- Although some workshop files may open successfully in previous releases, backward compatibility is somewhat unlikely and is not guaranteed.

# Overview

- Completion of this workshop involves:
  - Named Selections creation
  - Driving Dimensions creation
  - Exporting Named Selections and Parameters to Workbench

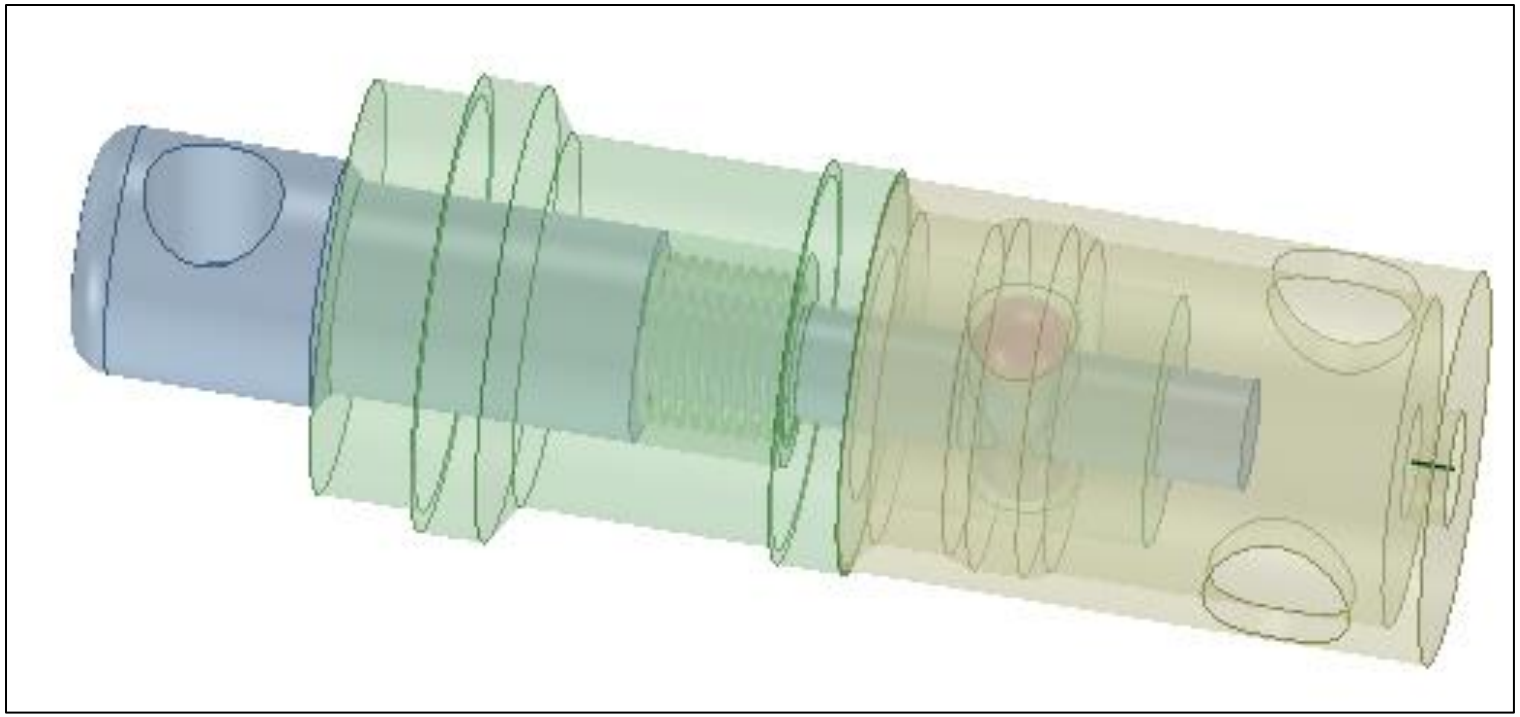
# Objectives

- Goals:
  - Create parameters
  - Update parameters from workbench



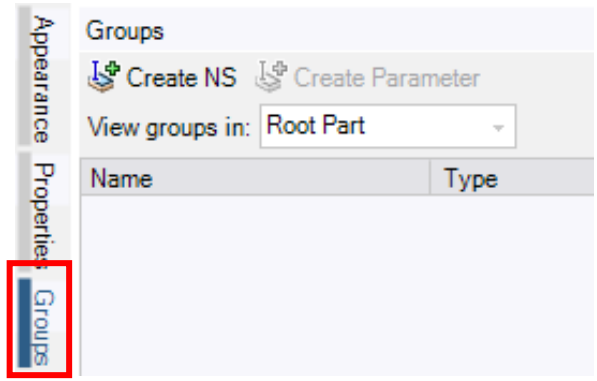
# Importing the geometry

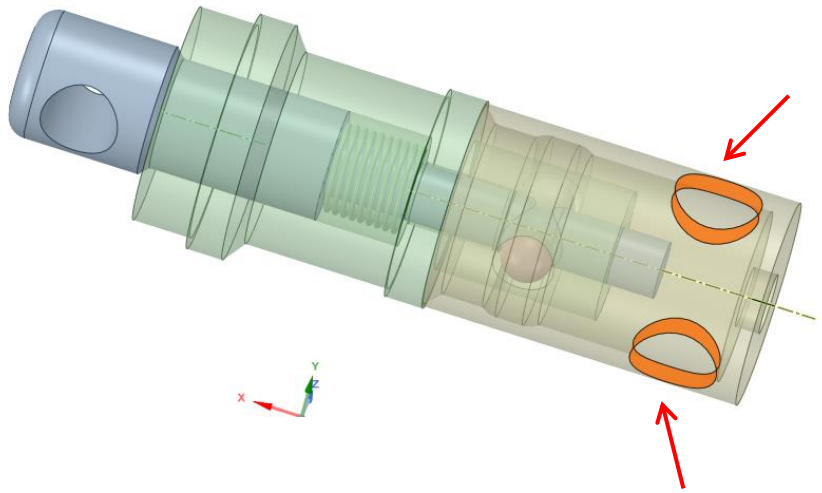
- Opening Document
  - Open parameter.scdoc from the Module06 workshop input files

