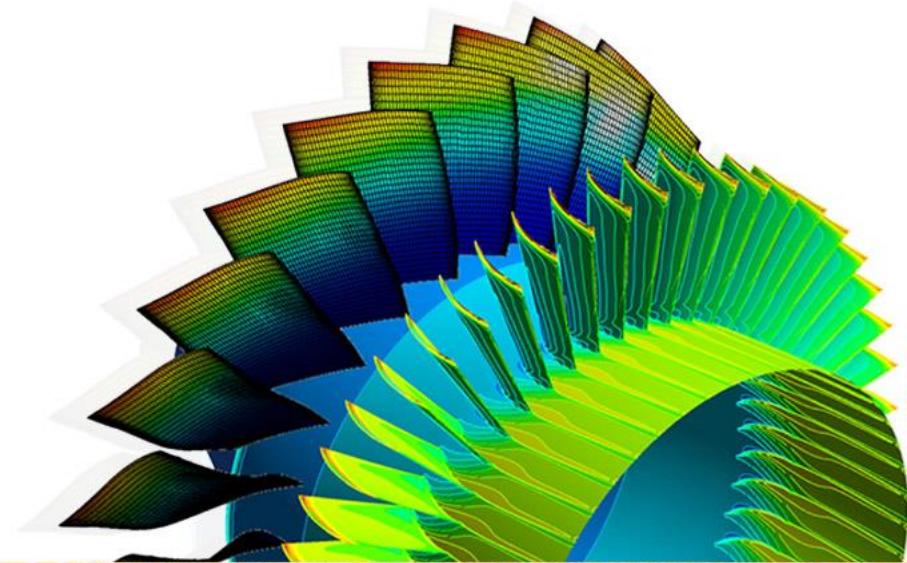




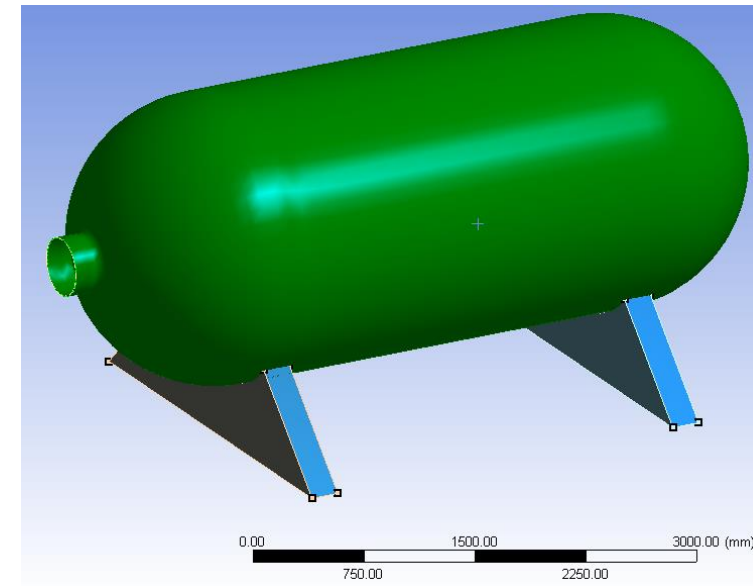
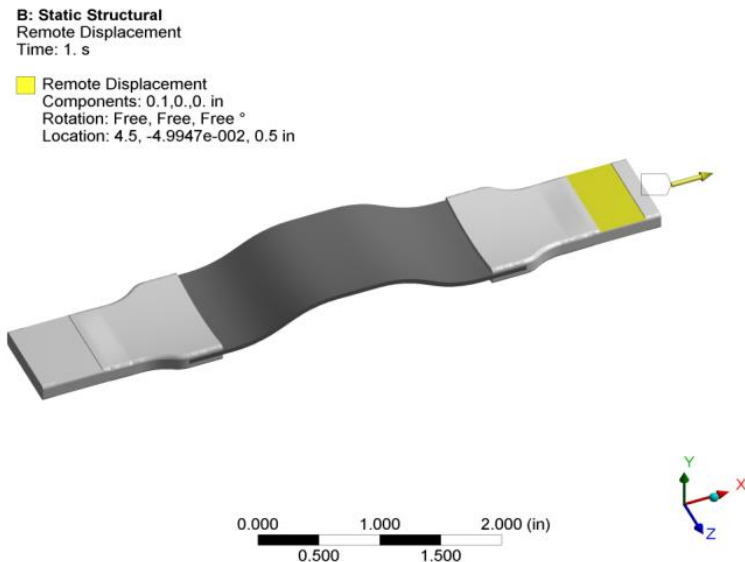
# ANSYS Composite PrepPost 19.0

Workshop 07.1 – Solid Modeling



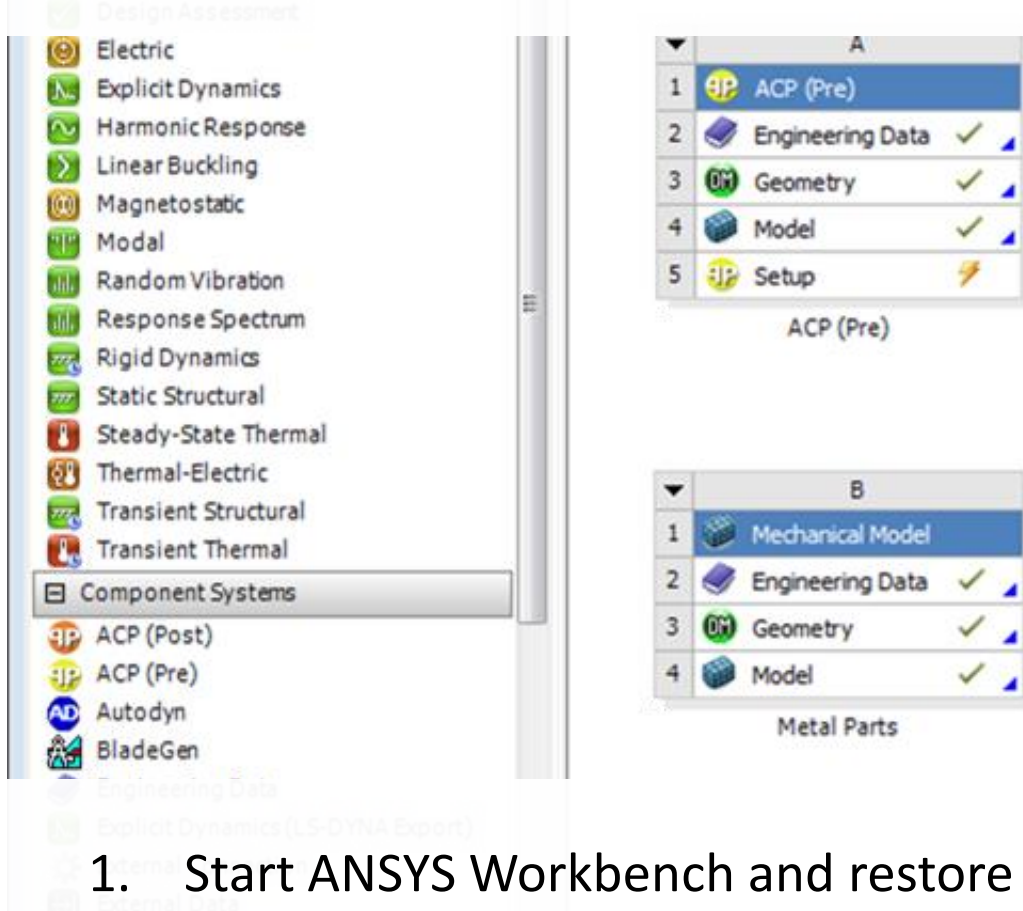
# 7. Workshop Solid Modeling

- This workshop will cover the solid model workflow using two simple composite assemblies
- The composite part will be combined with metallic parts into an assembly, and then analyzed



# 7.1 Workshop Solid Modeling

## Start ANSYS Workbench and Restore Archive

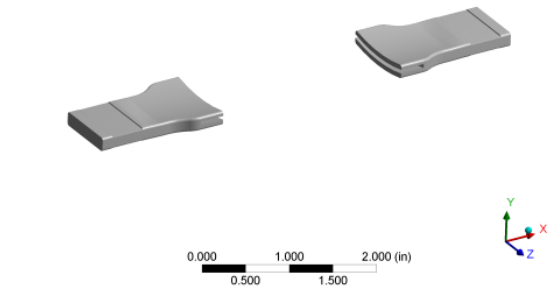
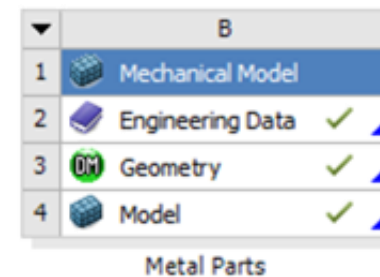
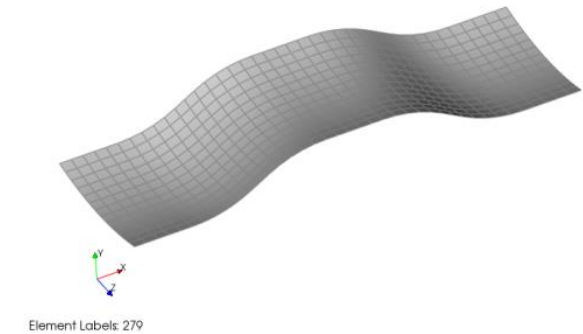
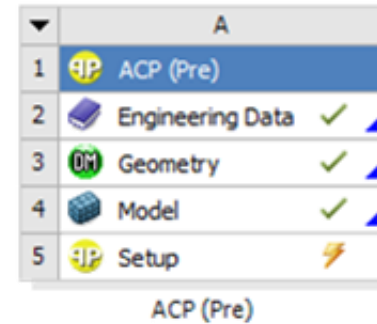


1. Start ANSYS Workbench and restore Archive  
*Solid\_Modeling\_FROM\_START\_19.0.wbpz*
2. Save the Workbench project

# 7.1 Workshop Solid Modeling

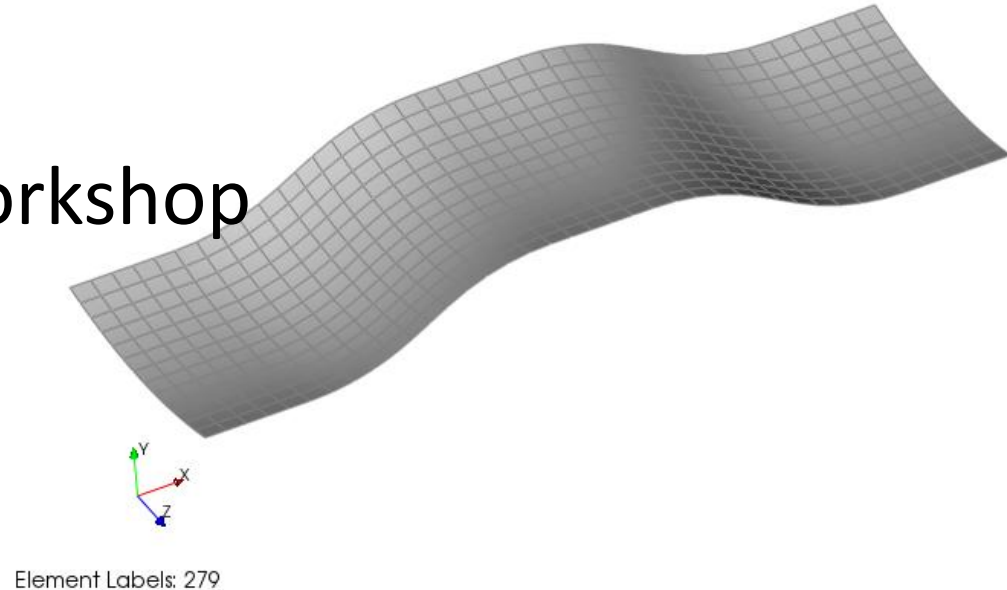
The archived project contains two systems:

- ACP(Pre) → The system with the composite part
- Metal Parts → The system with the non composite metal parts of the assembly



# 7.1 Workshop Solid Modeling

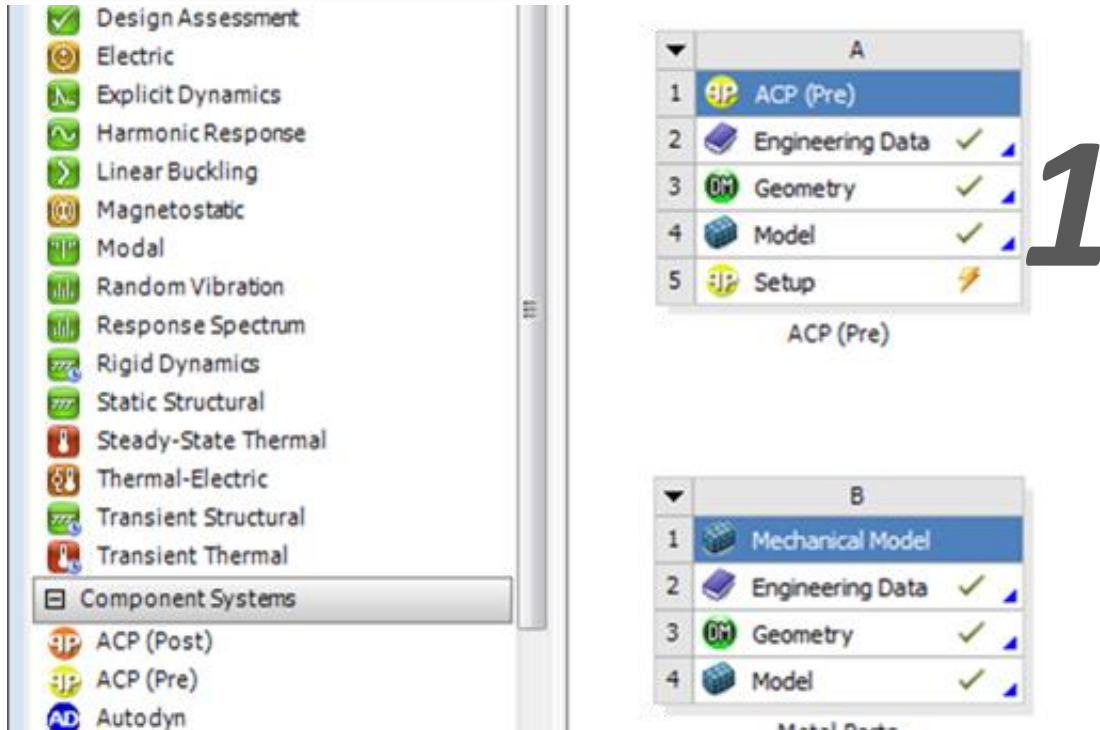
- The composite part and the metal parts will be assembled later
- The solid composite model is generated based on a shell model and extruded to a solid model
- This is the first step of the workshop





