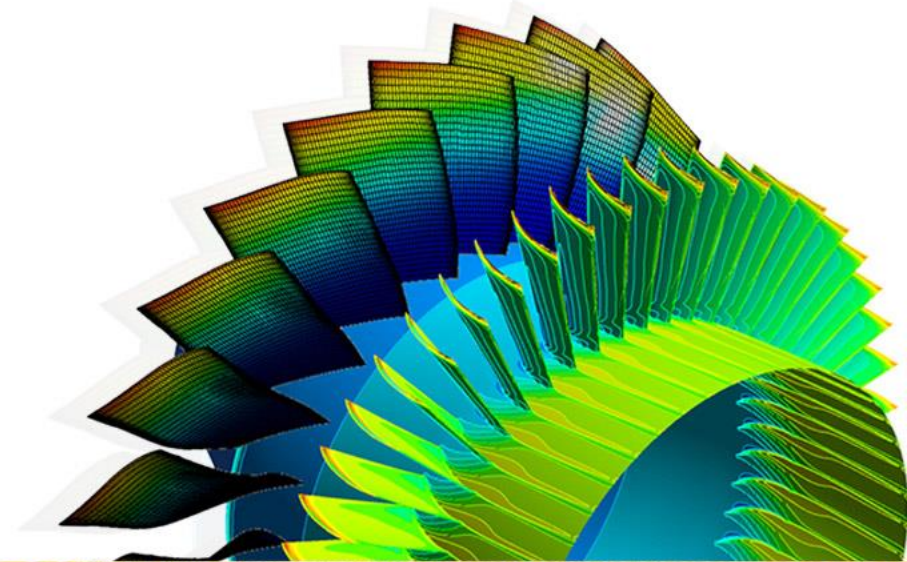




ANSYS Composite PrepPost 19.0

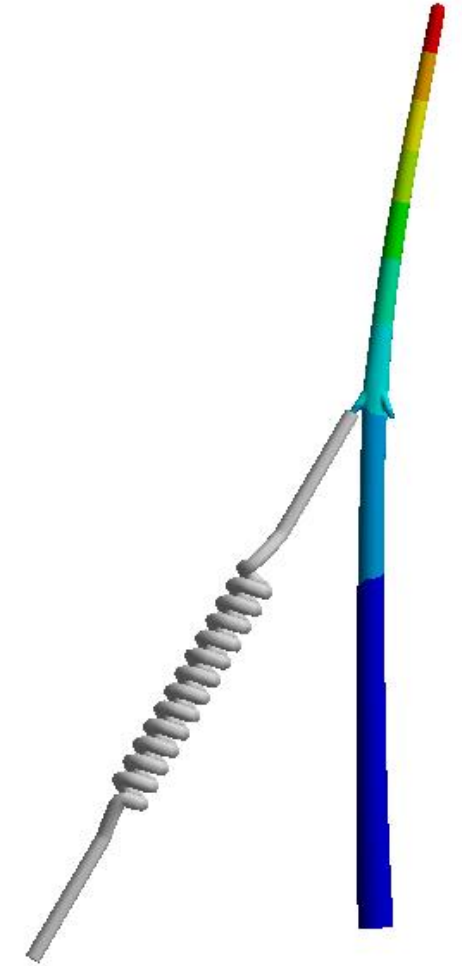
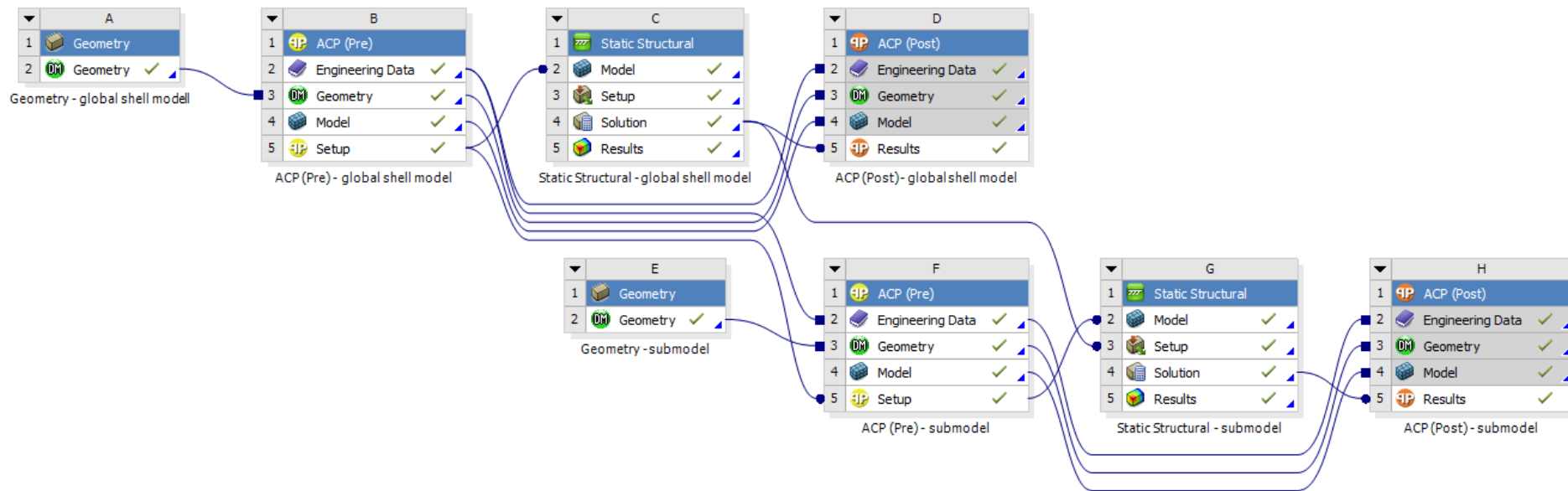
Workshop 10.4 – Submodelling