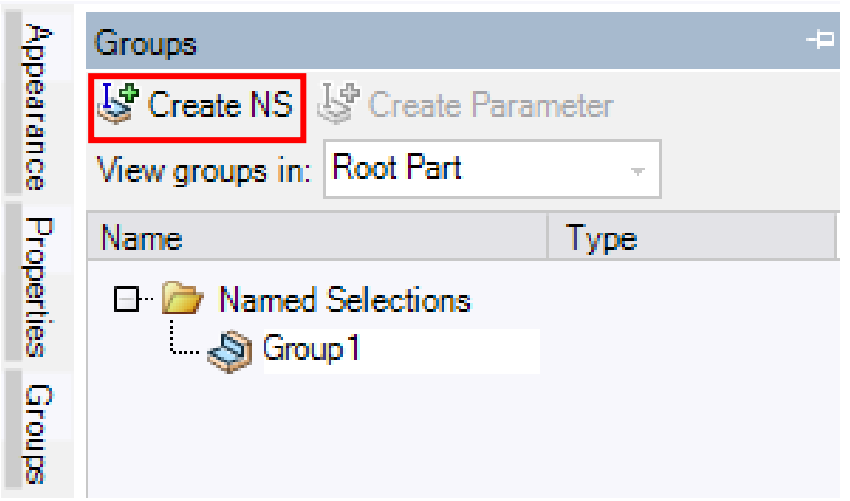


# Creating Named Selections (1)

- Groups Panel
  - Right side of the structure tree there are 3 tabs of which “Groups” tab
  - Click the two cylindrical surfaces (right of the model)
  - Click the Create Group button on the Groups panel.  This will create Group1 in the Named Selections folder. *The shortcut for creating a group is ctrl+G*
  - Named selections will be transferred with the geometry in the Ansys Workbench

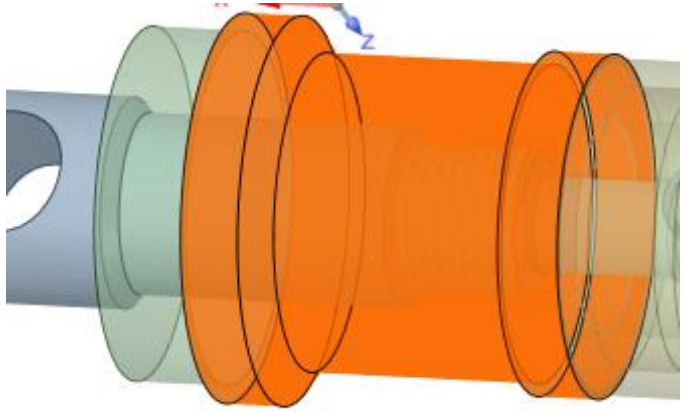


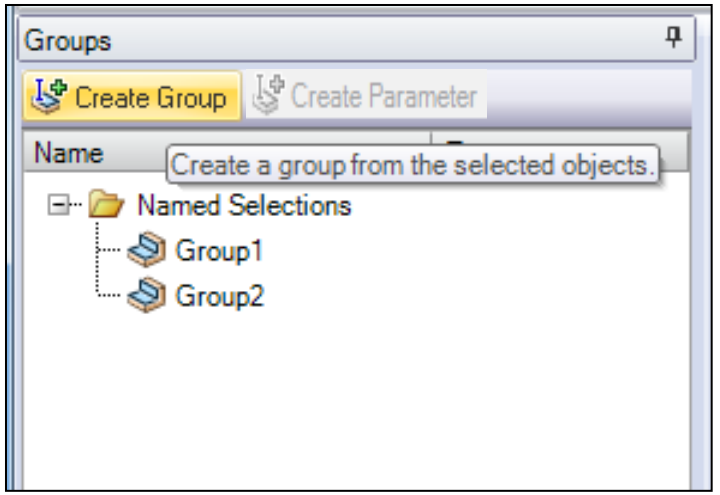



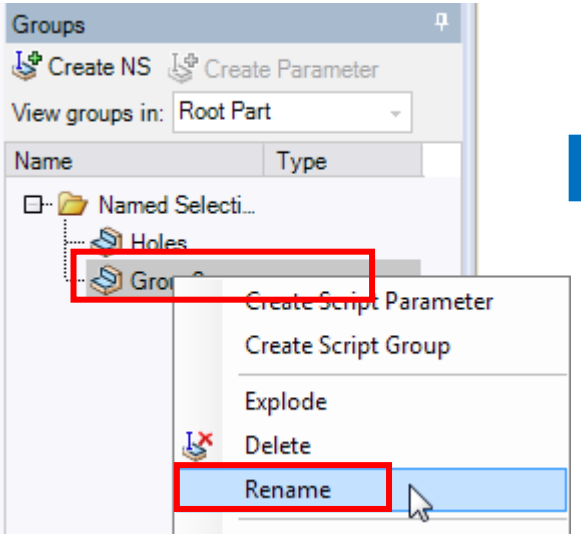


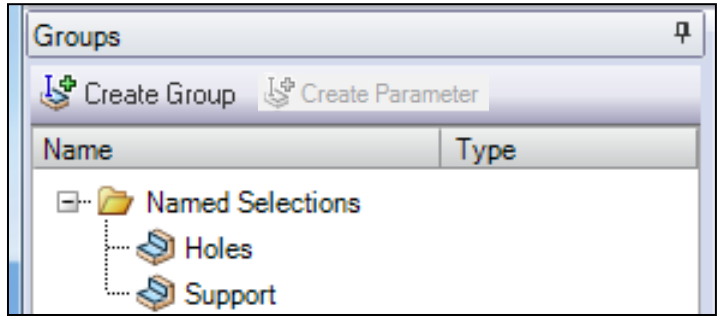
# Creating Named Selections (2)

- Named Selections
  - Select a few surfaces on the green solid.
    - Using the Design/select menu
  - Click Create Group to make another named selection.
  - Right click on the groups and rename them based on the selection.