# 7.1 Workshop Solid Modeling

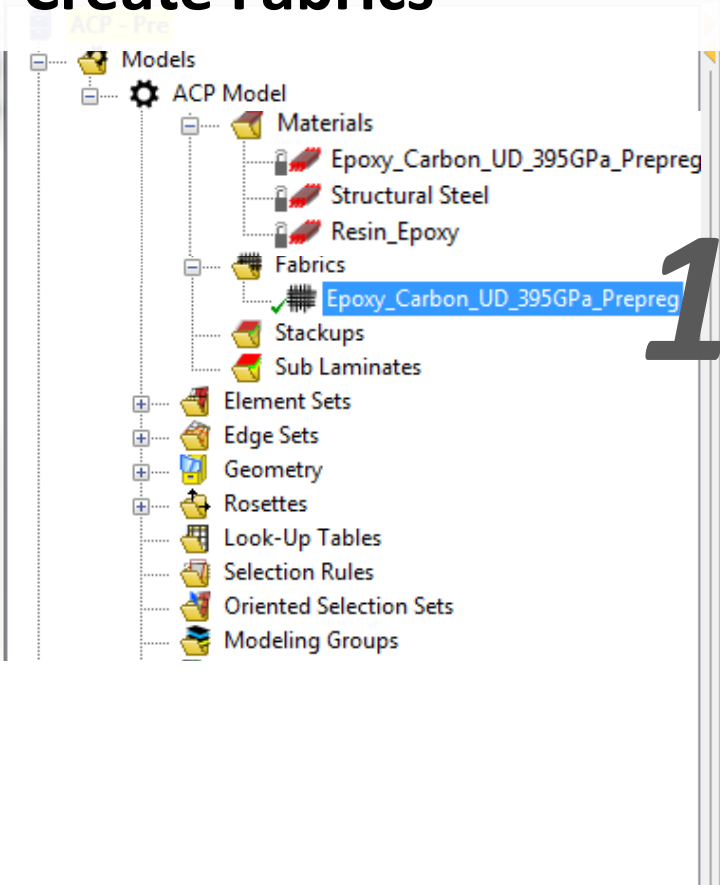
## Open ANSYS Composite PrepPost Model



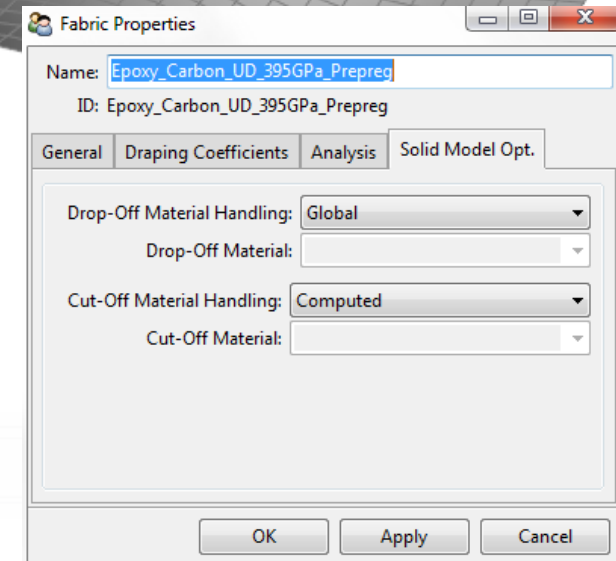
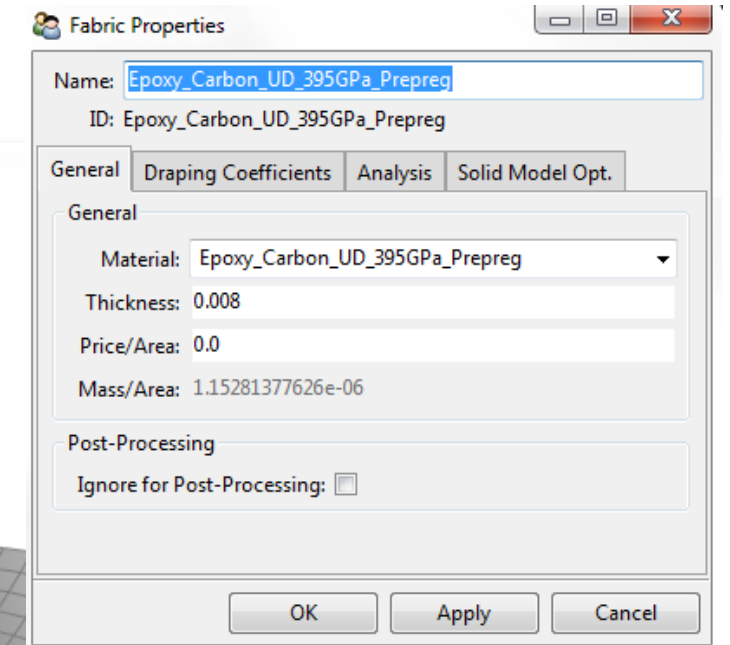
1. Open ANSYS Composite PrepPost; The shell mesh and materials are already defined

# 7.1 Workshop Solid Modeling

## Create Fabrics

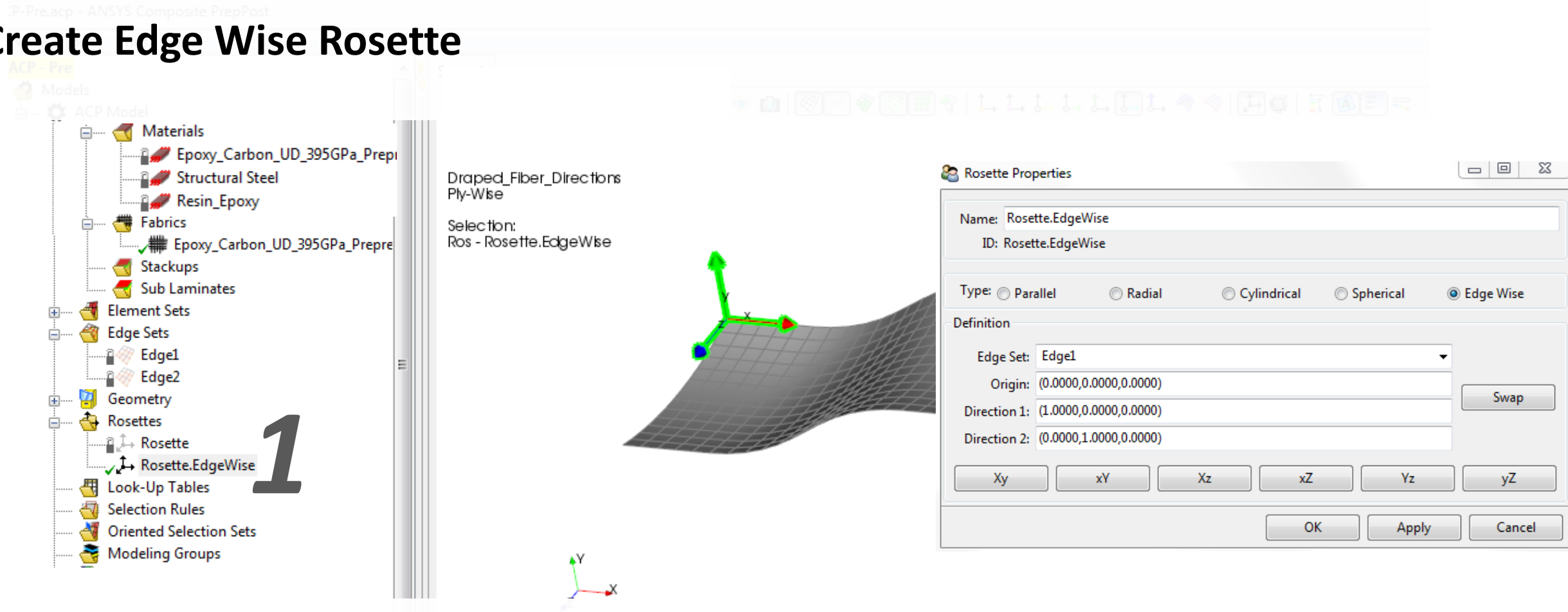


1. Create a new fabric using Epoxy Carbon Prepreg material and a thickness of 0.008 in



# 7.1 Workshop Solid Modeling

## Create Edge Wise Rosette



1. We will use an edge wise rosette to specify the fiber reference direction. Create a new edge wise rosette based on *Edge1*. This edge has been defined as named selection in the model.



# 7.1 Workshop Solid Modeling

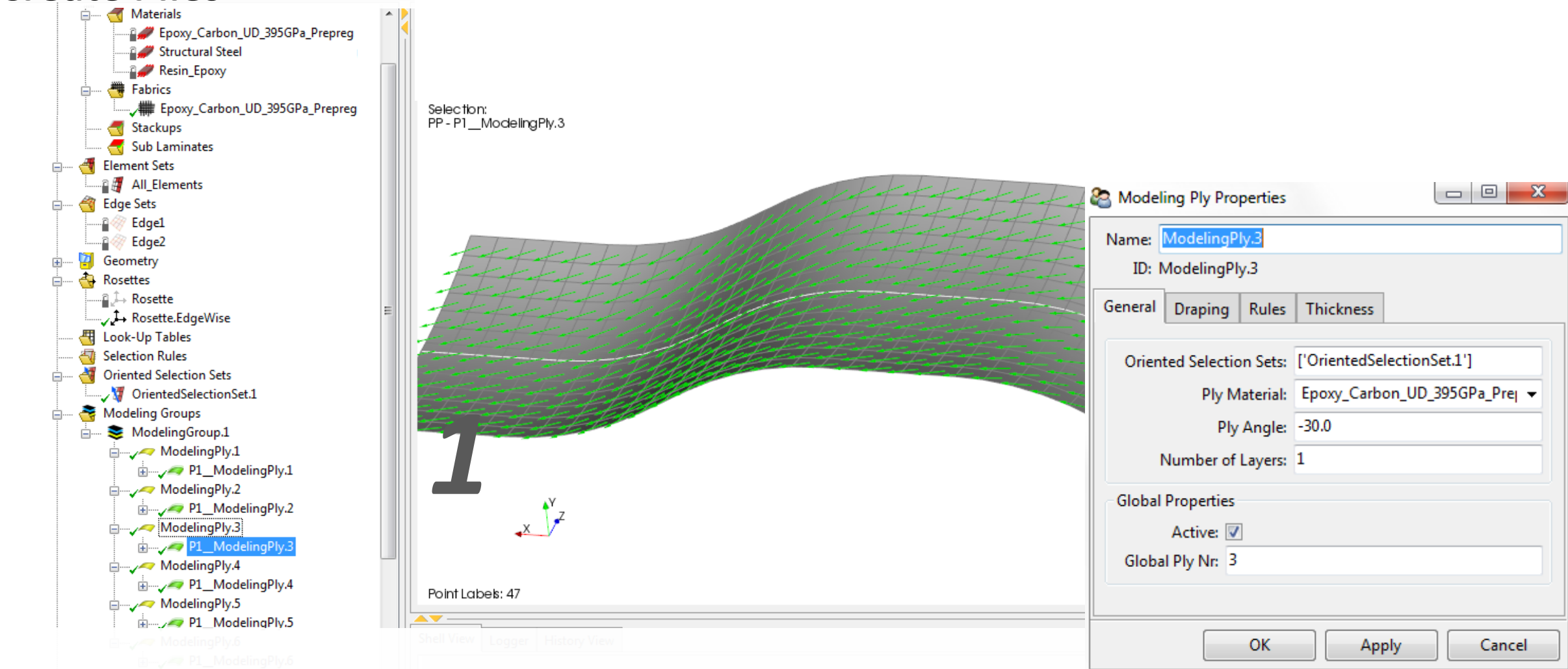
## Create Oriented Selection Set

1

1. Create a new Oriented Selection Set based on all elements. Use the edge wise rosette created in the previous step and an orientations direction as shown above.

# 7.1 Workshop Solid Modeling

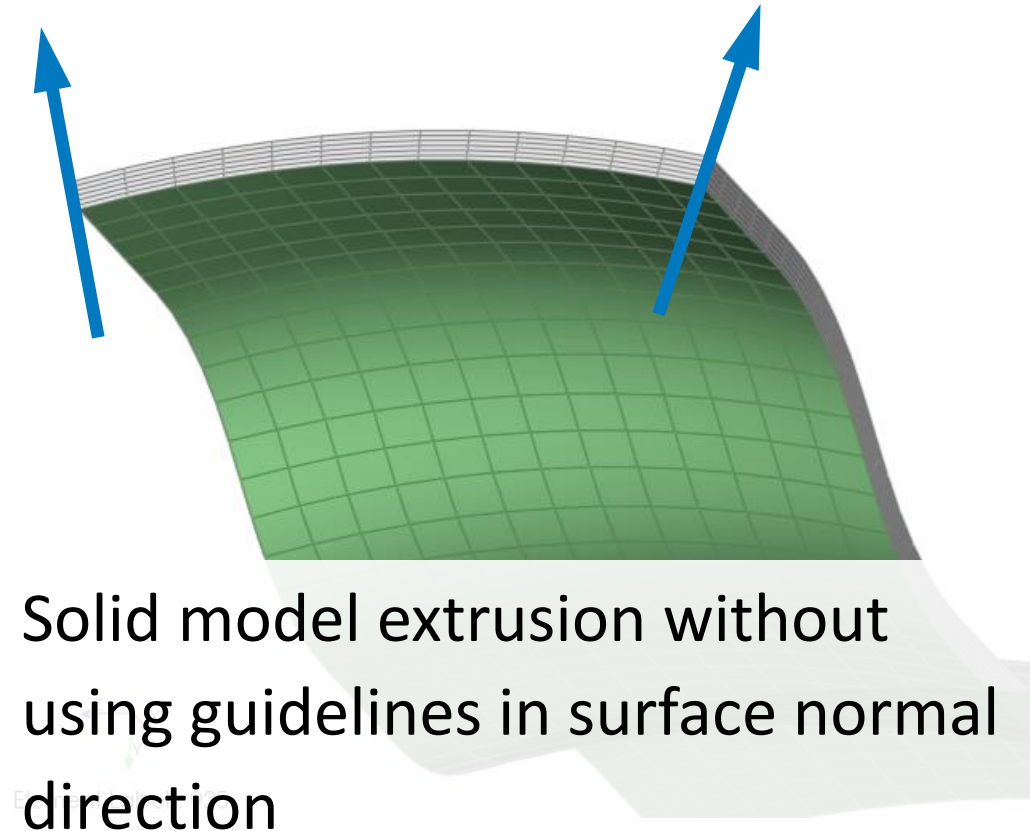
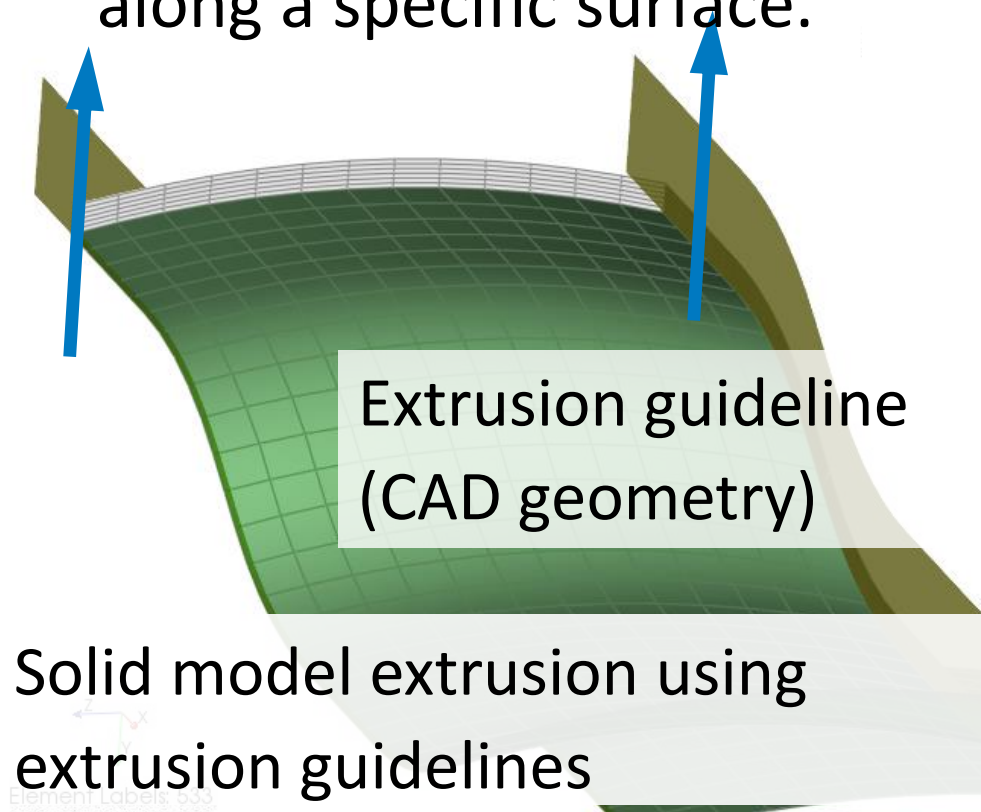
## Create Plies