13. Workshop Submodelling

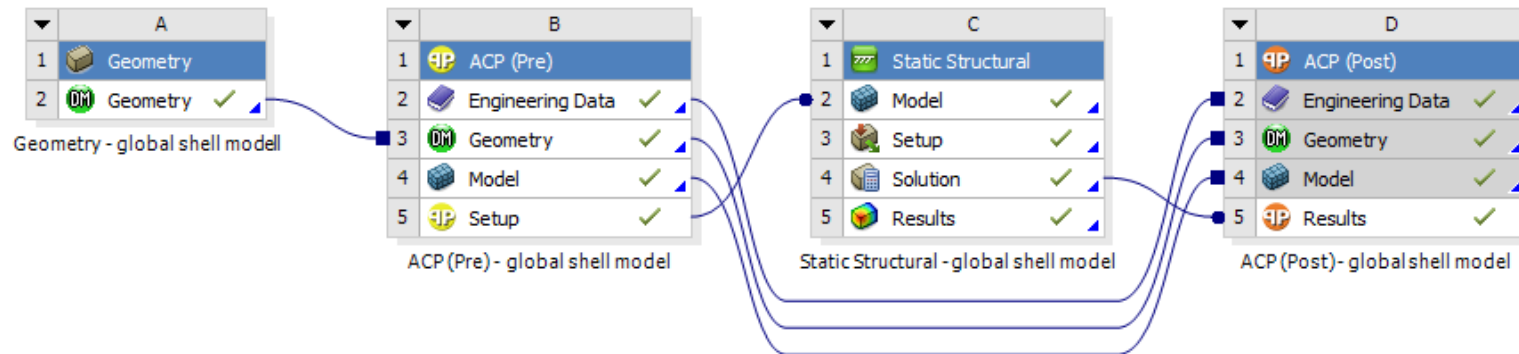
Agenda

- Analyze global static model
- Define Submodel analysis on Project schematic



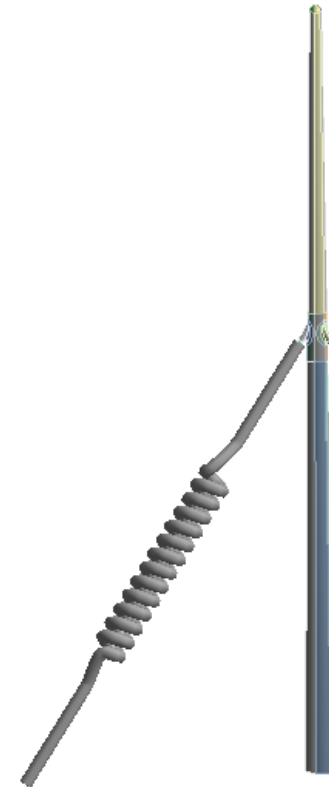
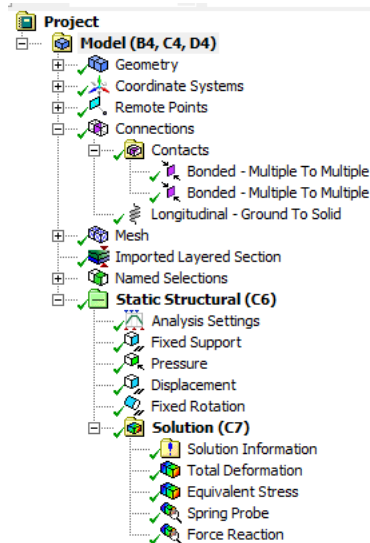
13. Workshop Submodelling

Open Workbench archive *Submodeling_FROM_START_19.0.wbpz*:



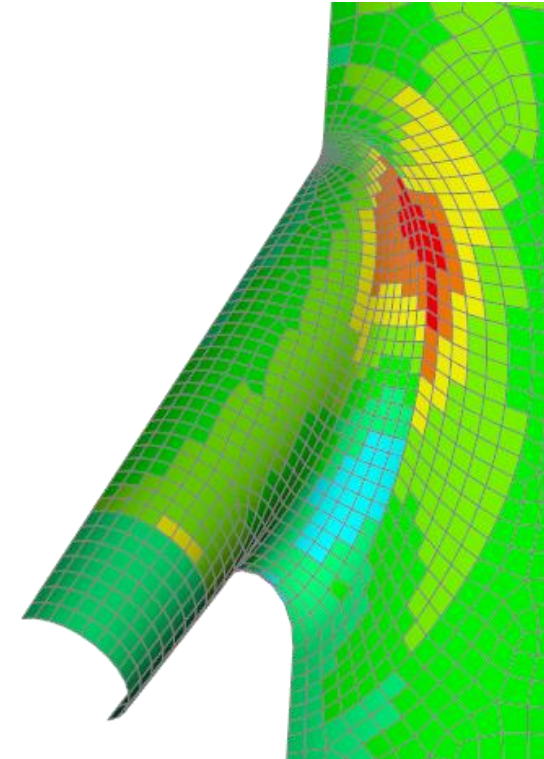
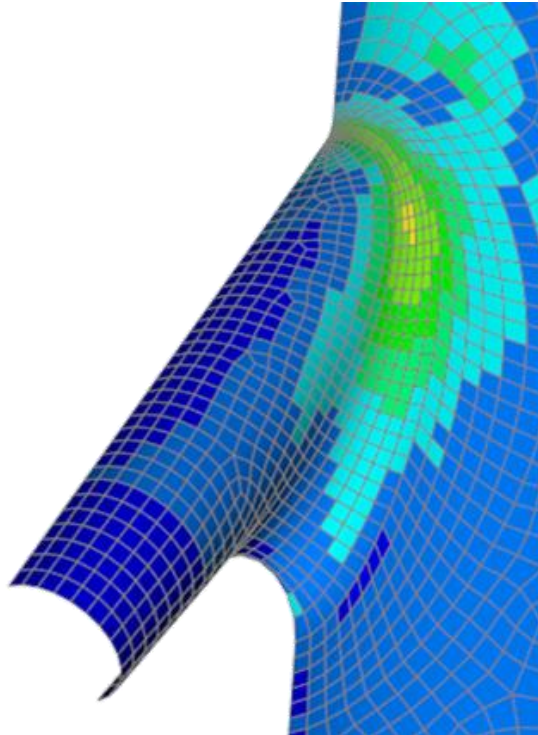
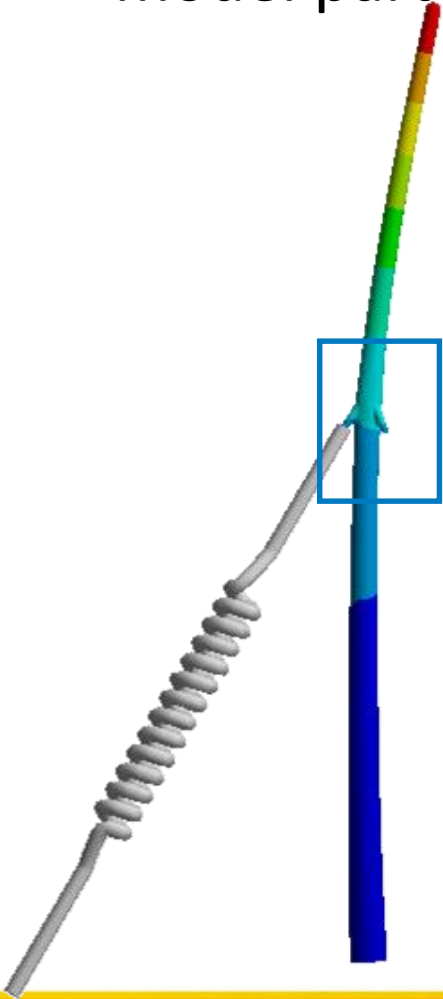
The Archive contains the fully defined global model.