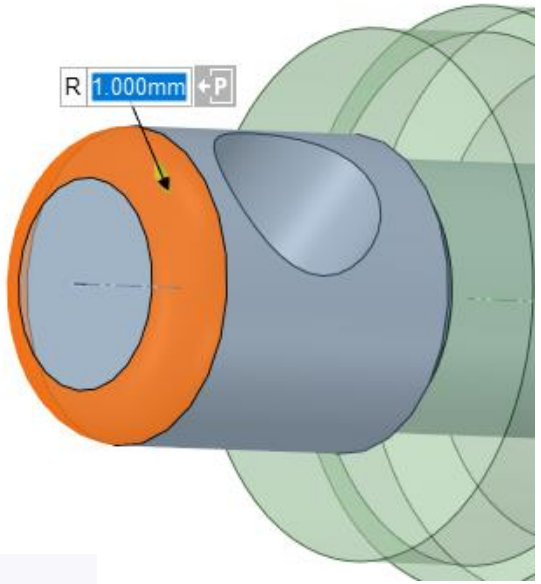



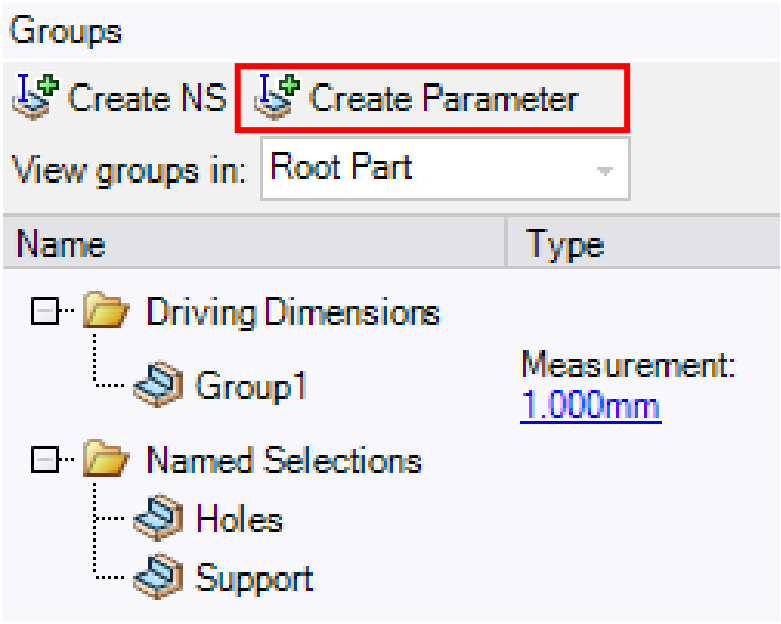



# Creating Driving Dimensions (1)

- Driving Dimensions
  - Activate the Pull tool
  - Select the round surface on the left side of the part.
    - A radius dimension does appear in the display
    - This size is a property of the fillet which can be controlled from the property panel
  - Click Create Parameter.
    - Instead of a named selection a driving dimension was created.

Note: clicking on the P symbol at the right of the radius will also create the driving dimension

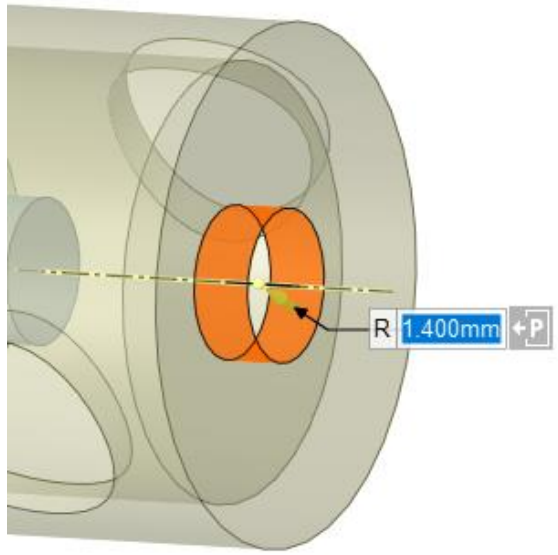
# Creating Driving Dimensions (2)

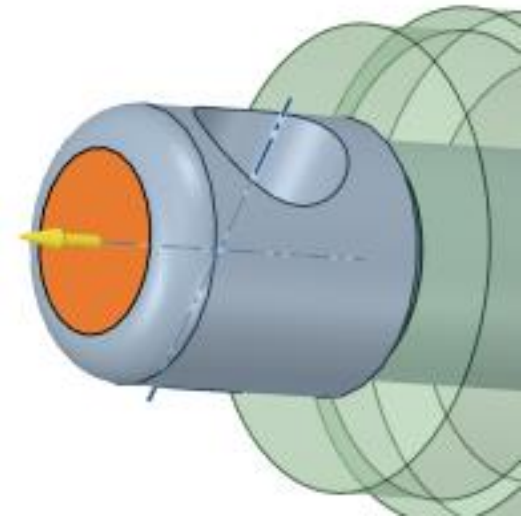
- Adding Driving Dimension
  - Pull tool activated
  - Select the hole
    - A dimension on screen (radius)
  - Click Create Group or ctrl+D to add the Driving Dimension.

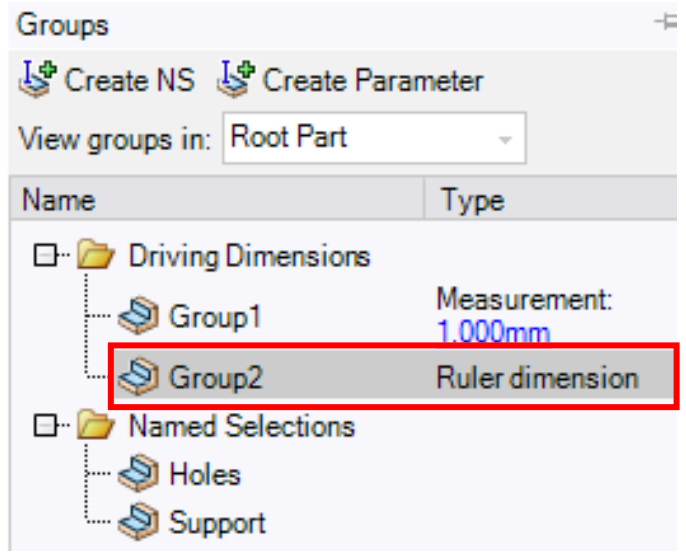
    Note: clicking on the P symbol at the right of the radius will also create the driving dimension

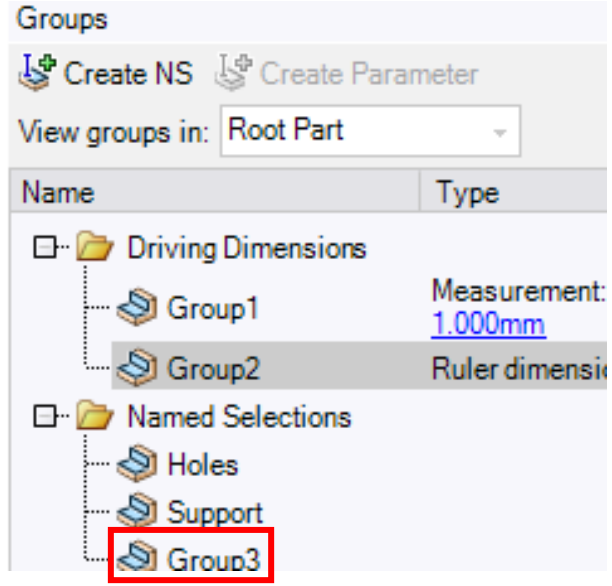
  - Click the top face with the pull tool
  - Click Create Group

    Note : As no dimension was shown with the Pull tool, a Named Selection was created.