1. Create a new ply group
2. Create six new layers (0°, 0°, -30°, 30°, 0°, 0°)

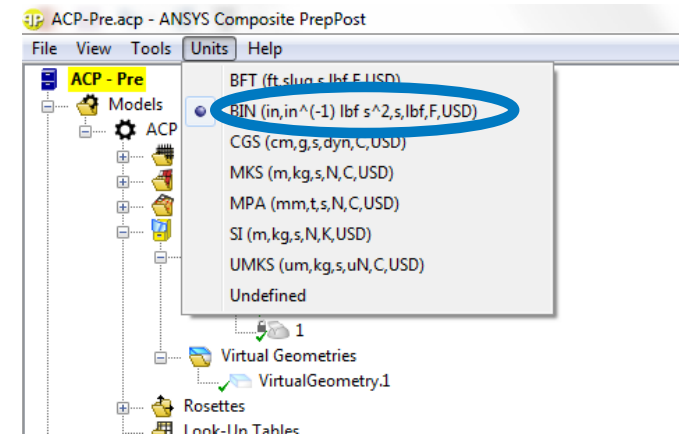
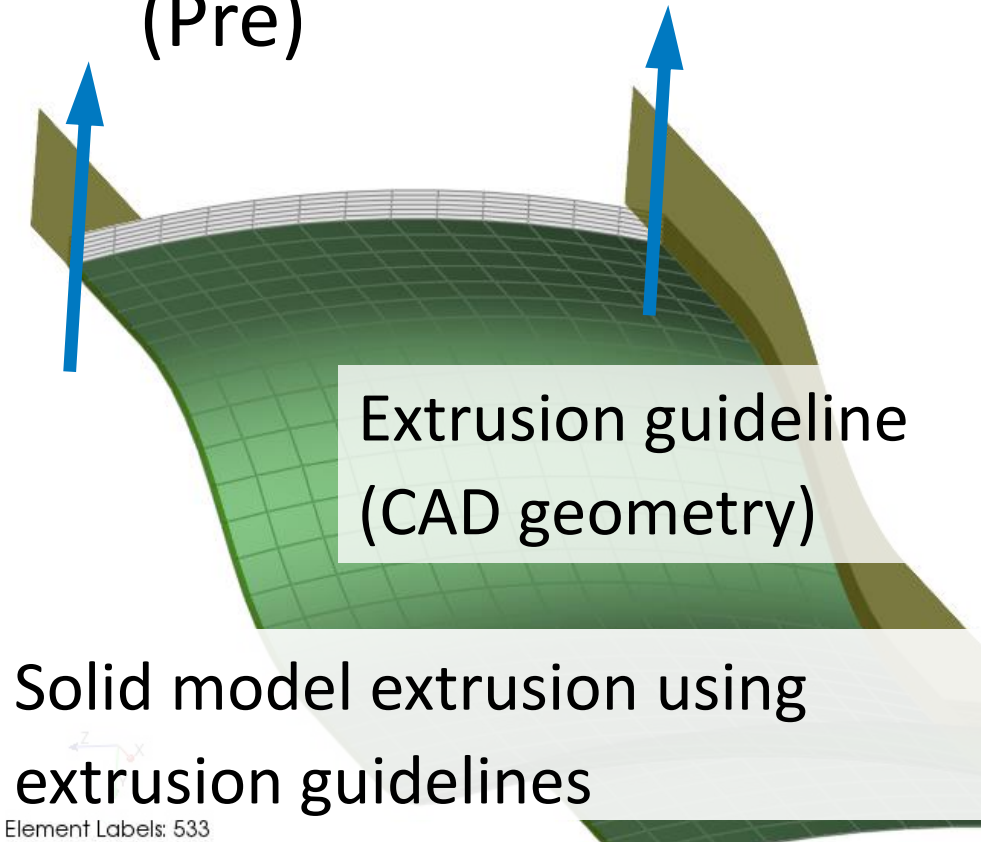
# 7.1 Workshop Solid Modeling

- When the composite layup is defined the solid model is created by extrusion.
- We will use extrusion guides imported as CAD file to extrude the model along a specific surface.



# 7.1 Workshop Solid Modeling

- The unit system of the CAD file imported for the extrusion guide is US Customary (inch), please change the unit system of ACP (Pre)





# 7.1 Workshop Solid Modeling

## Import CAD Geometry

The screenshot illustrates the process of importing CAD geometry into ANSYS Workbench. It shows three panels: the Component Systems toolbox, the Project Schematic, and the Import Geometry dialog box.

**1. Insert *Geometry* from the Component Systems**

The Component Systems toolbox on the left shows the **Geometry** component under the **Component Systems** category. A large arrow labeled '1' points from this component to the Project Schematic.

**2. Link *Geometry* to *Setup* of ACP (Pre)**

The Project Schematic shows three models: A, B, and C. Model A has a **Geometry** component. Model B has an **ACP (Pre)** component. A blue line connects the **Geometry** component of Model A to the **Setup** component of Model B, indicating a link.

**3. Import the cad file *extrusion\_guide.stp*, the step file we will import can be found in the workshop folder.**

The **Import Geometry** dialog box is open, showing the **extrusion\_guide.stp** file selected in the **Browse...** list.

**4. Update ACP (Pre) setup and return to ACP (Pre)**



# 7.1 Workshop Solid Modeling

## Import CAD Geometry

1

2

3

1. The CAD file imported before is present in CAD Geometries
2. Create a new virtual geometry (Right mouse button on Virtual Geometries  
→ Create Virtual Geometry)
3. Check whether your unit system is US Customary (in, lbf, s), the unit system of the imported CAD is US Customary

Virtual Geometry Properties

Name: CADGeometry.1  
ID: CADGeometry.1  
Characteristic: Surface  
['extrusion\_guide.stp']  
Sub Shapes:

OK Apply Cancel

Unit System: BIN (in,in<sup>-1</sup> lbf s<sup>2</sup>,s,lbf,F,USD)

# 7.1 Workshop Solid Modeling

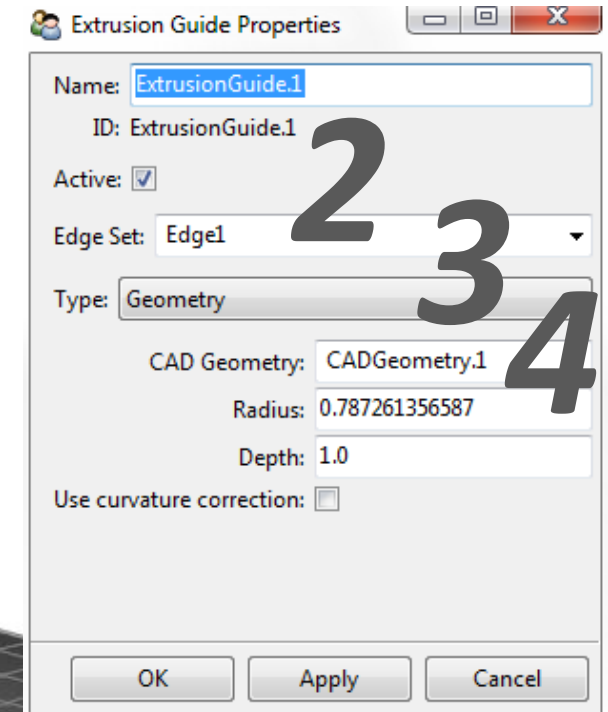
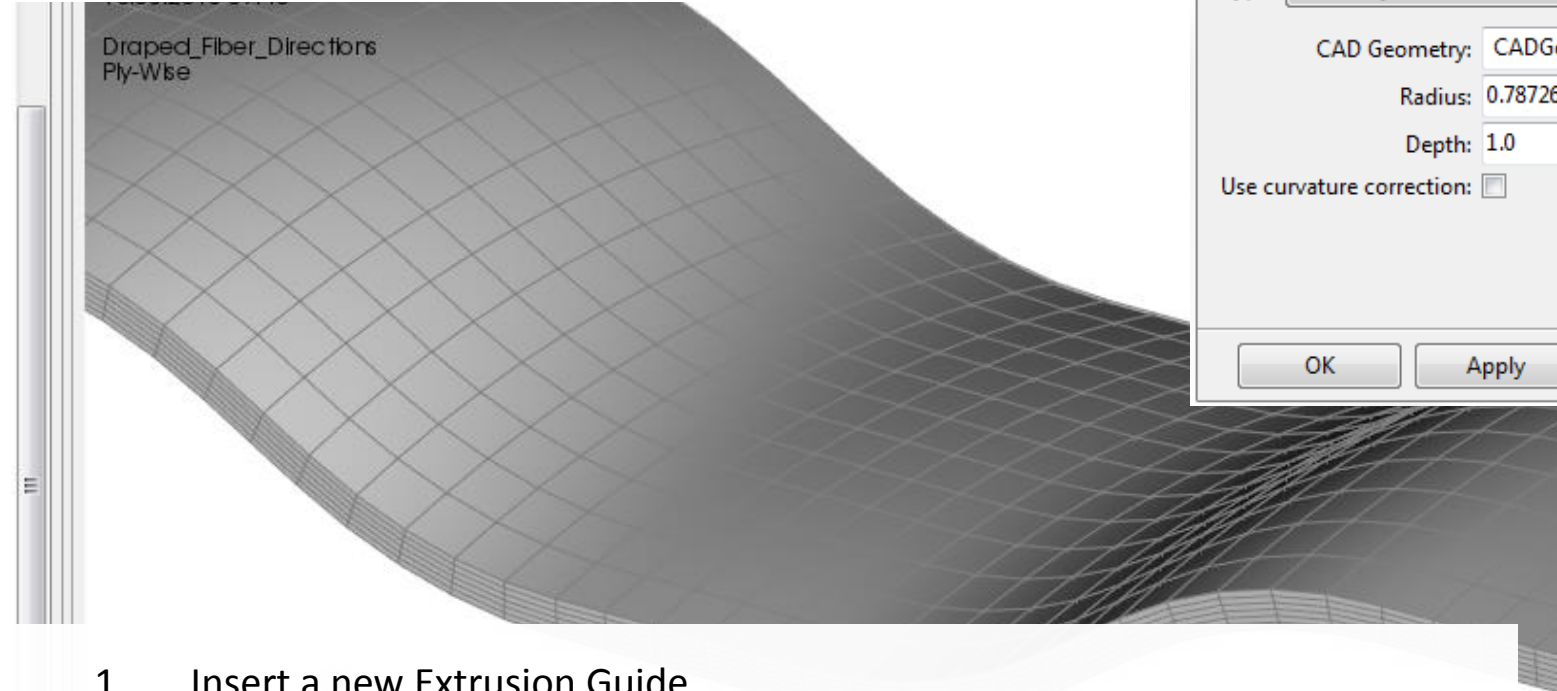
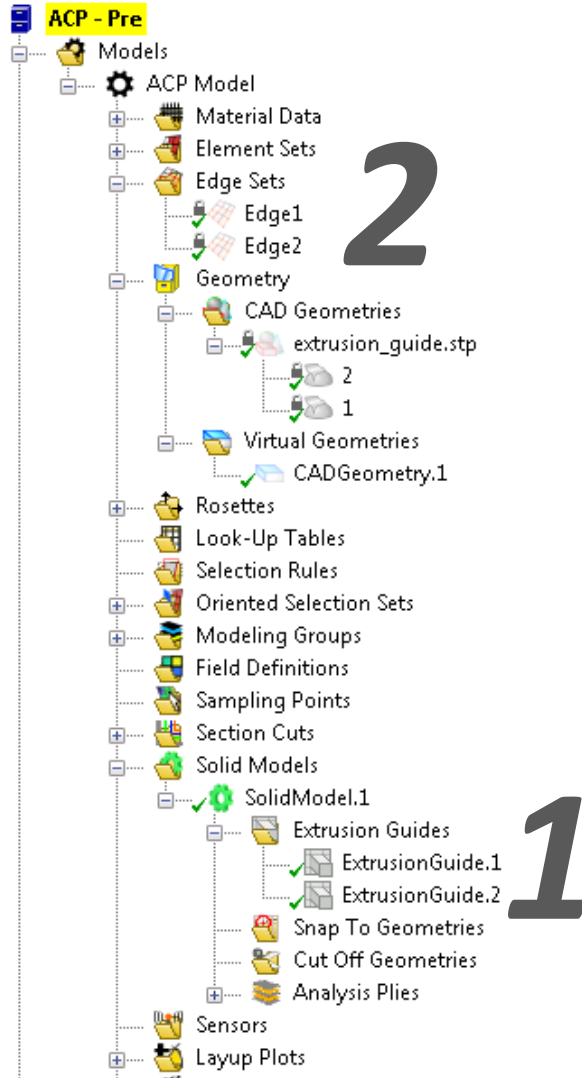
## Create a Solid Model

The screenshot displays the ANSYS Workbench interface. On the left, the Project Schematic shows a tree structure with 'Solid Models' expanded, highlighting 'SolidModel.1' with a green checkmark. A large number '1' is placed next to this tree. The main area shows a 3D model of a curved plate with a mesh. A large number '2' is placed over the 'Element Sets' field in the 'Solid Model Properties' dialog, which is set to '["All\_Elements"]'. A large number '3' is placed over the 'Extrusion Method' dropdown, which is set to 'Analysis Ply Wise'. A large number '4' is placed over the 'Global Drop-Off Material' dropdown, which is set to 'Resin\_Epoxy'. The 'Solid Model Properties' dialog is open, showing the 'Export' tab. The 'Active' checkbox is checked. The 'Extrusion Properties' section includes 'Extrusion Method' (Analysis Ply Wise), 'Max. Element Thickness' (1.0), 'Start Ply Groups at' (empty), 'Connect Butt-Joined Plies' (checked), and 'Drop-Off Method' (Inside Ply). The 'Offset Direction' is set to 'Shell Normal'. The 'Drop-Offs And Cut-Offs' section includes 'Write Degenerated Elements' (checked), 'Global Drop-Off Material' (Resin\_Epoxy), and 'Global Cut-Off Material' (empty). The 'Element Quality' section includes 'Delete Bad Elements' (checked) and 'Warping Limit' (0.4). The dialog has 'OK', 'Apply', and 'Cancel' buttons at the bottom.