Update all:



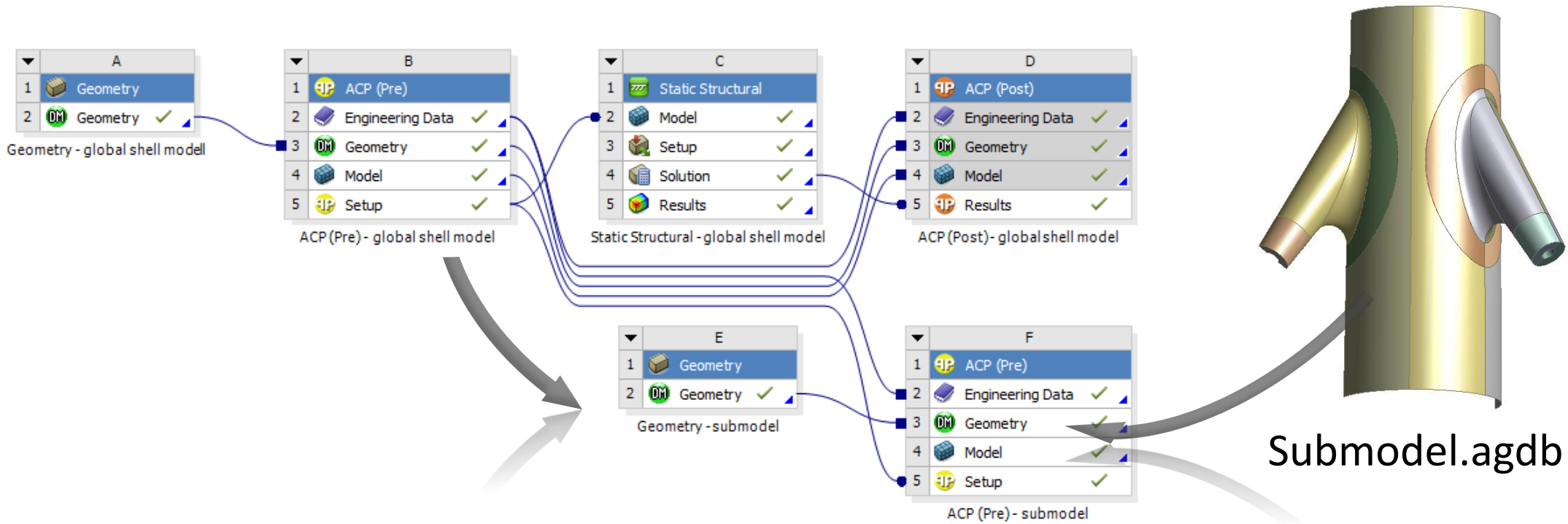
13. Workshop Submodelling

- Checking results in ACP-Post, shows critical area at mountings. We will use this model part for Submodel



13. Workshop Submodelling

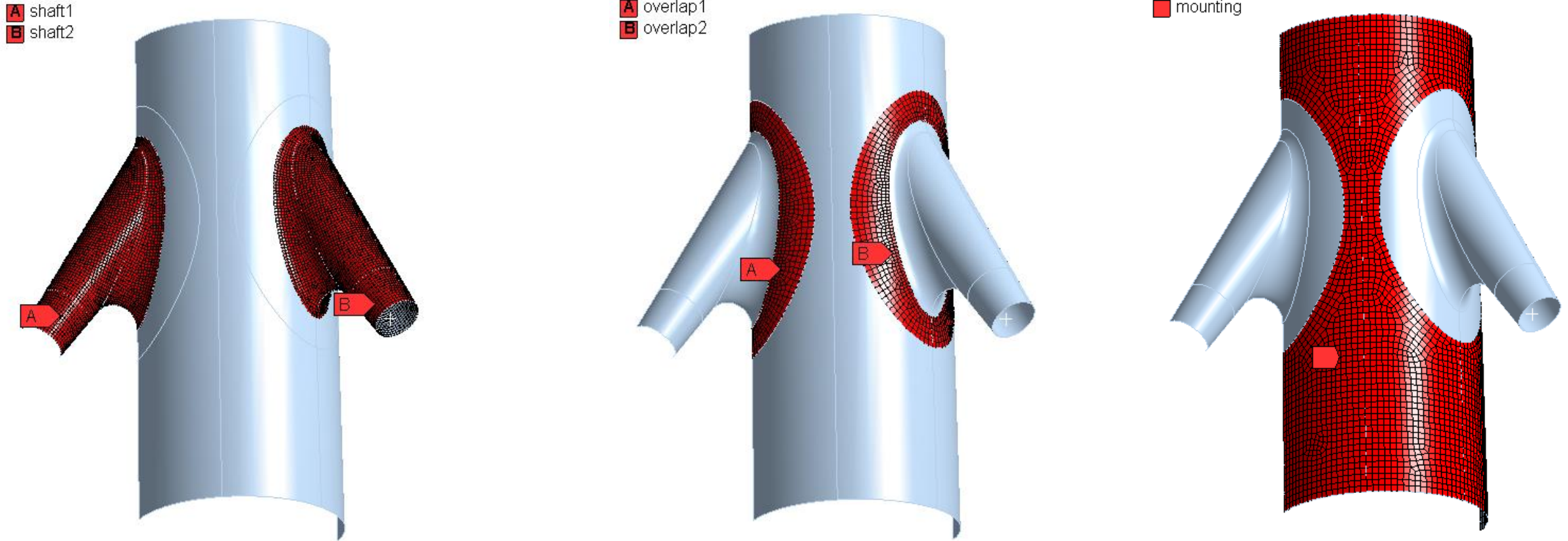
- Add a new ACP-Pre System to Project Schematic and import Submodel Geometry file.