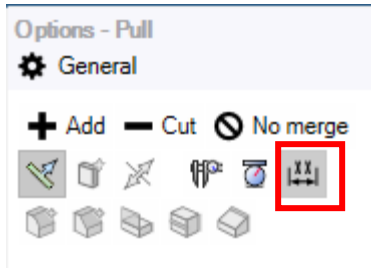



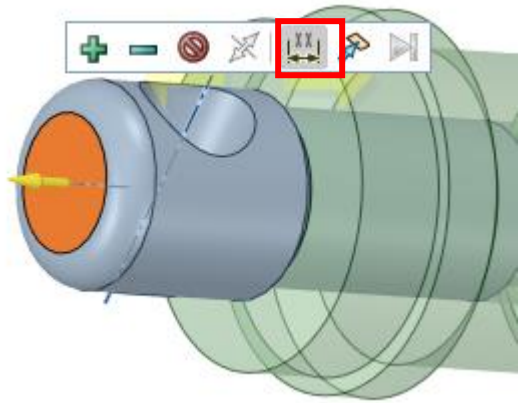


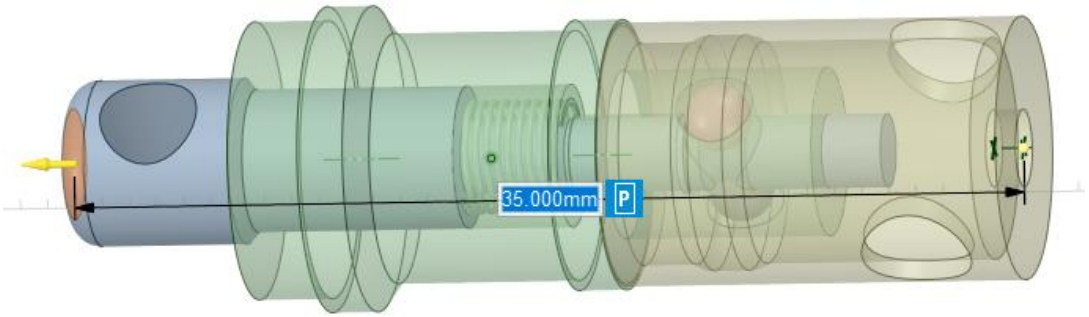



# Creating Driving Dimensions (3)

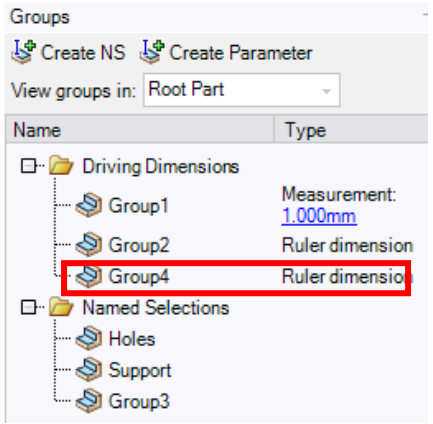
- Creating Ruler Dimensions
  - Pull tool activated
  - Click the top face and then click the Ruler icon in the mini toolbar.
  - *NOTE: The ruler option is also located on the left side of the screen under the Pull Options.*



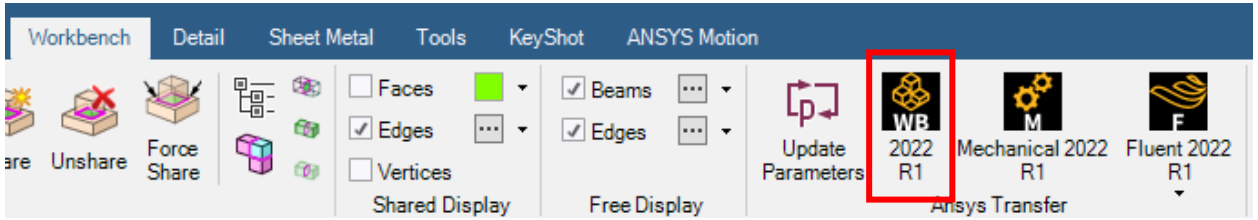


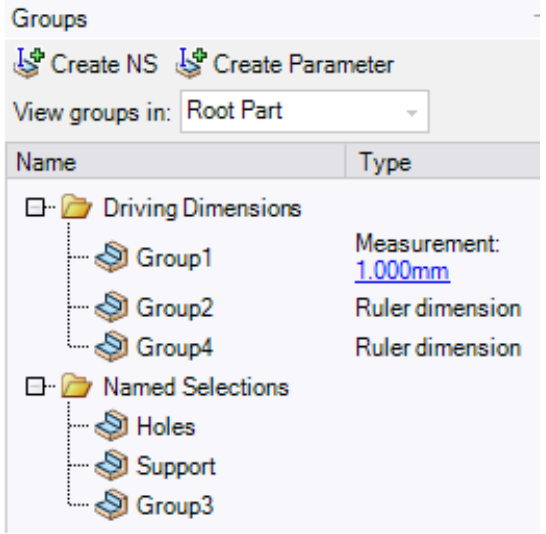

  - After clicking the Ruler button select the opposite face of the model to define a dimension
    - control the total length.
  - Click Create Group or the P symbol to add our last Driving Dimension.



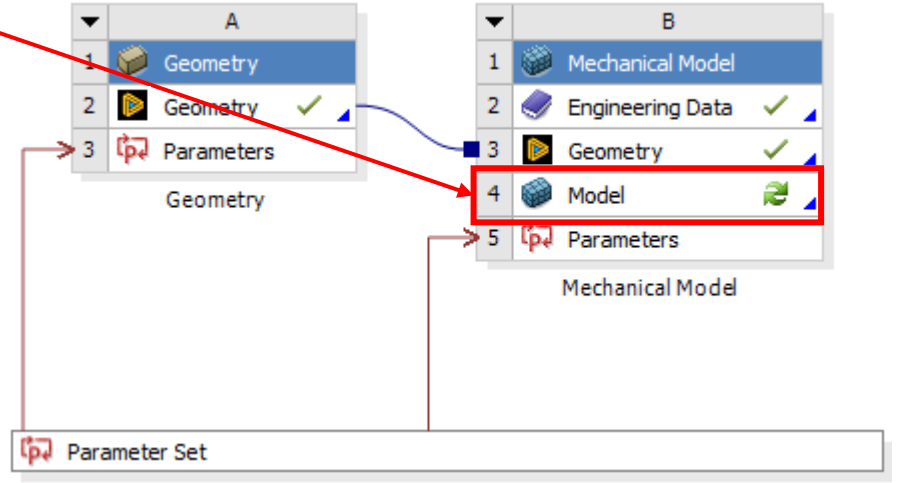
# Transfer to Ansys (1)