1. Create a new Solid Model (Right Mouse button on Solid Models  
→ Create Solid Model)
2. Select All Elements in element sets
3. Select extrusion method Analysis Ply Wise
4. Select Resin Epoxy as Global Drop-Off Material

# 7.1 Workshop Solid Modeling

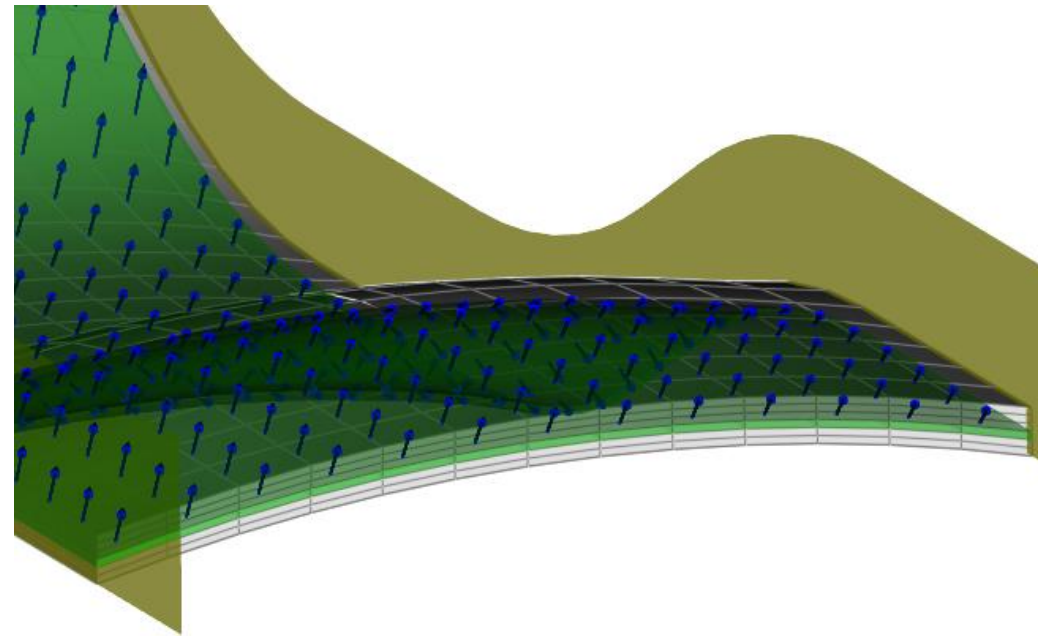
## Create a Solid Model



1. Insert a new Extrusion Guide
2. Select *Edge1* for the first extrusion guide
3. Select Type Geometry
4. Select the imported CAD geometry
5. Repeat Steps 1-4 for Edge Set *Edge2* using same geometry

## 7.1 Workshop Solid Modeling

- The solid model has been extruded by ANSYS Composite PrepPost and can now be used for different analyses.
- In the next steps we will assemble composite and metal parts within ANSYS Mechanical.

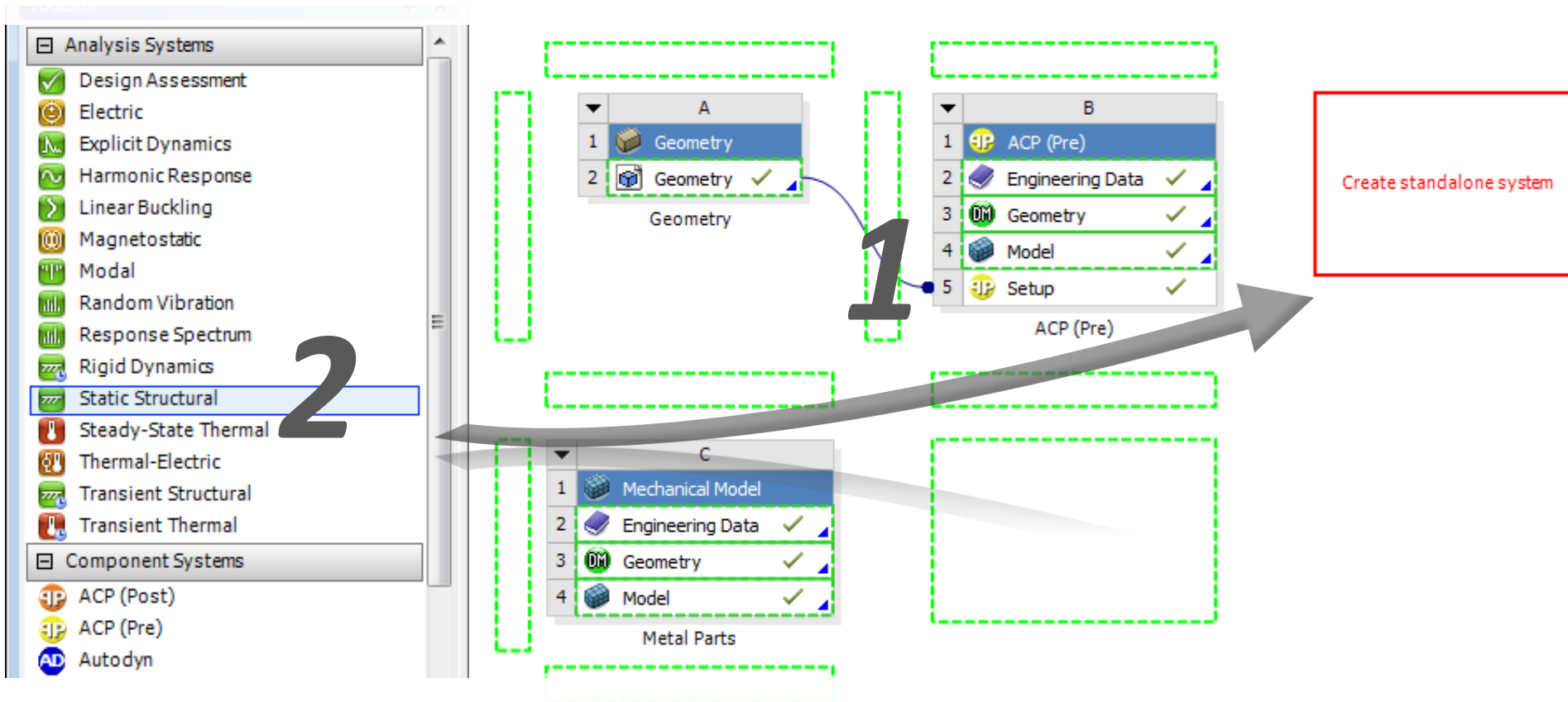


Element Labels: 3086



# 7.1 Workshop Solid Modeling

## Transfer composite and metal parts to ANSYS Mechanical

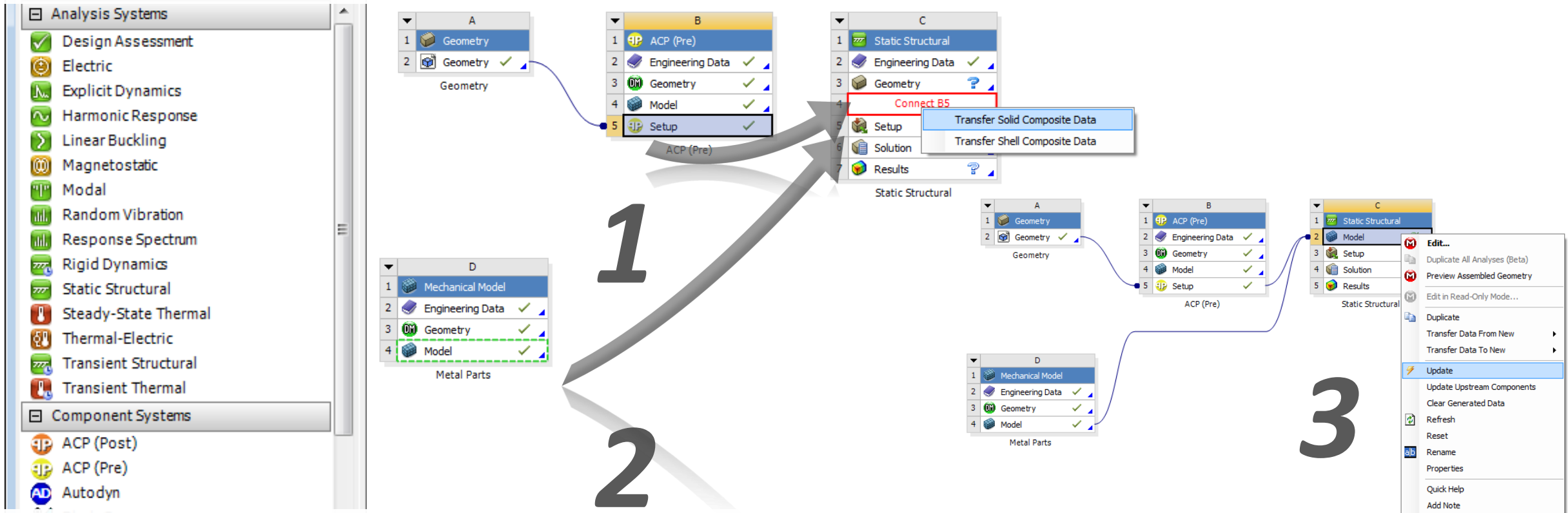


1. Update the ACP (Pre) Setup
2. Drag and Drop a new Static Structural system into the project schematic. Do not drop the new system onto any of the existing systems.



# 7.1 Workshop Solid Modeling

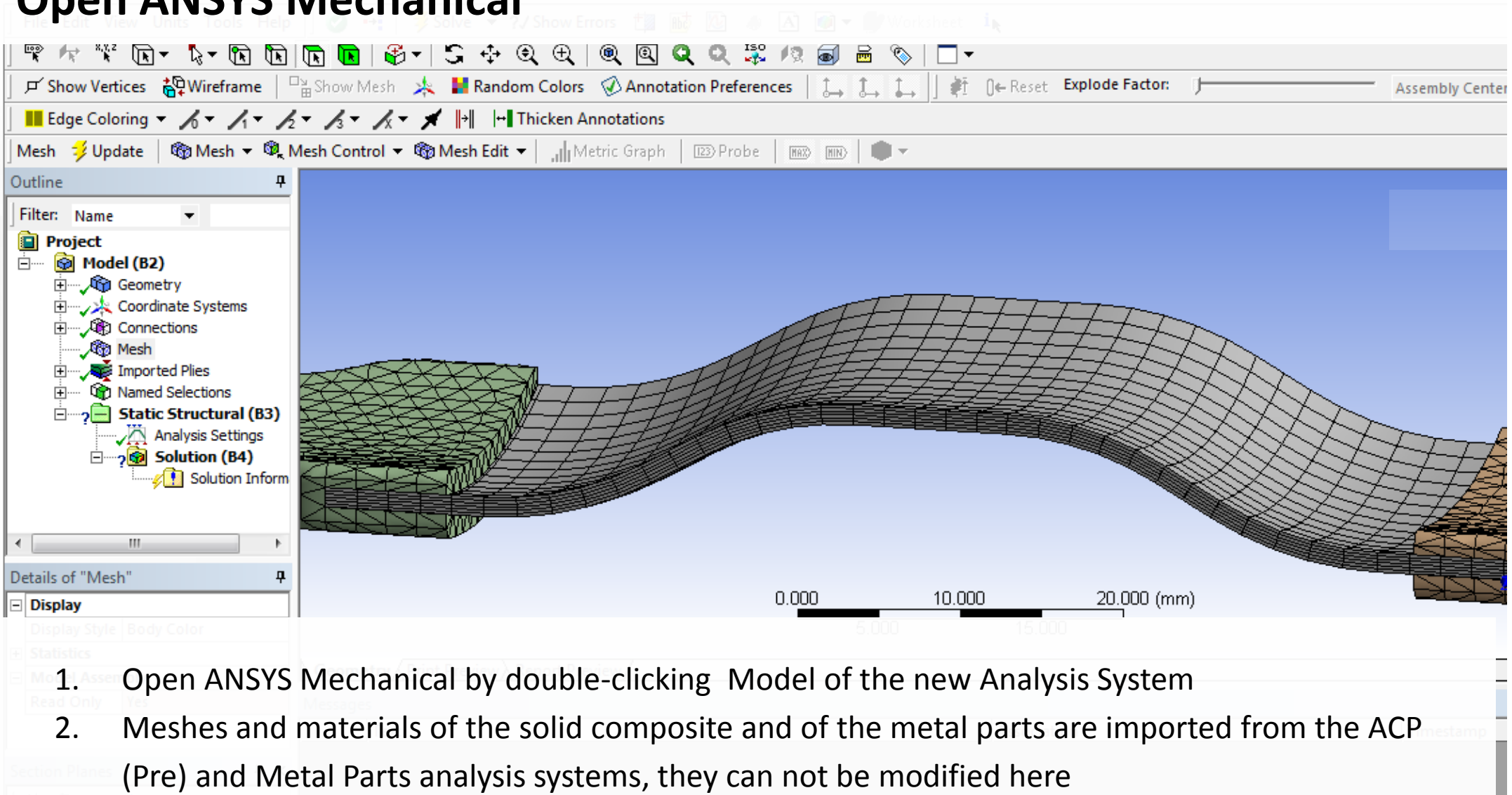
## Transfer composite and metal parts to ANSYS Mechanical



1. Drag and Drop **Setup of ACP (Pre)** onto **Model** of the new Static Structural (select Transfer Solid Composite Data)
2. Drag and Drop **Model of Metal Parts** onto **Model** of the new Static Structural
3. Update **Model** of the new Static Structural

# 7.1 Workshop Solid Modeling

## Open ANSYS Mechanical



1. Open ANSYS Mechanical by double-clicking Model of the new Analysis System

2. Meshes and materials of the solid composite and of the metal parts are imported from the ACP (Pre) and Metal Parts analysis systems, they can not be modified here

# 7.1 Workshop Solid Modeling

## Check Automatically Defined Contacts

The image displays the ANSYS Mechanical software interface. The main window shows a 3D model of a curved, red, wavy structure. A scale bar at the bottom indicates dimensions from 0.000 to 10.000 (mm). The left sidebar contains the Outline tree, which lists the model hierarchy: Project, Model (B2), Geometry, Coordinate Systems, Connections, Contacts, Contact Region, Contact Region 2, Contact Region 3, Contacts(Metal Parts), Mesh, Imported Plies, Named Selections, Static Structural (B3), and Analysis Settings. The Details panel for "Contact Region 2" is open, showing the following information:

Details of "Contact Region 2"	
Scoping Method	Geometry Selection
Contact	3 Faces
Target	3 Faces
Contact Bodies	SolidModel.1
Target Bodies	Solid(Metal Parts)
<b>Definition</b>	
Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	0.35517 mm

Below the Details panel, the Messages section shows "No Messages" and "No Selection". The right side of the interface features two inset views: "Contact Body View" and "Target Body View". The "Contact Body View" shows a red, wavy surface with a scale bar from 0.00 to 30.00. The "Target Body View" shows a green, flat surface with a scale bar from 0.00 to 20.00. The ANSYS 16.0 logo is visible in the top right corner of the main window.