Connect Engineering Data and ACP Setup Cell from global ACP Model. This will share Material Definitions as well as Stack Up of Laminate

13. Workshop Submodelling

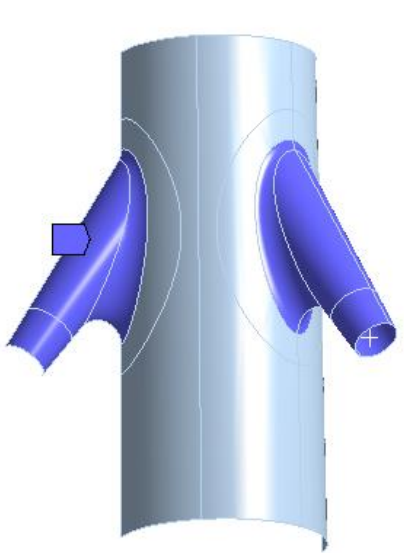
The Geometry file already contains defined Named Selections. These are used to correctly apply composite definitions from global model



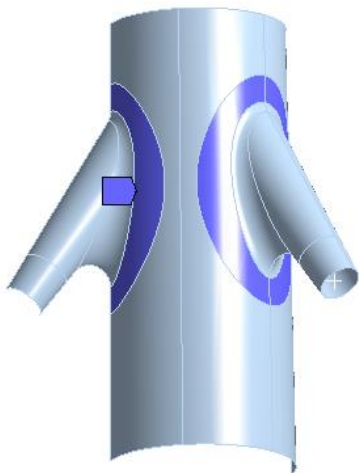
Additional edge components are also defined

13. Workshop Submodelling

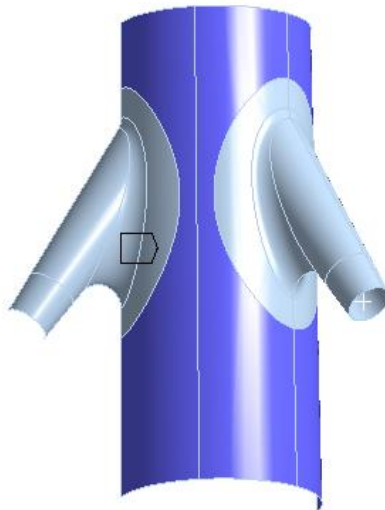
- Open Mechanical
- Set Thickness's and Materials
(Epoxy_Carbon_UD_230GPa_Preg, Stainless Steel for the inserts, 1 mm thickness for all surfaces)
- and add a body sizing to each of the shown body



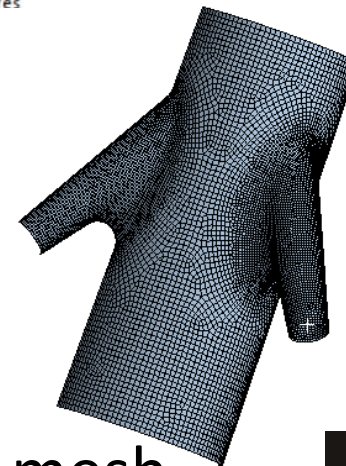
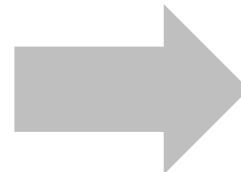
10 mm



15 mm



20 mm



Update mesh

The screenshot displays the ANSYS Workbench interface. On the left, the 'Outline' pane shows a hierarchical tree of the model. The 'Surface' is expanded, showing a list of 'Surface Body' entities. The 'Details of "Surface Body"' panel is open, showing properties for a selected surface body. The 'Definition' section includes a 'Thickness' property set to '1. mm'. The 'Material' section shows 'Epoxy_Carbon_UD_230GPa_Frepreg'. On the right, the 'Model' tree shows the overall structure, including 'Geometry', 'Coordinate Systems', 'Connections', 'Mesh', 'Body Sizing', 'Imported Layered Section', 'Named Selections', 'Static Structural (F6)', 'Analysis Settings', and 'Displacement SHELLS'. The 'Details of "Body Sizing" - Sizing' panel is also open, showing the 'Scope' (Geometry Selection, 3 Bodies) and 'Definition' (Element Size, 20. mm).

Details of "Surface Body"

Graphics Properties	
Definition	
<input type="checkbox"/> Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
<input type="checkbox"/> Thickness	1. mm
Thickness Mode	Manual
Offset Type	Middle
Material	
Assignment	Epoxy_Carbon_UD_230GPa_Frepreg
Nonlinear Effects	Yes

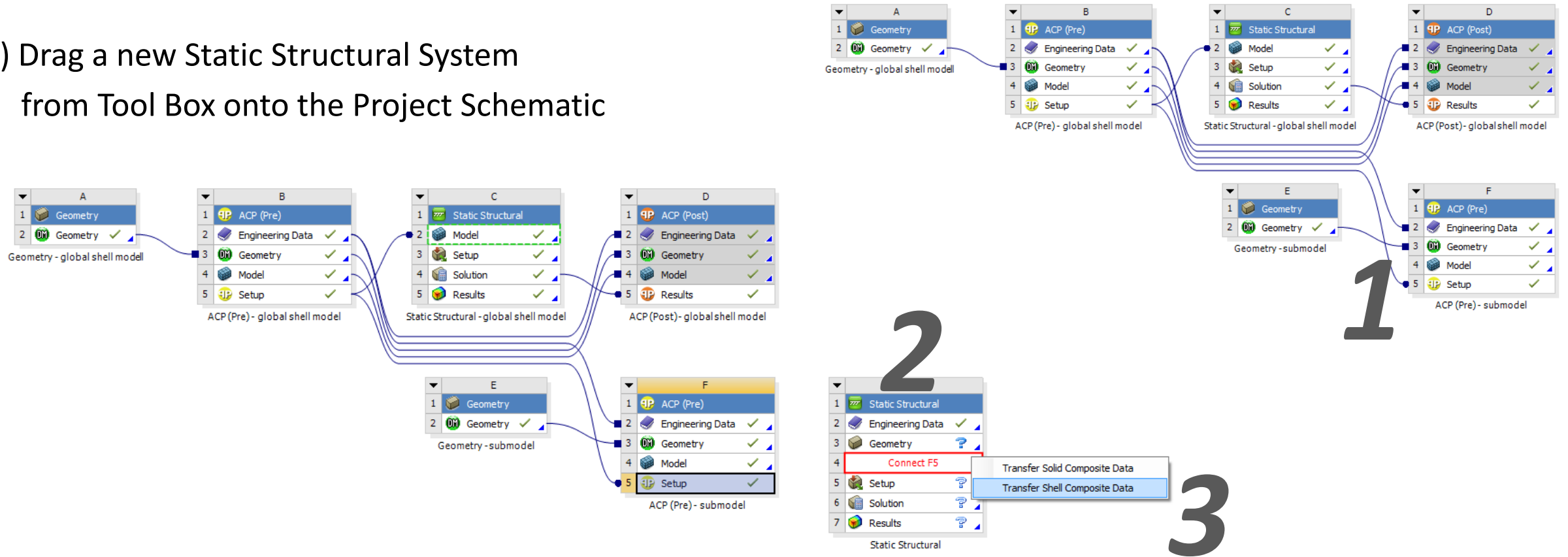
Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	3 Bodies
Definition	
Suppressed	No
Type	Element Size
<input type="checkbox"/> Element Size	20. mm
Behavior	Soft
<input type="checkbox"/> Curvature Normal Angle	Default
<input type="checkbox"/> Growth Rate	Default
<input type="checkbox"/> Local Min Size	Default (6.5954 mm)