- Transfer to Ansys
  - Driving Dimensions and Named Selections can be renamed (RMB click)
  - Click the Ansys Workbench button to launch the geometry into Workbench



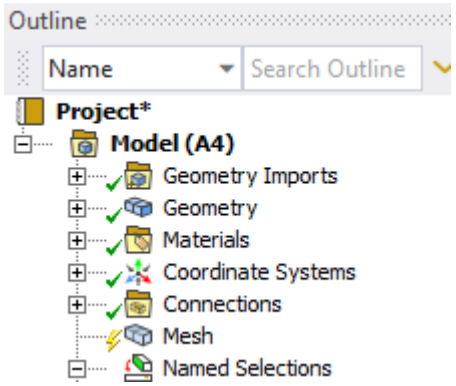



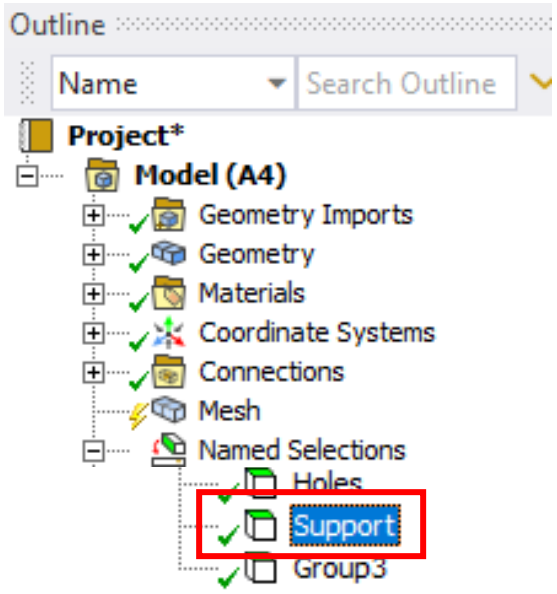

  - Drag and drop a ‘Mechanical Model’ box to the geometry from the project page
  - Double click on the ‘Model’ cell

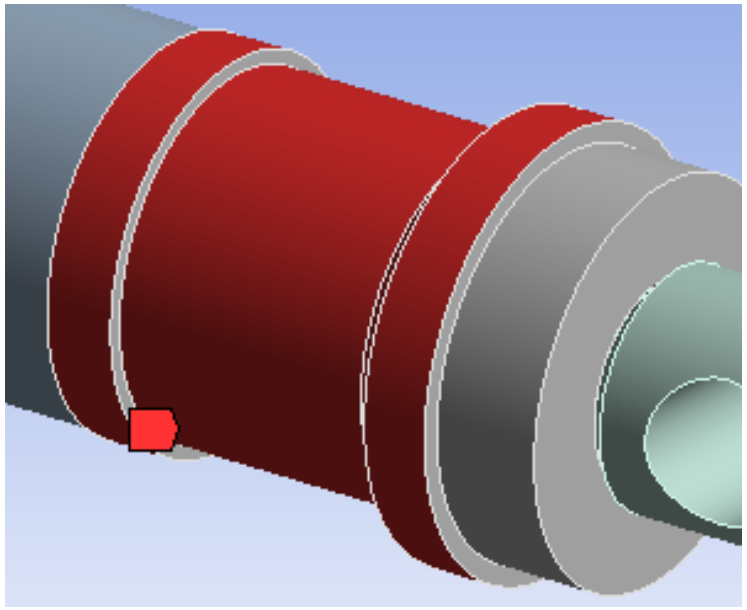


# Transfer to Ansys (2)

- Named Selections
  - The Project Tree will contain the Named selections from SpaceClaim.
  - Expand the group to see the different selections.
  - Select one Named selection
    - It will highlight in red
    - A boundary condition (load, support) can be applied to the faces.