1. The contact regions for the assembly have been automatically defined by ANSYS Mechanical. Please check both contact regions. Bonded contacts are used to connect the metal parts to the composite specimen.

# 7.1 Workshop Solid Modeling

## Apply Boundary Conditions

**1**

1. Apply a fixed support to one end and a remote displacement to the other end. Define a displacement of 0.01 inch in x-direction and fix all other displacements and rotations.

**Details of "Remote Displacement"**

Geometry	2 Faces
Coordinate System	Global Coordinate System
<input type="checkbox"/> X Coordinate	3.9131 in
<input type="checkbox"/> Y Coordinate	-4.9947e-002 in
<input type="checkbox"/> Z Coordinate	0.5 in
Location	Click to Change
<b>Definition</b>	
Type	Remote Displacement
<input type="checkbox"/> X Component	1.e-002 in (ramped)
<input type="checkbox"/> Y Component	0. in (ramped)
<input type="checkbox"/> Z Component	0. in (ramped)

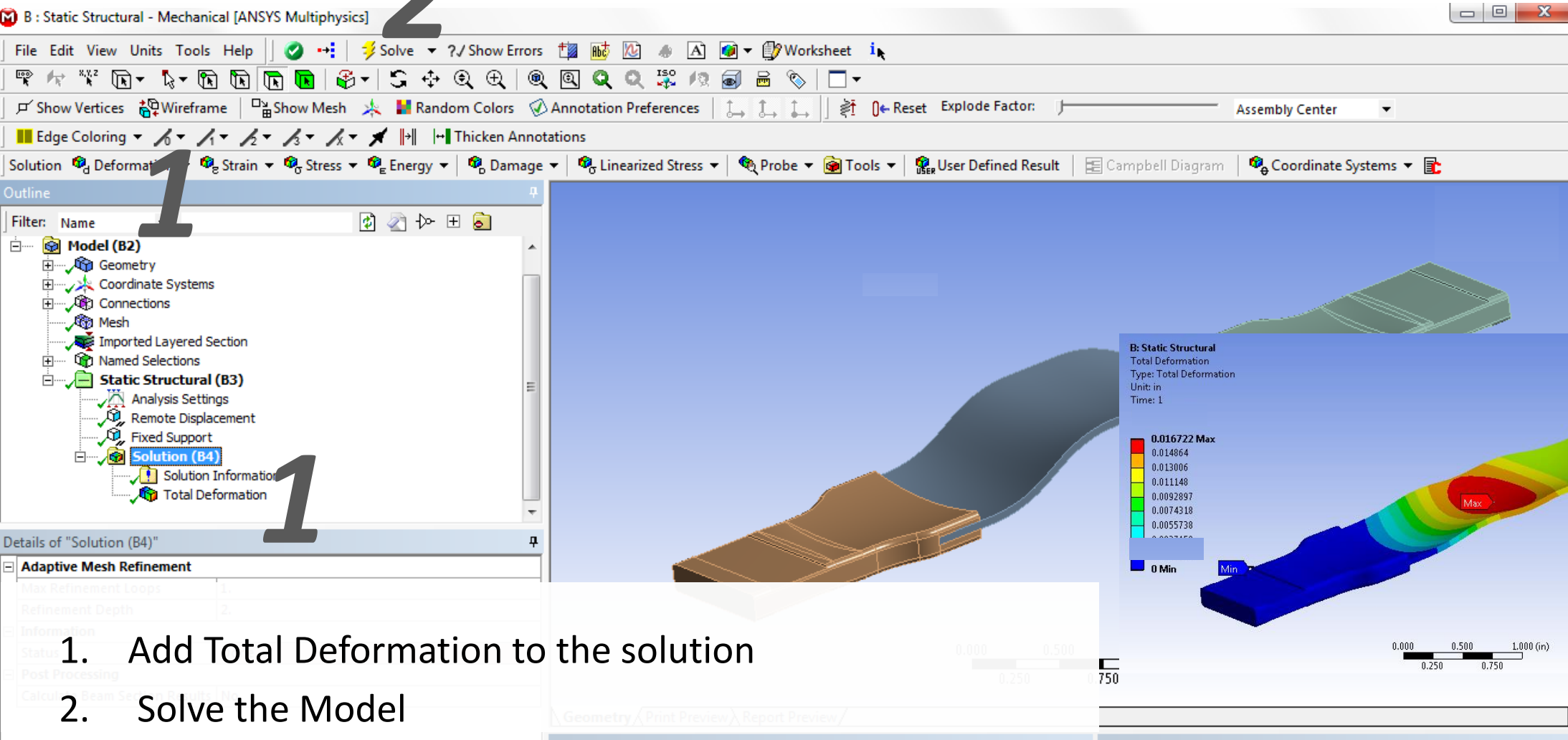
**Details of "Fixed Support"**

<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	2 Faces
<b>Definition</b>	
Type	Fixed Support
Suppressed	No



# 7.1 Workshop Solid Modeling

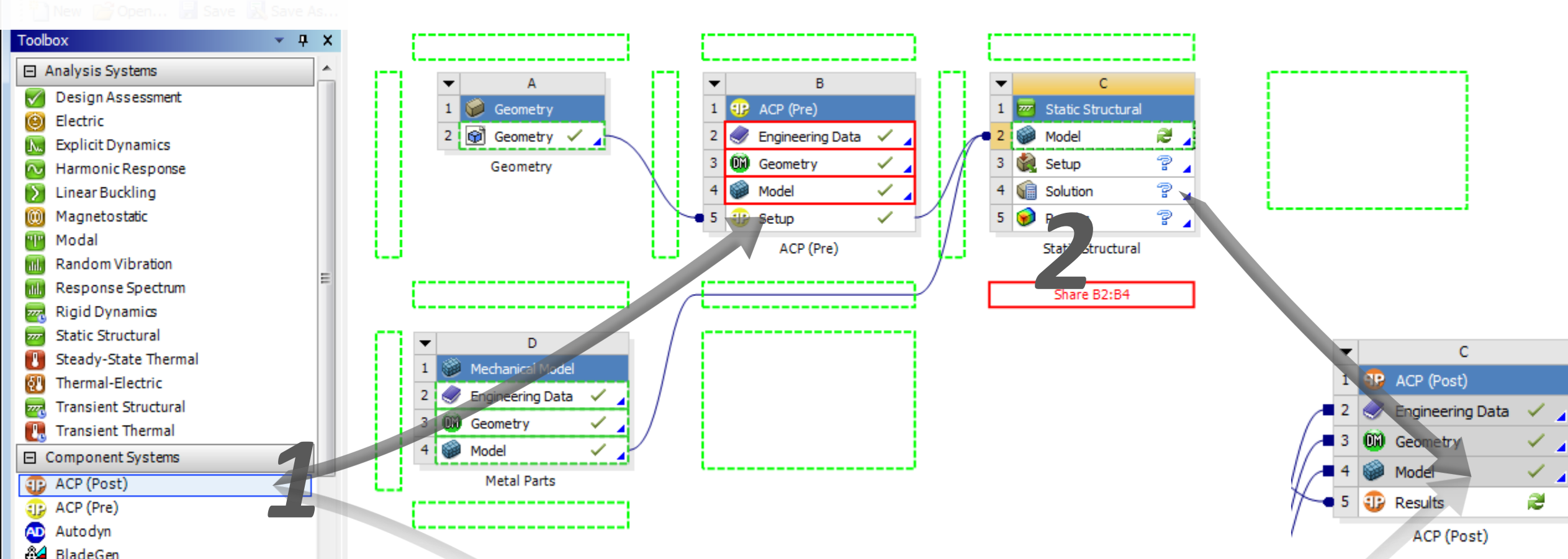
## Solve the Model **2**





# 7.1 Workshop Solid Modeling

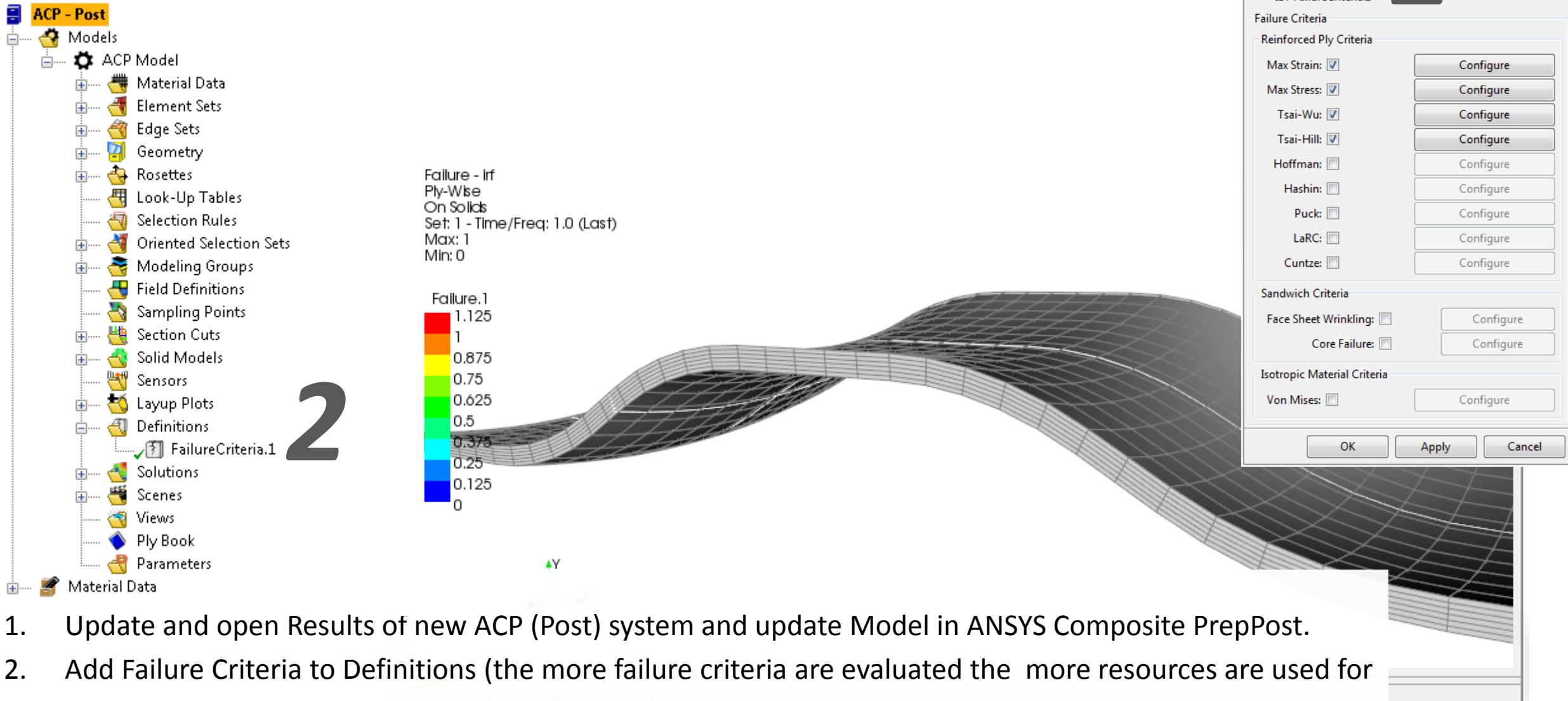
## Postprocessing in ANSYS Composite PrepPost



1. Drag and Drop ACP (Post) onto Model of ACP (Pre) system

2. Drag and Drop Solution of Static Structural analysis system onto Results of new ACP (Post) system

# 7.1 Workshop Solid Modeling



Failure - lrf  
Ply-Wise  
On Solids  
Set: 1 - Time/Freq: 1.0 (Last)  
Max: 1  
Min: 0

Failure.1  
1.125  
1  
0.875  
0.75  
0.625  
0.5  
0.375  
0.25  
0.125  
0

Failure Criteria Definition

Name: FailureCriteria.1  
ID: FailureCriteria.1

Failure Criteria

Reinforced Ply Criteria

Max Strain: ☒ Configure  
Max Stress: ☒ Configure  
Tsai-Wu: ☒ Configure  
Tsai-Hill: ☒ Configure  
Hoffman: ☐ Configure  
Hashin: ☐ Configure  
Puck: ☐ Configure  
LaRC: ☐ Configure  
Cuntze: ☐ Configure

Sandwich Criteria

Face Sheet Wrinkling: ☐ Configure  
Core Failure: ☐ Configure

Isotropic Material Criteria

Von Mises: ☐ Configure

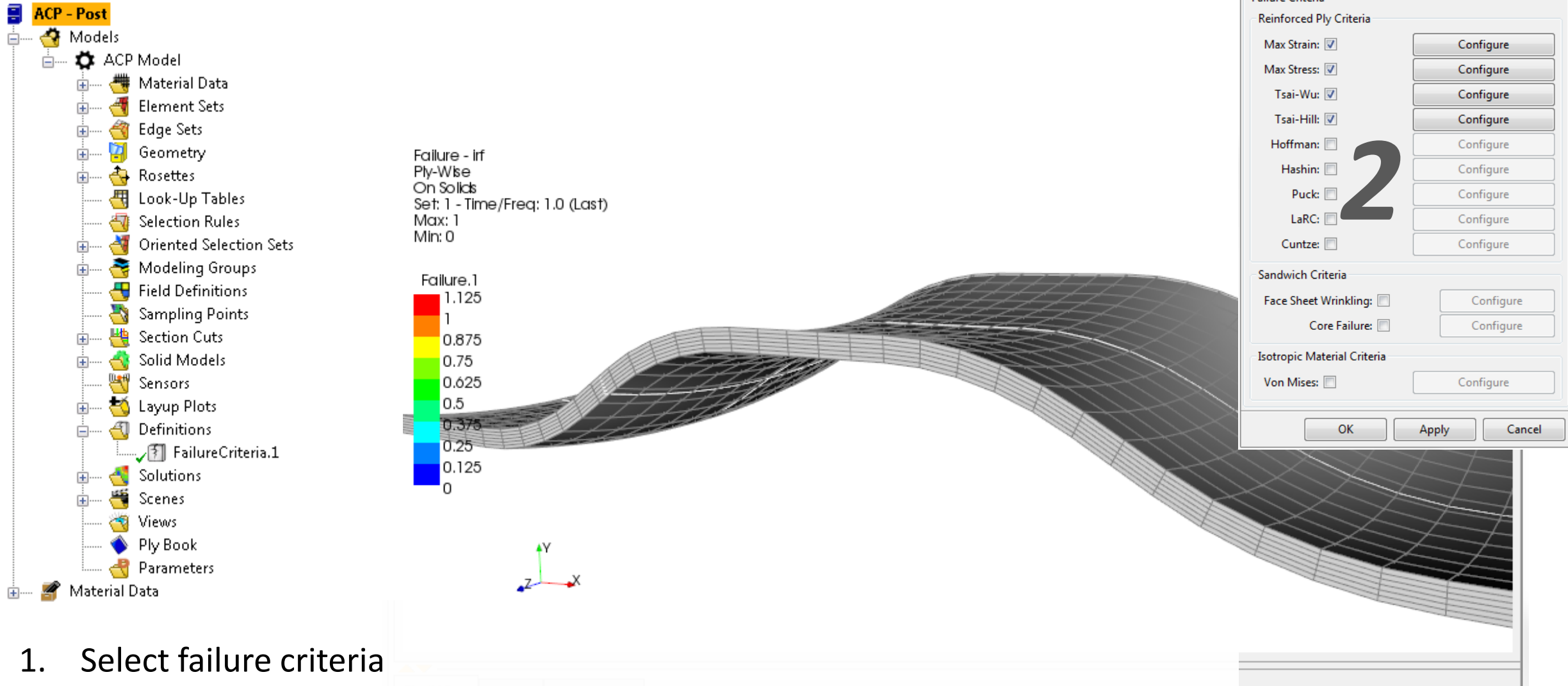
OK Apply Cancel

1. Update and open Results of new ACP (Post) system and update Model in ANSYS Composite PrepPost.
2. Add Failure Criteria to Definitions (the more failure criteria are evaluated the more resources are used for postprocessing)