13. Workshop Submodelling

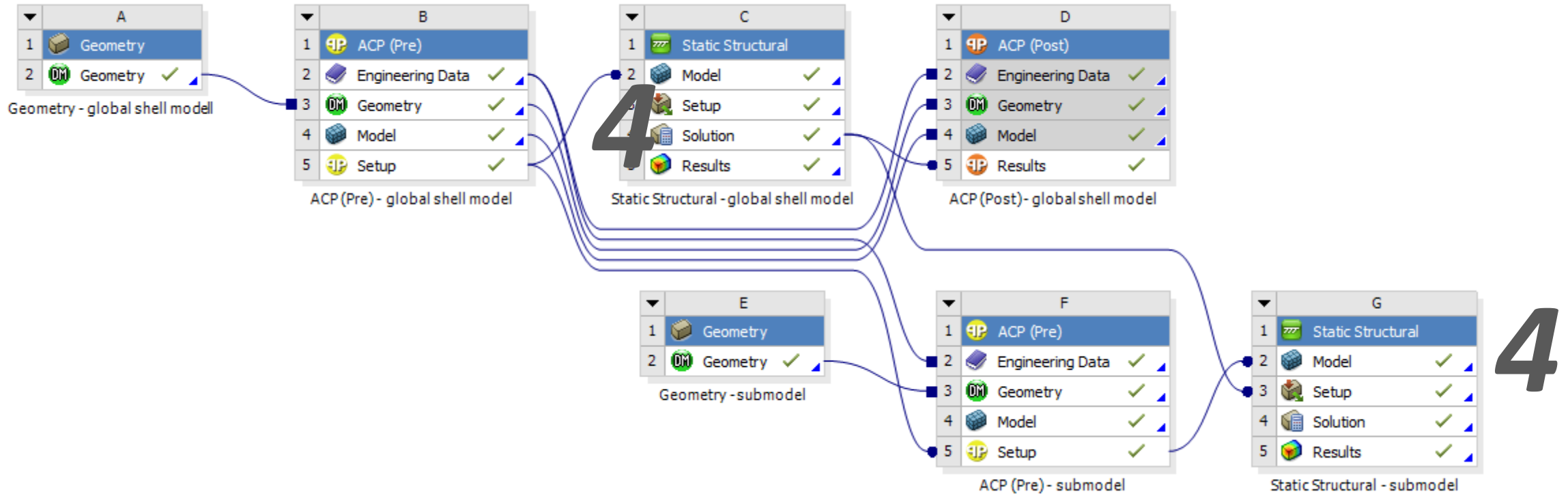
1) Close Mechanical and refresh / update Project Schematic

2) Drag a new Static Structural System
from Tool Box onto the Project Schematic



3) Manually connect Model cell of new Static Structural with Setup of ACP (Pre), transfer shell composite data

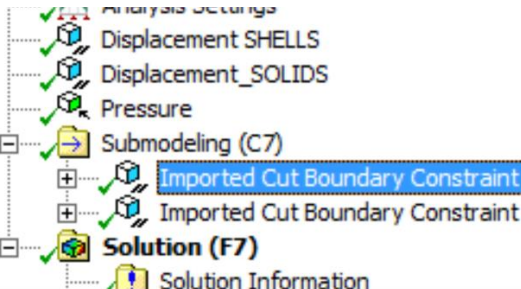
13. Workshop Submodelling



4) Manually connect Solution Cell of Global Model with Setup Cell of Static Structural system. This shares Results to Submodel

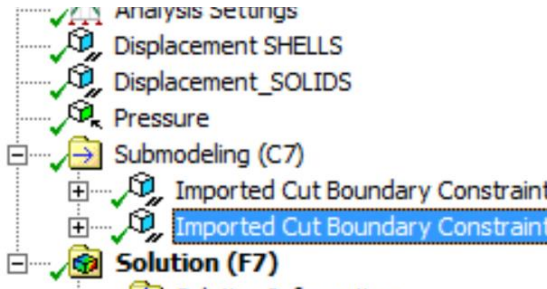
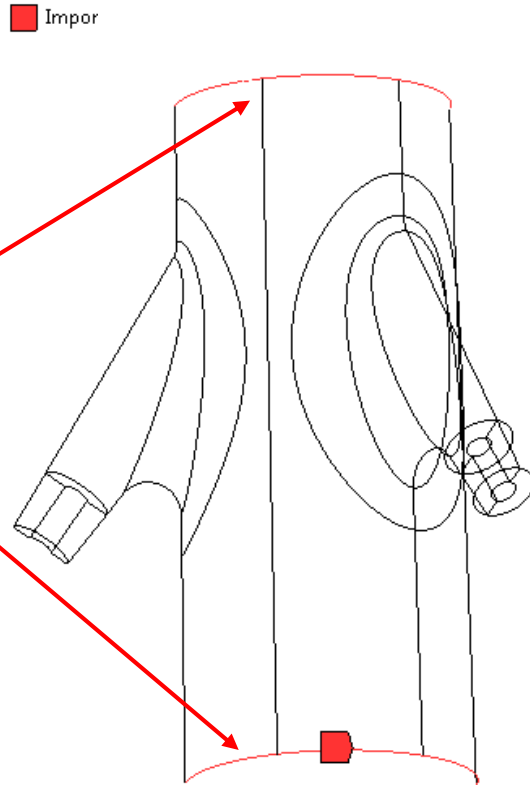
13. Workshop Submodelling

- Open Mechanical and add imported Boundary Conditions to Submodelling folder. Apply an *Imported Cut Boundary Constraint* to shell edges and one to Solid Faces at the Inserts