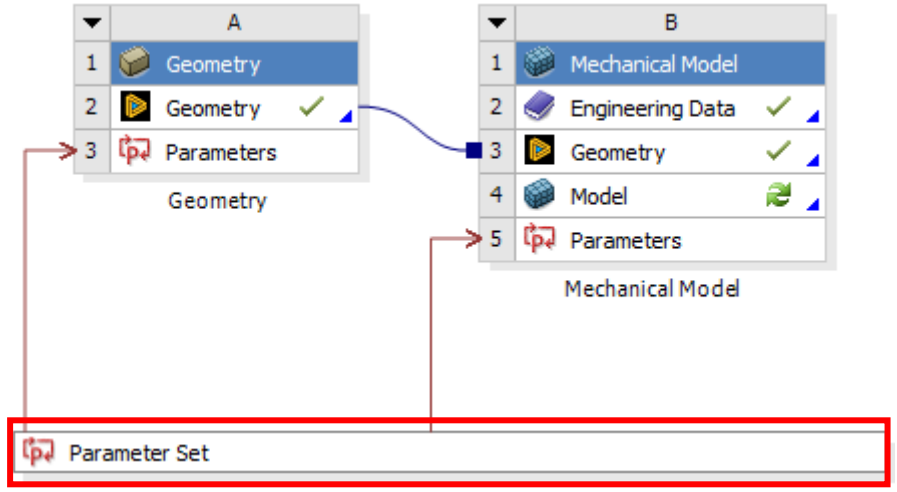


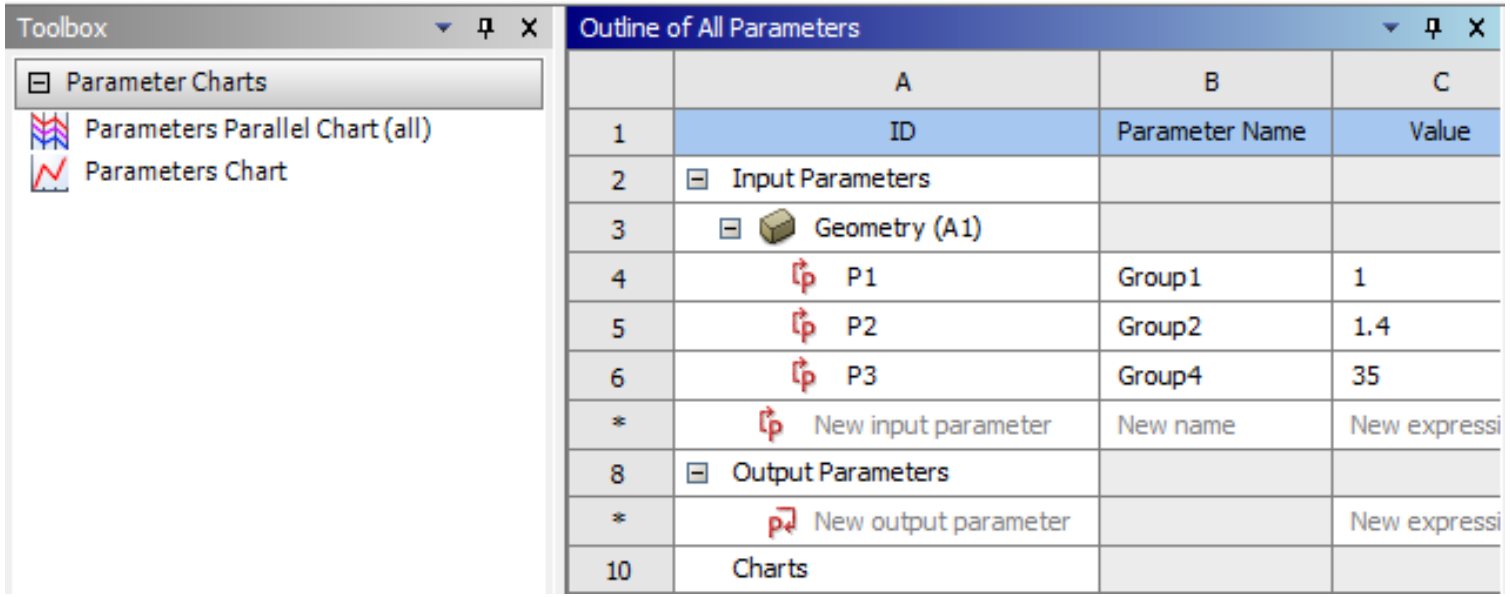



# Transfer to Ansys (3)

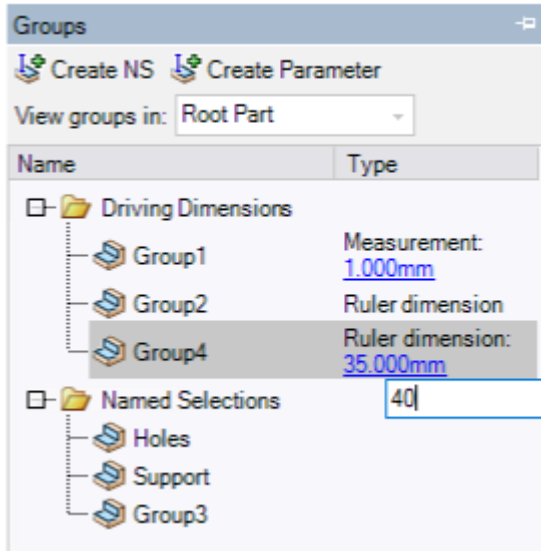
- CAD Parameters
  - As the transfer is operated a Parameter Set box will appear in the Workbench Project page
    - Click on this latter
      - Gives you access to all the driving parameters previously defined into SpaceClaim
      - Design points can now be defined.

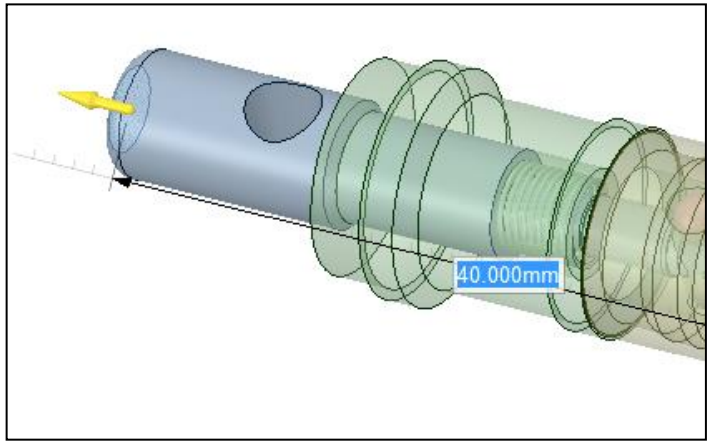


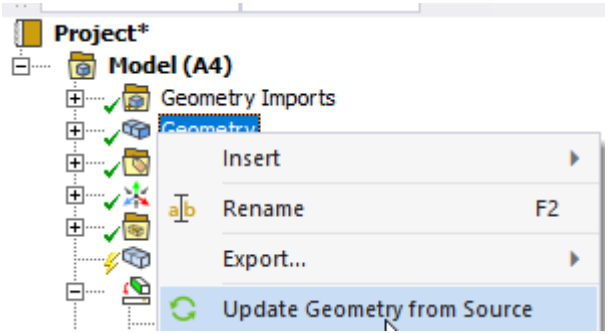



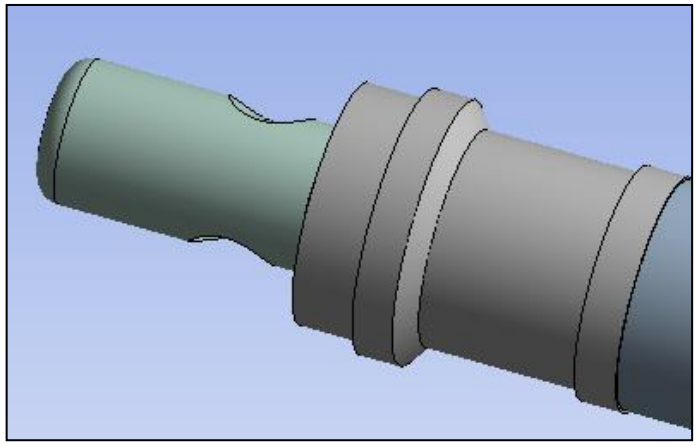
# Transfer to Ansys (4)

- Model Updates
  - The model will change in both Ansys and SpaceClaim
  - In SpaceClaim / “Groups” tab
    - Click on one of the driving dimensions
    - Give access to the value initially defined
    - Change one (or more) of this value
      - Click on the value (in blue) and dial a new one
      - The model will be updated in SpaceClaim
  - In Ansys Meshing/Mechanical
    - RMB click on Geometry
    - Update Geometry from Source