## 7.1 Workshop Solid Modeling

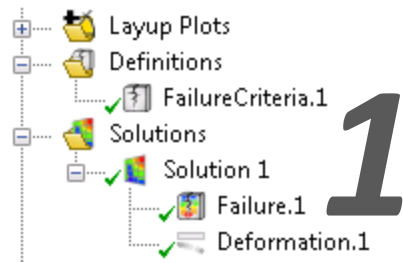
- Interlaminar strains and stresses for solid models are calculated by ANSYS and automatically available in ANSYS Composite PrepPost.
- The following failure criteria should be switched to 3D: Maximum Strain, Maximum Stress, Tsai-Wu, Tsai-Hill, Hashin, Puck, Cuntze
- All failure criteria can be evaluated at the same time  
(If hardware resources allow this, ANSYS Composite PrepPost utilizes multiple CPUs to evaluate failure criteria)

# 7.1 Workshop Solid Modeling

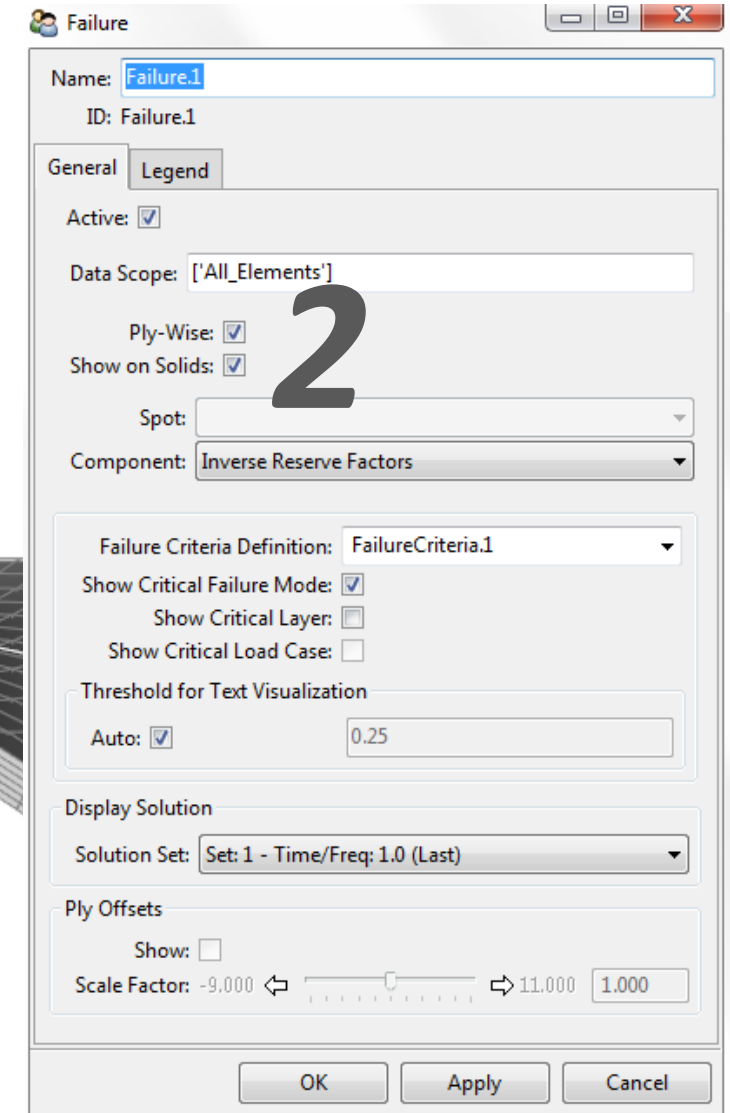
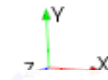
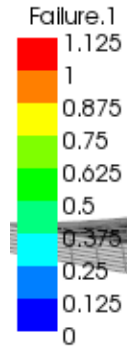


1. Select failure criteria
2. Configure failure criteria and switch failure criteria to 3D (if available)

# 7.1 Workshop Solid Modeling



Failure - lrf  
Ply-Wise  
On Solids  
Set: 1 - Time/Freq: 1.0 (Last)  
Max: 1  
Min: 0