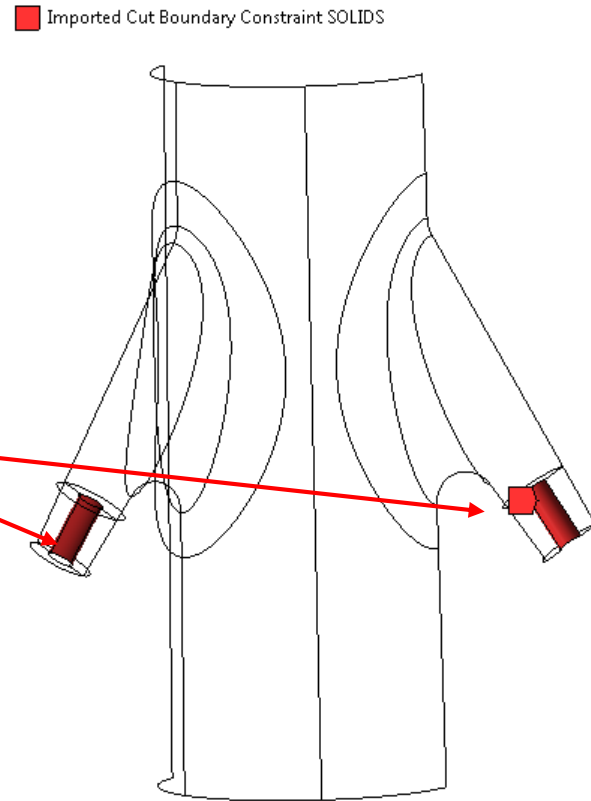
Details of "Imported Cut Boundary Constraint"

Scope	
Scoping Method	Geometry Selection
Geometry	6 Edges
<input type="checkbox"/> Shell Thickness Factor	0.
Definition	
Type	Imported Displacement and R
Sub Type	Both
Tabular Loading	Program Controlled
Suppressed	No
Override Constraints	No
Source Bodies	All
Submodeling Type	3D to 3D
Transfer Key	Shell-Shell
Source Time	Worksheet



Details of "Imported Cut Boundary Constraint"

Scope	
Scoping Method	Geometry Selection
Geometry	3 Faces
Definition	
Type	Imported Displacement
Tabular Loading	Program Controlled
Suppressed	No
Override Constraints	No
Source Bodies	All
Submodeling Type	3D to 3D
Transfer Key	Solid-Solid
Source Time	Worksheet



- Update Submodeling Folder (Import Load) to map displacements

13. Workshop Submodelling

Outline

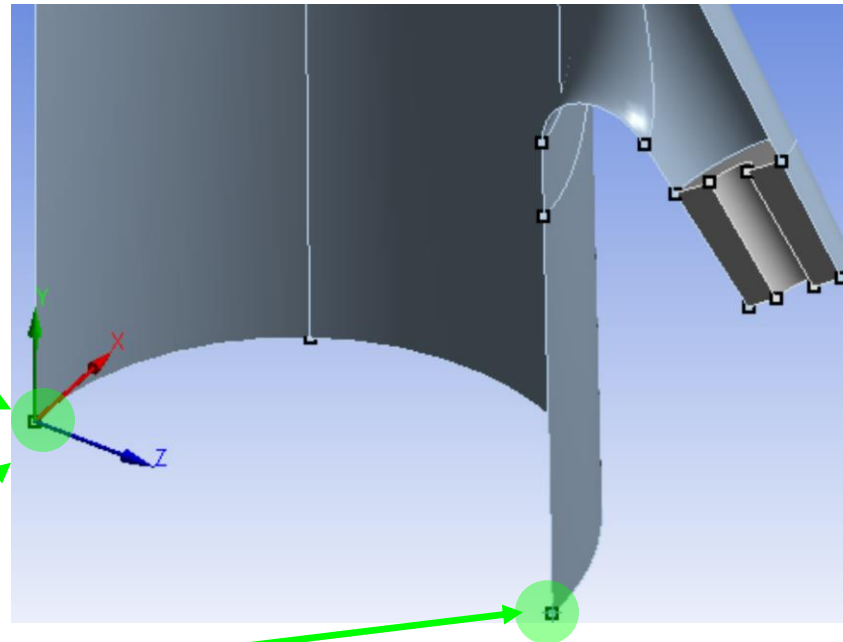
Filter: Name

- Geometry
- Coordinate Systems
 - Global Coordinate System
 - Coordinate System Symmetry
- Connections
- Mesh

Details of "Coordinate System Symmetry"

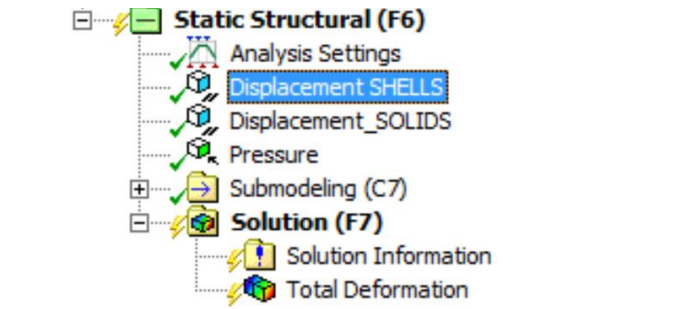
Definition	
Type	Cartesian
Coordinate System	Program Controlled
Suppressed	No
Origin	
Define By	Geometry Selection
Geometry	Click to Change
Origin X	-169. mm
Origin Y	13500 mm
Origin Z	-292.72 mm
Principal Axis	
Axis	Y
Define By	Geometry Selection
Geometry	Click to Change
Orientation About Principal Axis	
Axis	Z
Define By	Geometry Selection
Geometry	Click to Change

Add a coordinate system to the model and orient it to have one axis to be normal to the cutting plane of the symmetric model. We will use this to define symmetry boundary conditions.



13. Workshop Submodelling

Define symmetry boundary conditions using displacement objects in Solution (one for faces and one for edges)