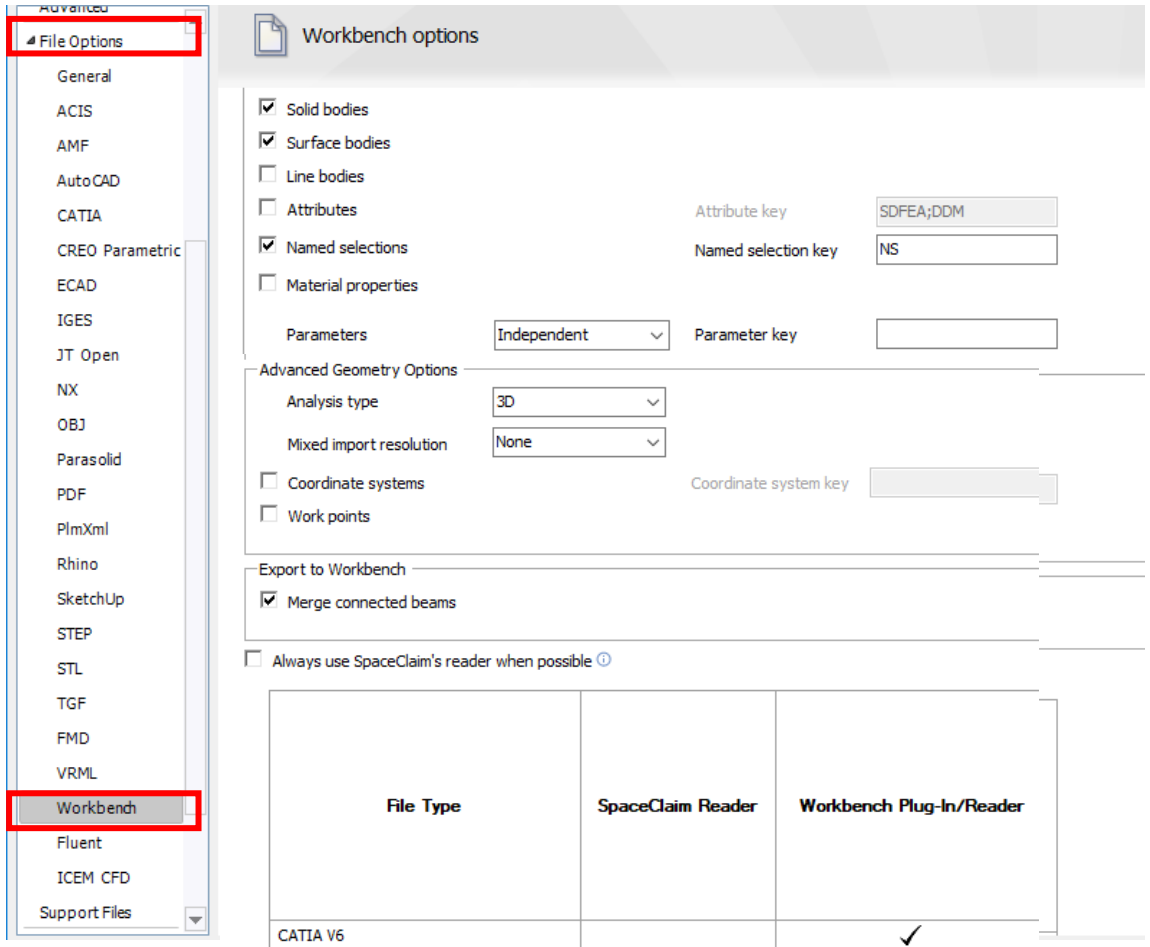



# Transfer to Ansys (5)

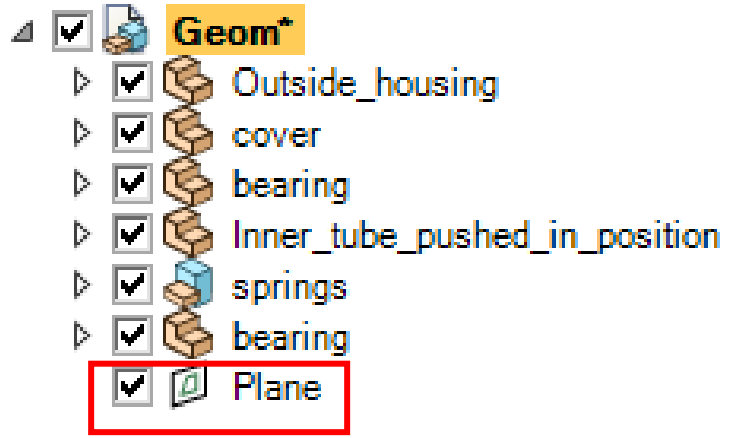
- SpaceClaim options
  - These options will have effect only if SpaceClaim is defined as an external CAD software.
  - In our case, these options will change nothing.