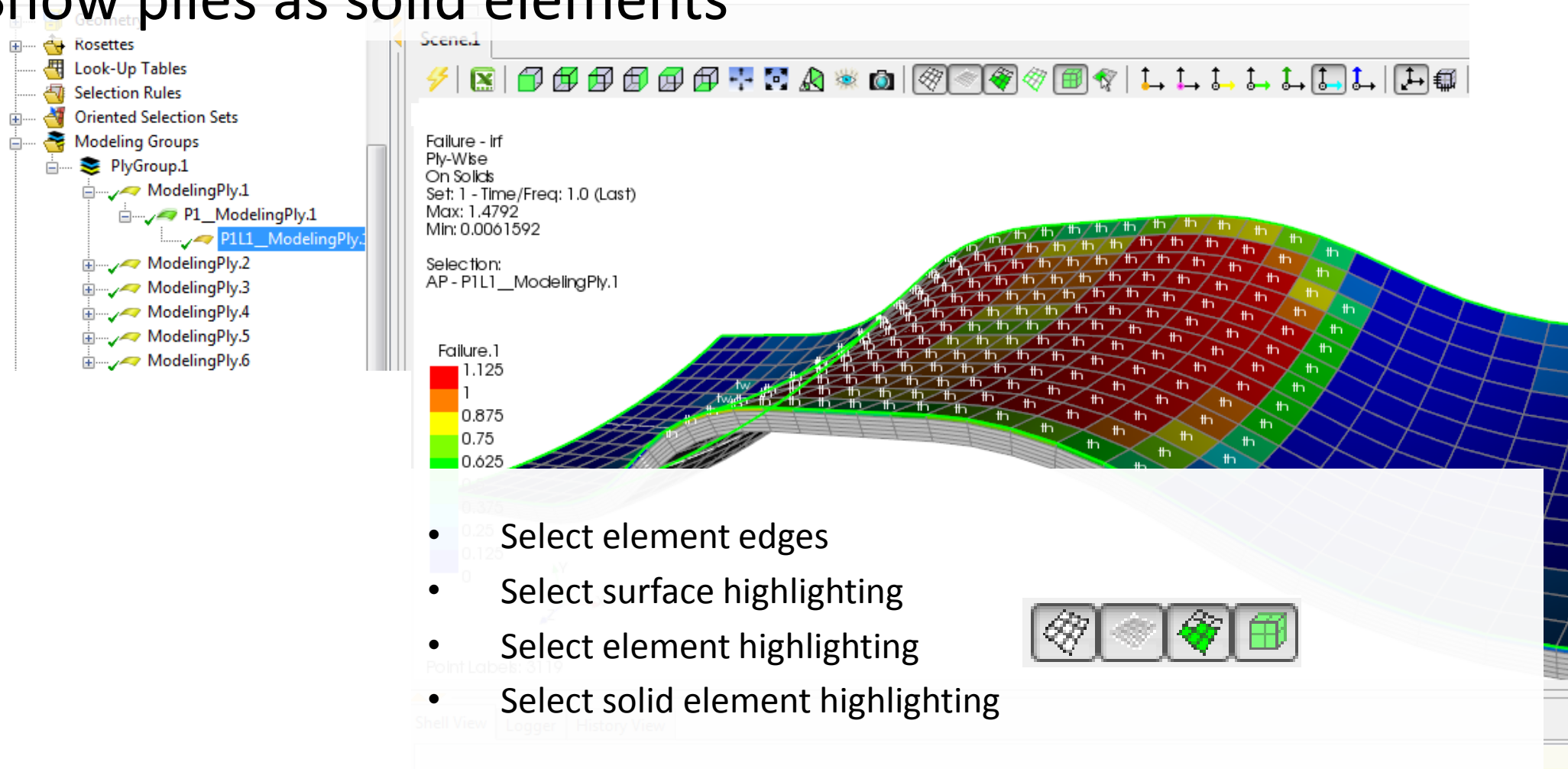


1. Add Failure Plot to Solution 1
2. show failure criteria ply wise



# 7.1 Workshop Solid Modeling

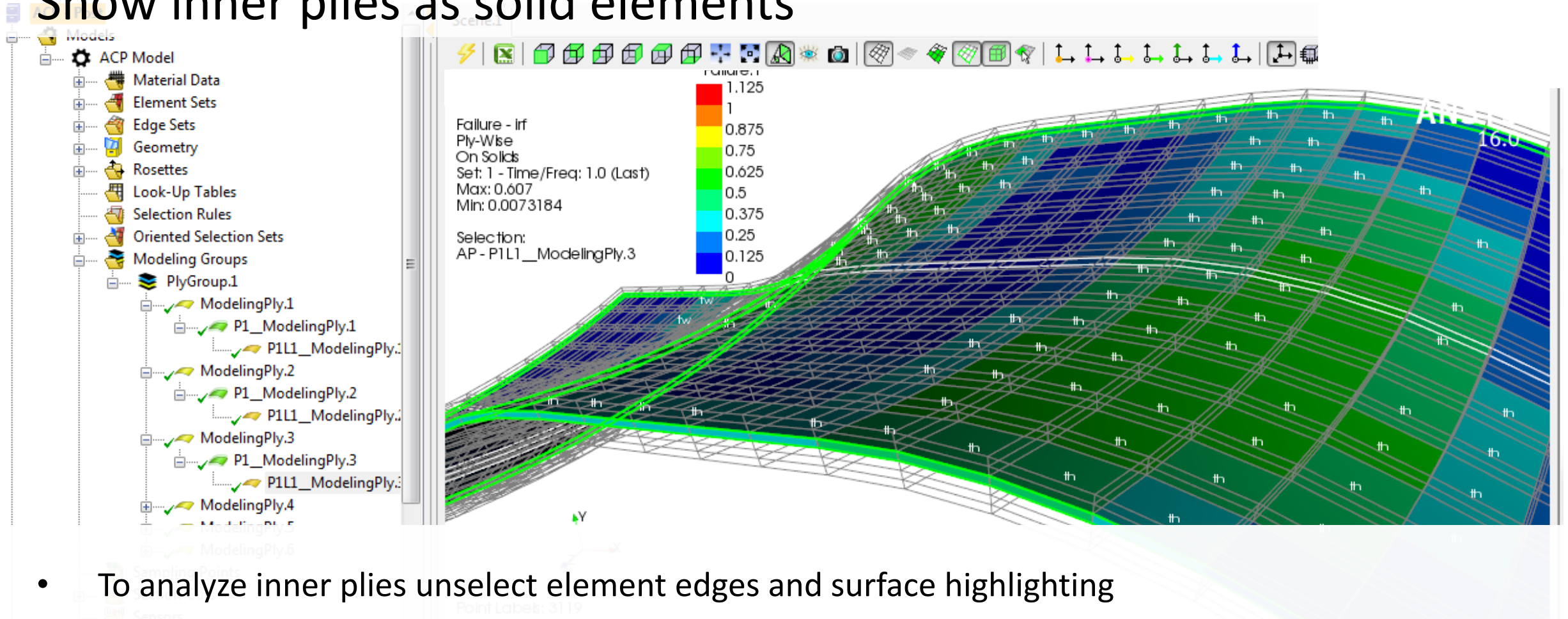
## Show plies as solid elements



- Sometimes it is necessary to reselect plies

# 7.1 Workshop Solid Modeling

## Show inner plies as solid elements

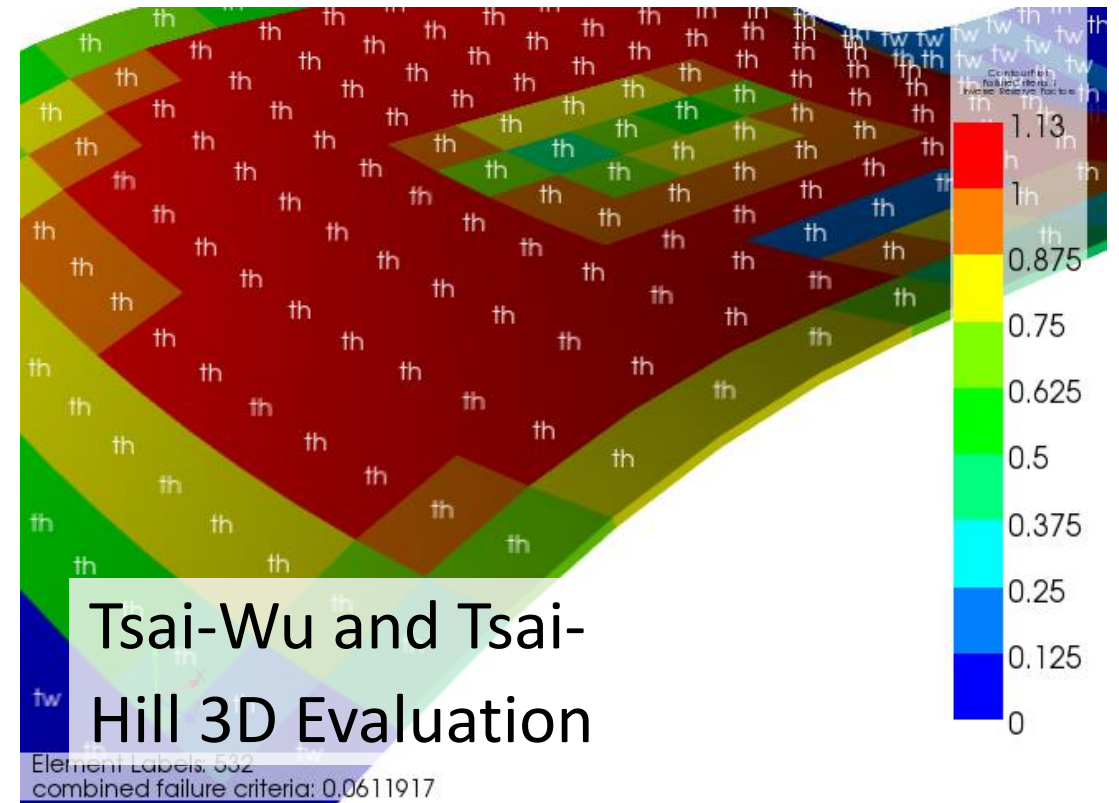
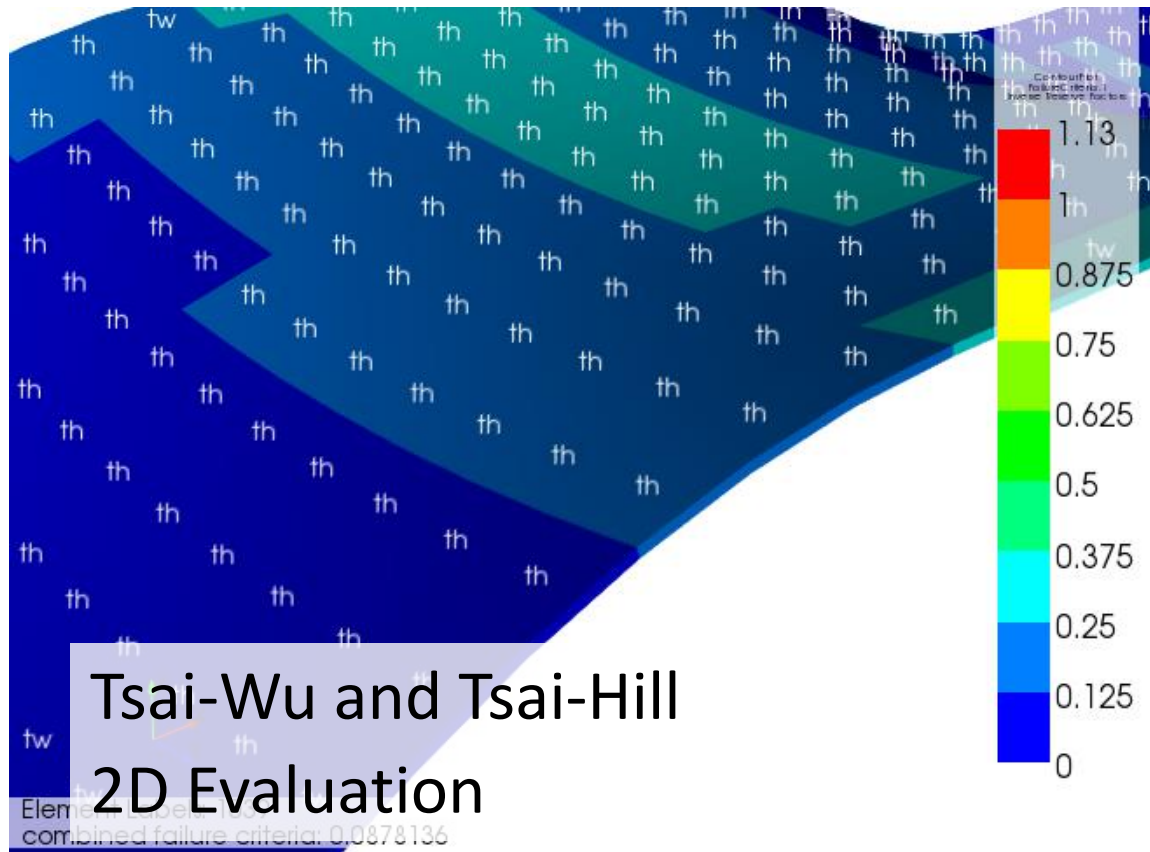


- To analyze inner plies unselect element edges and surface highlighting
- Failure criteria for different plies by selecting plies in the model tree



# 7.1 Workshop Solid Modeling

- Compare 2D Failure Criteria evaluation vs. 3D Failure Criteria evaluation



# 7.2 Workshop Solid Modeling

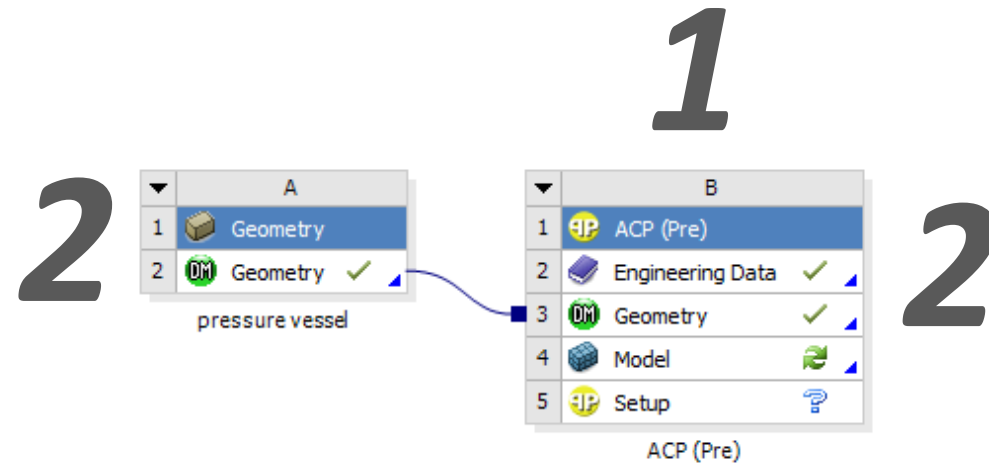
## Start ANSYS Workbench and Restore Archive

The screenshot displays the ANSYS Workbench interface. On the left is the 'Component Systems' tree, listing various analysis types such as Electric, Explicit Dynamics, Harmonic Response, Linear Buckling, Magnetostatic, Modal, Random Vibration, Response Spectrum, Rigid Dynamics, Static Structural, Steady-State Thermal, Thermal-Electric, Transient Structural, and Transient Thermal. Below these are 'ACP (Post)', 'ACP (Pre)', 'Autodyn', and 'BladeGen'. The main area shows three component systems: A, B, and C. Component A, labeled 'pressure vessel', contains '1 Geometry' and '2 DM Geometry' (marked with a green checkmark). Component B, labeled 'support', contains '1 Geometry' and '2 DM Geometry' (marked with a green checkmark). Component C, labeled 'Mechanical Model', contains '1 Mechanical Model', '2 Engineering Data' (marked with a green checkmark), '3 DM Geometry' (marked with a green checkmark), and '4 Model' (marked with a green checkmark). A blue arrow points from the 'DM Geometry' of Component B to the 'DM Geometry' of Component C.

1. Start ANSYS Workbench and restore Archive *pressure\_vessel\_from\_start\_19.0.wbpz*
2. Save the Workbench project



## 7.2 Workshop Solid Modeling



1. Drag and Drop ACP (Pre) in the project schematic
2. Connect the Geometry Cells

# 7.2 Workshop Solid Modeling

1

Outline of Schematic B2, F2: Engineering Data

A	
Contents of Engineering Data	
1	
2	Material
3	Epoxy_Carbon_UD_230GPa_Pregreg
4	Resin_Epoxy
5	Structural Steel

Click here to add a new material

Properties of Outline Row 3: Epoxy\_Carbon\_UD\_230GPa\_Pregreg

A		B	C
Property		Value	Unit
1			
2	Density	1.49E-09	mm <sup>3</sup> -3 t
3	Orthotropic Secant Coefficient of Thermal Expansion		
9	Orthotropic Elasticity		
10	Young's Modulus X direction	1.21E+05	MPa
11	Young's Modulus Y direction	8600	MPa
12	Young's Modulus Z direction	8600	MPa
13	Poisson's Ratio XY	0.27	
14	Poisson's Ratio YZ	0.4	
15	Poisson's Ratio XZ	0.27	
16	Shear Modulus XY	4700	MPa
17	Shear Modulus YZ	3100	MPa
18	Shear Modulus XZ	4700	MPa

1. In *Engineering Data* add composite material properties, Epoxy\_Carbon\_UD\_230GPa\_Pregreg, and specify the resin material, Resin\_Epoxy

2

Project

- Model (B4, F4)
  - Geometry
  - Coordinate Systems
  - Mesh
    - Face Sizing
    - Face Sizing cap
    - Named Selections

Details of "Face Sizing cap" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face

Definition	
Suppressed	No
Type	Element Size
<input type="checkbox"/> Element Size	50. mm
Behavior	Soft
<input type="checkbox"/> Curvature Normal Angle	Default
<input type="checkbox"/> Growth Rate	Default
<input type="checkbox"/> Local Min Size	Default (50. mm)

Face Sizing cap  
02.06.2015 13:13

Face Sizing cap

0 2e+003 (mm)  
1e+003

2

Details of "Face Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	3 Faces

Definition	
Suppressed	No
Type	Element Size
<input type="checkbox"/> Element Size	100. mm
Behavior	Soft
<input type="checkbox"/> Curvature Normal Angle	Default
<input type="checkbox"/> Growth Rate	Default
<input type="checkbox"/> Local Min Size	Default (70.085 mm)

Face Sizing  
02.06.2015 13:18

Face Sizing

0 2e+003 (mm)  
1e+003

2. Double click *Model* of ACP (Pre), add mesh properties for cap and lateral cylinder, generate the mesh on the surface

# 7.2 Workshop Solid Modeling

The image displays the ANSYS ACP (Advanced Composite Preparation) interface with three numbered callouts indicating the setup steps:

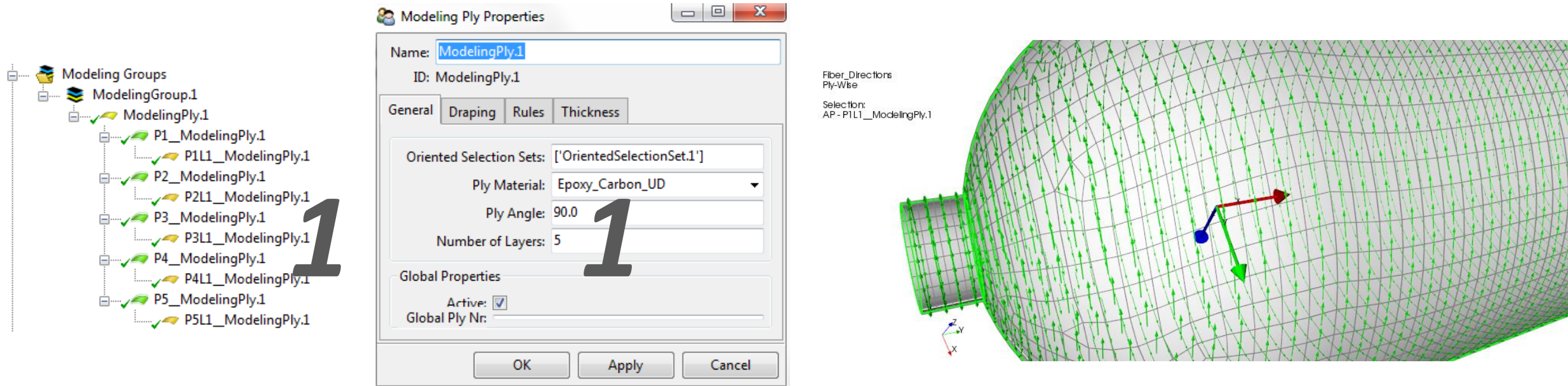
- 1** **Fabric Properties**: The **Fabric Properties** dialog box is shown with the **General** tab selected. The **Name** is **Epoxy\_Carbon\_UD** and the **ID** is **Epoxy\_Carbon\_UD**. The **Material** is set to **Epoxy\_Carbon\_UD\_230GPa\_Prepreg**. Other properties include **Thickness: 1.0**, **Price/Area: 0.0**, and **Mass/Area: 1.49e-09**. The **Post-Processing** section has **Ignore for Post-Processing** unchecked.
- 2** **Rosette Properties**: The **Rosette Properties** dialog box is shown with the **General** tab selected. The **Name** is **Rosette.1** and the **ID** is **Rosette.1**. The **Type** is set to **Parallel**. The **Definition** section shows **Origin: (0.0000,0.0000,0.0000)**, **Direction 1: (0.0000,1.0000,0.0000)**, and **Direction 2: (1.0000,0.0000,0.0000)**. The **Swap** button is visible.
- 3** **Oriented Selection Set Properties**: The **Oriented Selection Set Properties** dialog box is shown with the **General** tab selected. The **Name** is **OrientedSelectionSet.1** and the **ID** is **OrientedSelectionSet.1**. The **Orientation** section shows **Element Sets: ['All\_Elements']**, **Orientation Point: (0.0000,0.0000,0.0000)**, and **Orientations Direction: (0.0000,-1.0000,0.0000)**. The **Selection Method** is set to **Minimum Angle** and the **Rosettes** are set to **['Rosette.1']**.