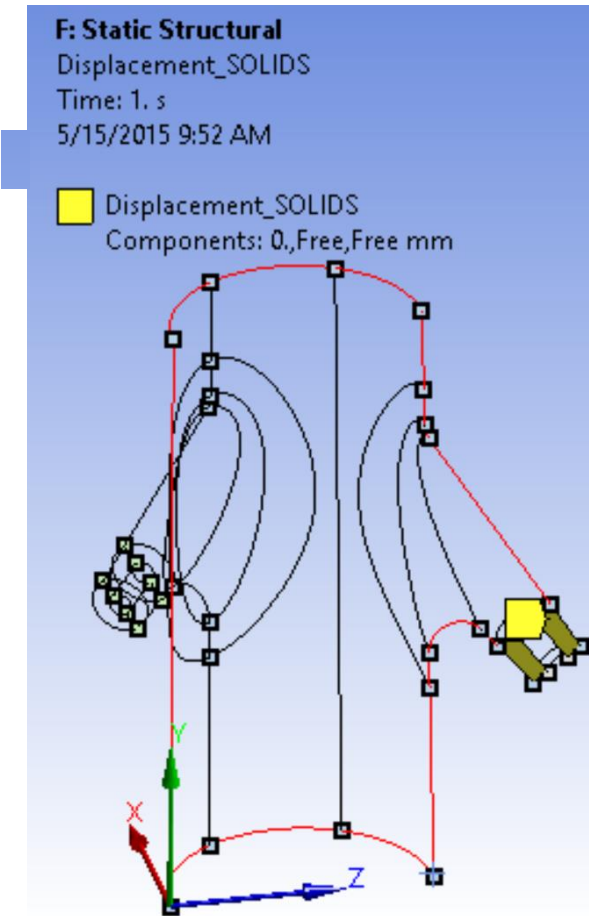
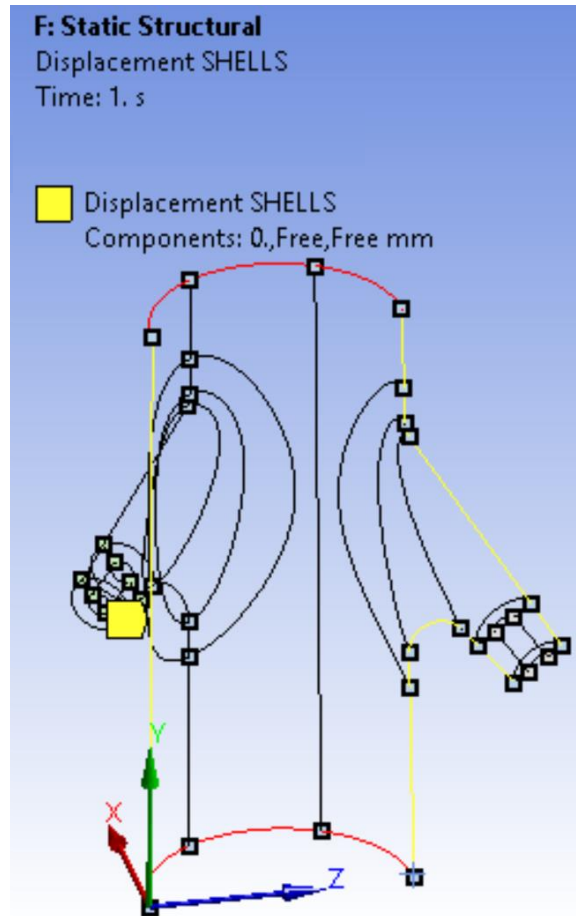


Details of "Displacement SHELLS"

Scope	
Scoping Method	Geometry Selection
Geometry	11 Edges
Definition	
Type	Displacement
Define By	Components
Coordinate System	Coordinate System Symmetry
<input type="checkbox"/> X Component	0. mm (ramped)
Y Component	Free
Z Component	Free
Suppressed	No

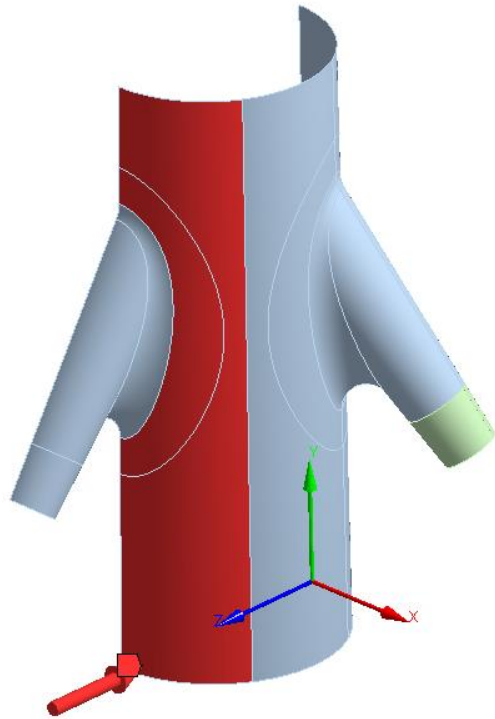
Details of "Displacement_SOLIDS"

Scope	
Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
Type	Displacement
Define By	Components
Coordinate System	Coordinate System Symmetry
<input type="checkbox"/> X Component	0. mm (ramped)
Y Component	Free
Z Component	Free
Suppressed	No



13. Workshop Submodelling

To model same loading as in global model, we have to add the pressure load too in the Solution. Use the newly created COS along with the shown settings to add pressure load



Details of "Pressure"	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	2 Faces
[-] Definition	
Type	Pressure
Define By	Components
Coordinate System	Coordinate System Symmetry
<input type="checkbox"/> X Component	0. MPa (ramped)
<input type="checkbox"/> Y Component	0. MPa (ramped)
<input type="checkbox"/> Z Component	-5.e-003 MPa (ramped)
Suppressed	No

13. Workshop Submodelling

The image displays the ANSYS Workbench interface for a submodelling analysis. The left pane shows the Project Outline with the following structure:

- Model (E4, F4, G4)
 - Geometry
 - Coordinate Systems
 - Connections
 - Contacts
 - Bonded - insert1 To Surface Body
 - Bonded - Multiple To Multiple
 - Mesh
 - Imported Layered Section
 - Named Selections
 - Static Structural (F6)
 - Analysis Settings
 - Displacement SHELLS
 - Displacement SOLIDS
 - Pressure
 - Submodeling (C7)
 - Solution (F7)
 - Solution Information

The bottom pane shows the details of the "Bonded - Multiple To Multiple" contact definition:

Details of "Bonded - Multiple To Multiple"	
Scope	
Scoping Method	Geometry Selection
Contact	2 Faces
Target	2 Faces
Contact Bodies	Multiple
Target Bodies	Multiple
Target Shell Face	Bottom
Shell Thickness Effect	No
Definition	
Type	Bonded
Scope Mode	Manual
Behavior	Program Controlled

The central 3D model shows a mechanical assembly with a red cylindrical component and a blue curved component. A red square indicates the contact area. The bottom scale bar shows dimensions from 0.00 to 500.00 (mm).

The right pane shows two views of the contact area:

- Contact Body View**: A 3D view of the red cylindrical component with a green circular contact area on its top surface. A scale bar shows dimensions from 0.00 to 90.00.
- Target Body View**: A 3D view of the blue curved component with a blue circular contact area on its inner surface.

Finally check contact definitions on inserts. Make sure to select correct shell face.