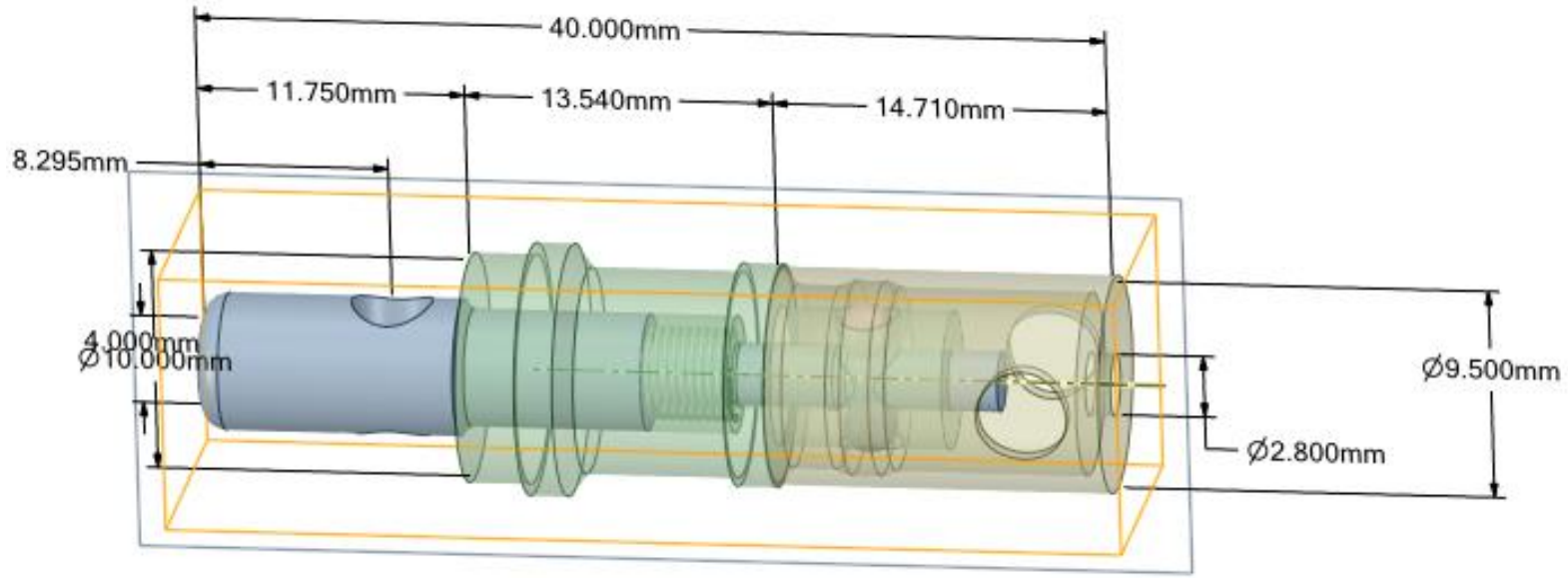



| File Type | SpaceClaim Reader | Workbench Plug-In/Reader |
|---|---|---|
| CATIA V6 | | ✓ |
| CFX Mesh | | ✓ |
| Creo Elements | | ✓ |
| DesignModeler | | ✓ |
| DesignSpace | | ✓ |
| ElectroMagnetics | | ✓ |
| GAMBIT | | ✓ |
| Mechanical | | ✓ |
| Mesh | | ✓ |
| Part Manager | | ✓ |
| Solid Edge Weldment | | ✓ |

# Annotation Plane (1)

- Annotation Plane
  - Turn the Plane on inside the structure window.
  - A plane will appear with dimensions already created.
    - These dimensions can be used to create driving dimensions on a part.
  - Select the Move tool to start creating these new Driving Dimensions.

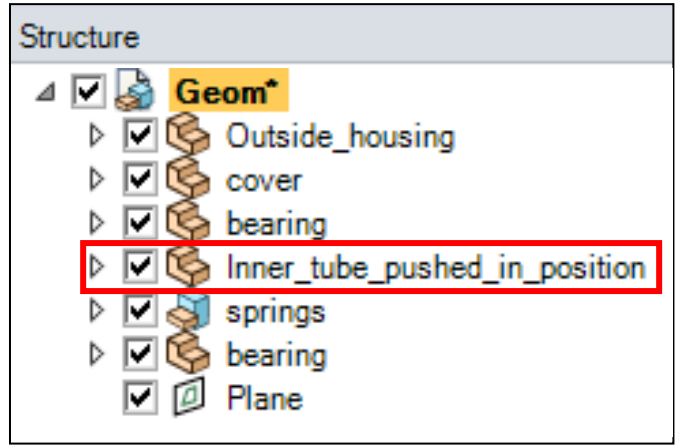


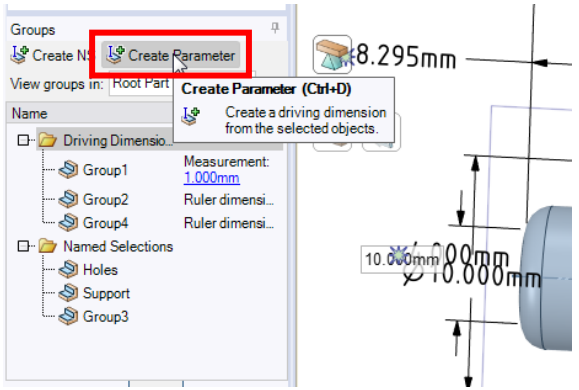



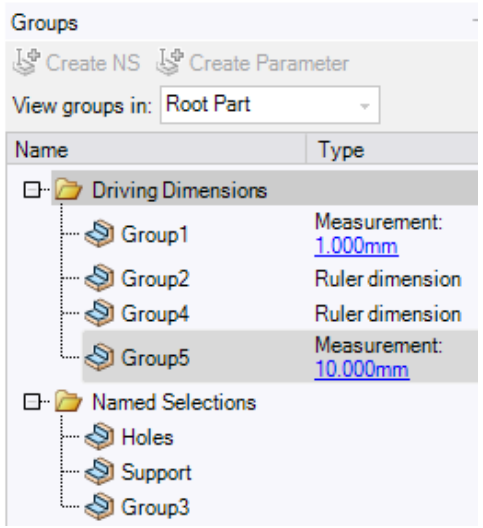
# Annotation Plane (2)

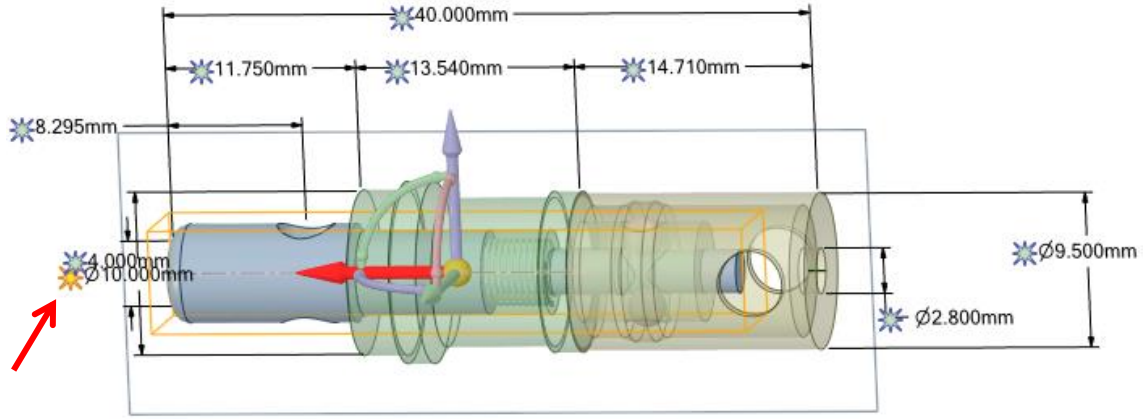
- Annotation Dimensions
  - Select the Inner tube pushed in position
    - In the structure tree.
  - To drive the model with the annotation dimensions a direction must be chosen with the Move tool.
    - Select the arrow parallel to the global x axis to specify a direction

    Note : when the arrow is chosen a spark boxes will appear next to the dimensions.
  - Click on the spark next to 10
  - Create a new Parameter



# Conclusion

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

Outline of All Parameters

| | A | B | C |
|---|---|---|---|
| 1 | ID | Parameter Name | Value |
| 2 | Input Parameters | | |
| 3 | Geometry (A1) | | |
| 4 | P1 | Group1 | 1 |
| 5 | P2 | Group2 | 1.4 |
| 6 | P3 | Group4 | 35 |
| 7 | P4 | Group5 | 5 |
| * | New input parameter | New name | New expressi |
| 9 | Output Parameters | | |
| * | New output parameter | | New expressi |
| 11 | Charts | | |

**End of presentation**