13. Workshop Submodelling

The screenshot displays the ANSYS Workbench interface with the following components:

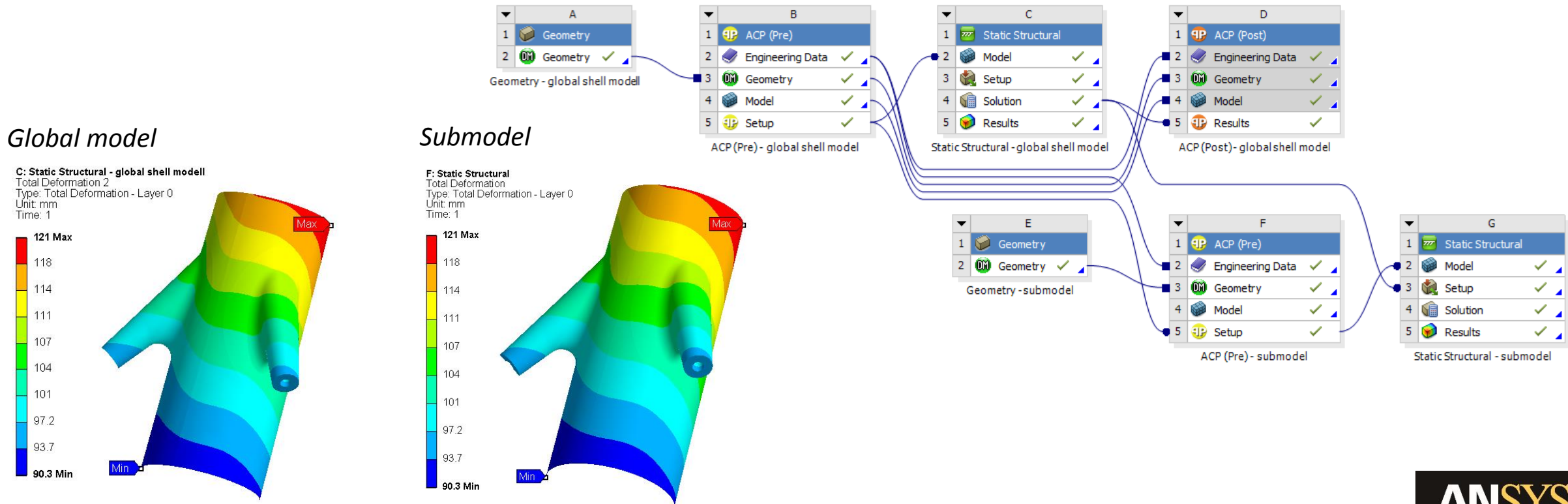
- Outline:** A tree view on the left showing the project hierarchy. The 'Contacts' folder is expanded, showing 'Bonded - insert1 To Surface Body' and 'Bonded - Multiple To Multiple'. The 'Static Structural (F6)' folder is also expanded, showing 'Analysis Settings', 'Displacement SHELLS', 'Displacement SOLIDS', 'Pressure', 'Submodeling (C7)', and 'Solution (F7)'.
- Details of "Bonded - insert1 To Surface Body":** A table on the left provides specific settings for the contact definition.
- Main View:** A 3D model of a mechanical part with a red insert and a blue surface body. A scale bar at the bottom indicates dimensions from 0.00 to 500.00 mm.
- Contact Body View:** A zoomed-in view of the contact area, showing the red insert and blue surface body. A scale bar at the bottom indicates dimensions from 0.00 to 80.00 mm.
- Target Body View:** A zoomed-in view of the target body, showing the blue surface body. A scale bar at the bottom indicates dimensions from 0.00 to 80.00 mm.

Details of "Bonded - insert1 To Surface Body"	
Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	insert1
Target Bodies	Surface Body
Target Shell Face	Bottom
Shell Thickness Effect	No
Definition	
Type	Bonded
Scope Mode	Manual
Behavior	Program Controlled
Trim Contact	Program Controlled
Suppressed	No

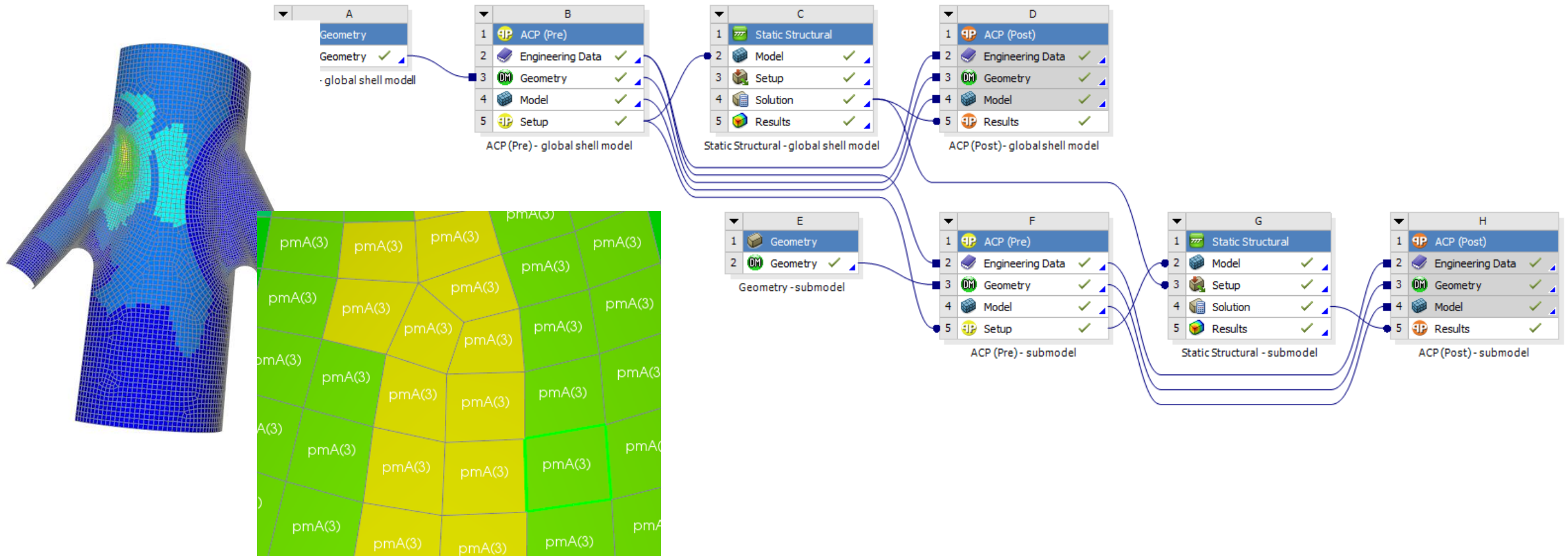
Finally check contact definitions on inserts. Make sure to select correct shell face.

13. Workshop Submodelling

- Update project schematic and solve Submodel.
- Check deformation results inside Mechanical to validate calculation



13. Workshop Submodelling



By adding a ACP-Post object, you can evaluate composite specific result as usual.