The background shows the ACP - Pre Models tree with the following structure:

- ACP - Pre
  - Models
    - ACP Model
      - Material Data
        - Materials
          - Epoxy\_Carbon\_UD (selected)
        - Fabrics
        - Stackups
        - Sub Laminates
      - Element Sets
        - All\_Elements
      - Edge Sets
      - Geometry
      - Rosettes
        - Rosette.1 (selected)
      - Look-Up Tables
      - Selection Rules
      - Oriented Selection Sets
        - OrientedSelectionSet.1 (selected)

Double click setup of ACP (Pre), define (1) a fabric material, (2) a new rosette, and (3) an oriented selection set as shown in this slide

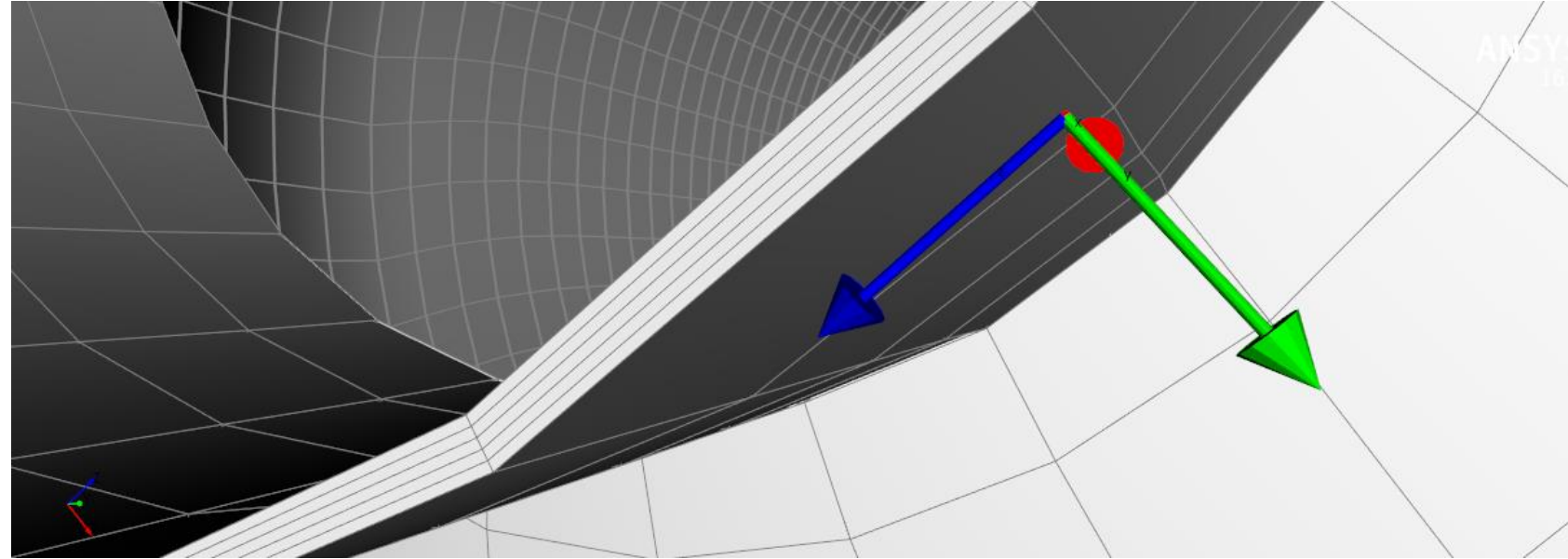
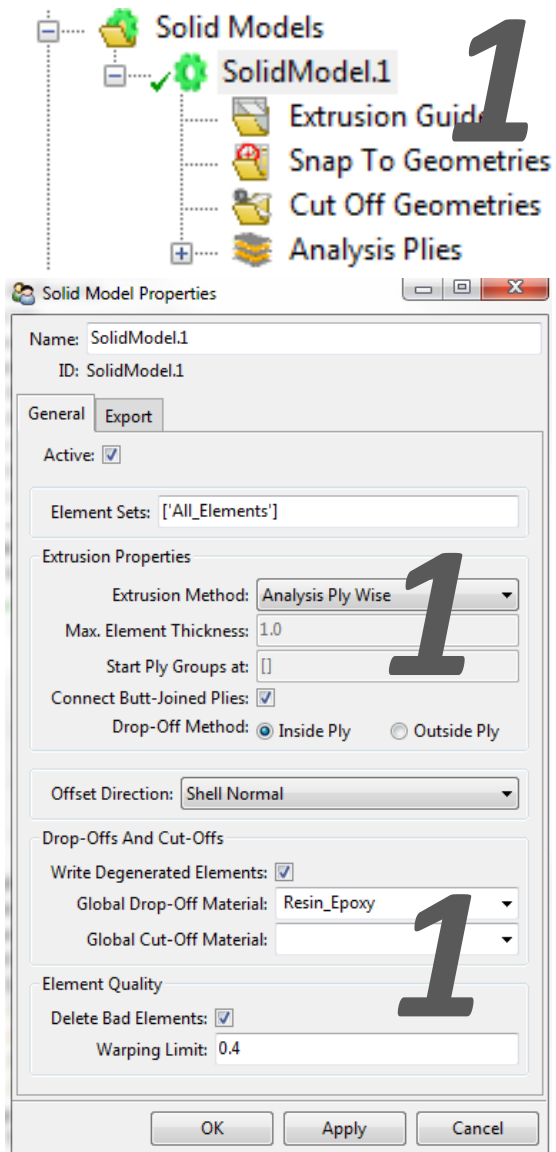
## 7.2 Workshop Solid Modeling



1. Create a Modeling Group and add 5 plies with fibers oriented in the cylinder tangential direction

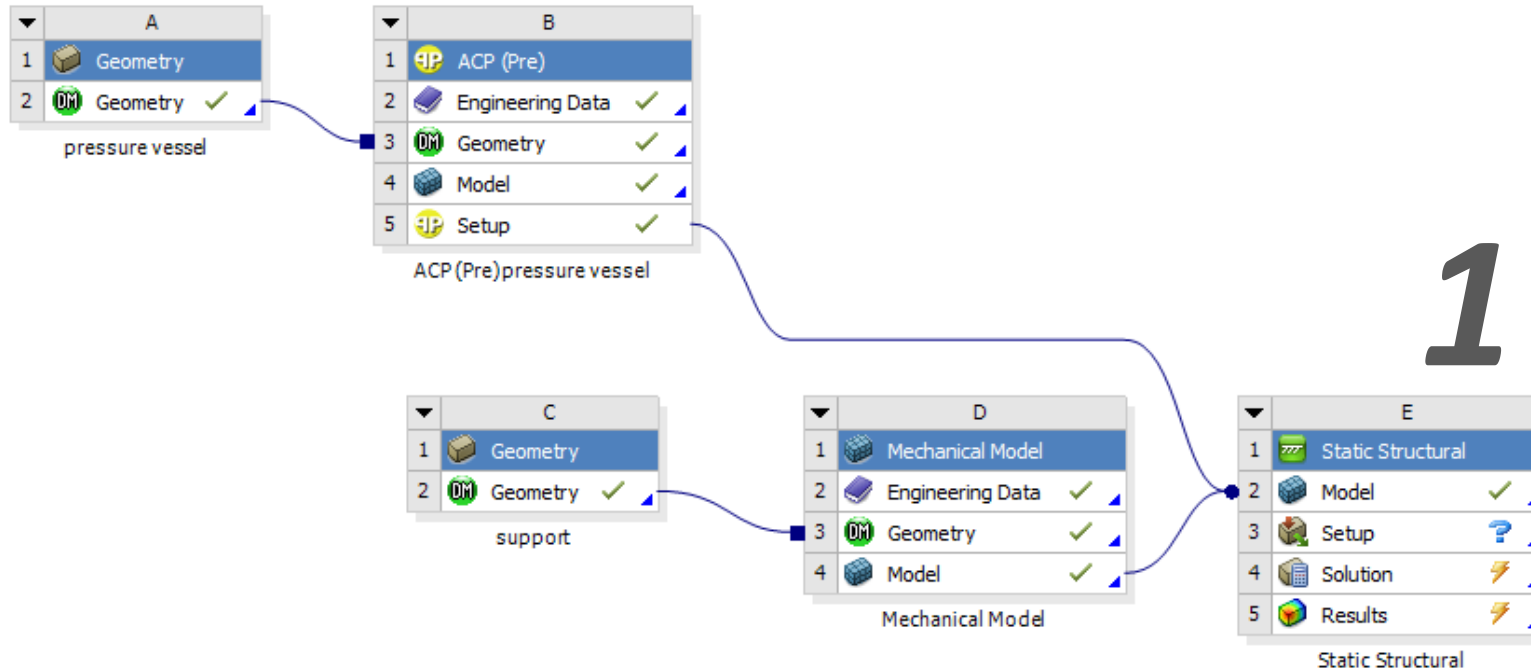


## 7.2 Workshop Solid Modeling



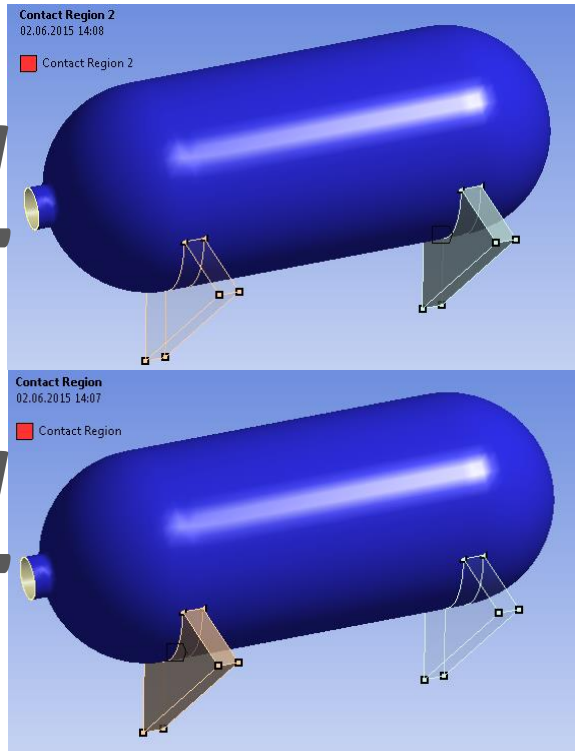
1. Extrude a solid model, extrusion method Analysis Ply Wise, Resin\_Epoxy as global drop-off material
2. Update the model and leave ACP (Pre)

## 7.2 Workshop Solid Modeling

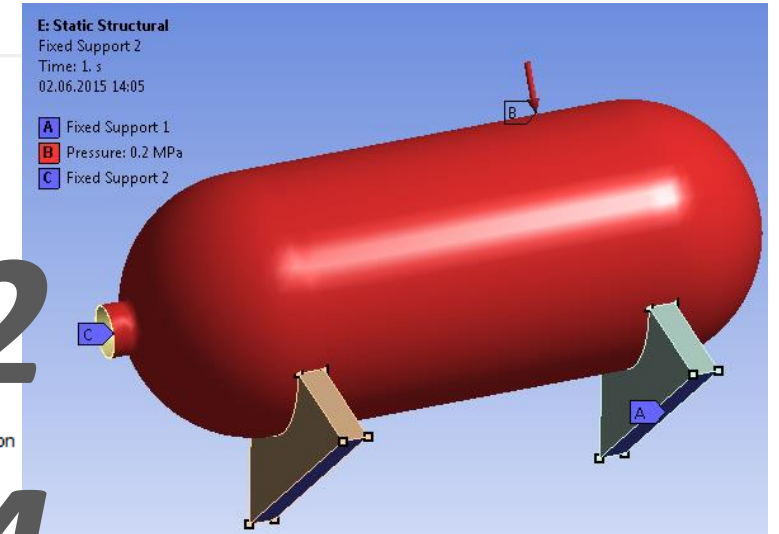
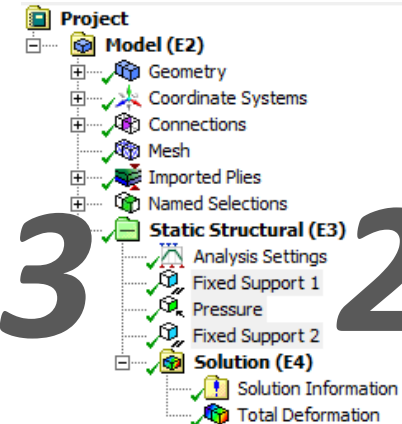


1. Add Static Structural to the project schematic and connect to the ACP (Pre) model of the vessel and the mechanical model of the supports
2. Open Mechanical in the Static Structural box

## 7.2 Workshop Solid Modeling

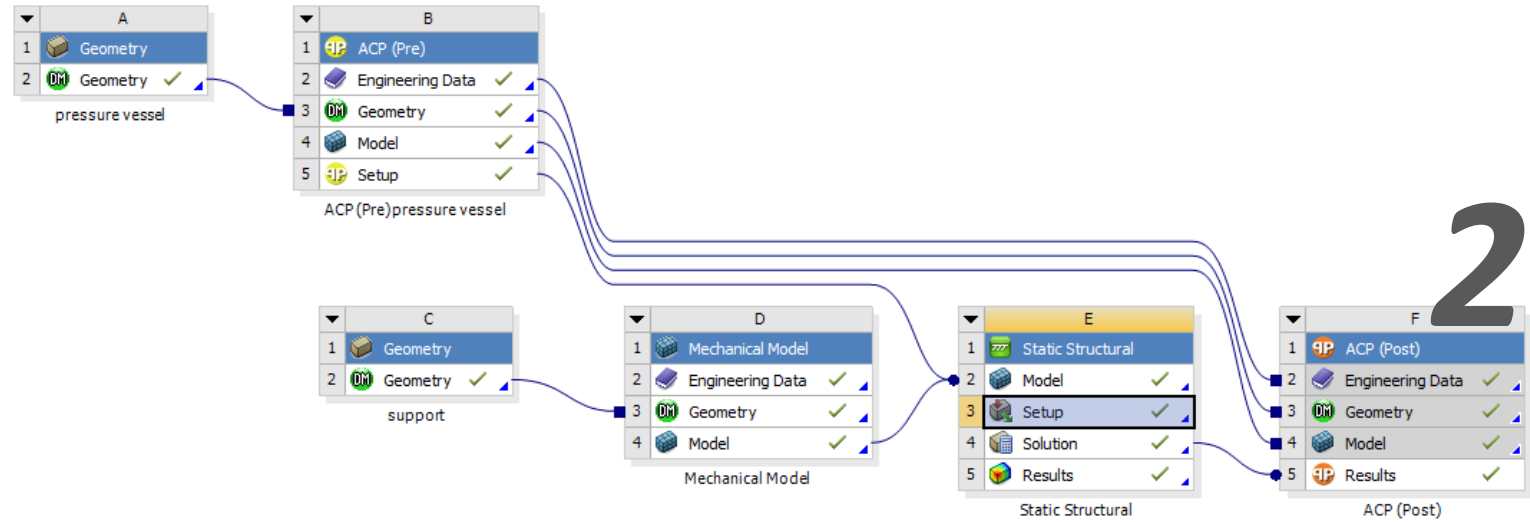
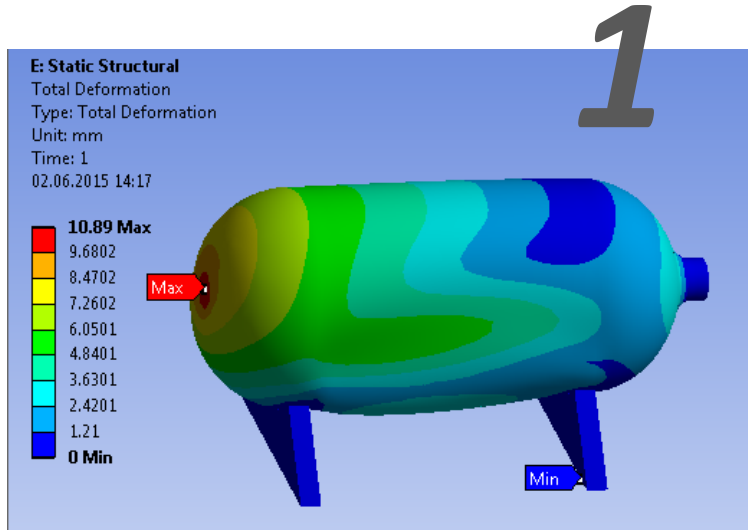


Details of "Contact Region"	
Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	Solid(Mechanical Model)
Target Bodies	SolidModel.1
Definition	
Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	15.909 mm
Suppressed	No



1. Add bonded contact between the vessel and the two supports in the connections
2. Fix the bottom of the two supports and the rim of the cap
3. Apply pressure on the inside surface of the vessel
4. Add total deformation in the solution for plotting purposes. Solve the model.

## 7.2 Workshop Solid Modeling



1. Check the total deformation in Mechanical
2. Drag and Drop ACP (Post) on ACP (Pre) in the project schematic and connect its Results cell with Solution of the Static Structural Analysis
3. Update the project and open ACP (Post) to check the different laminae of the composite



## 7.2 Workshop Solid Modeling

The screenshot displays the ANSYS ACP-Post interface. On the left is a tree view of the model hierarchy. The 'Failure Criteria Definition' dialog box (labeled 1) is open, showing 'FailureCriteria.1' with 'Max Strain' and 'Max Stress' selected under 'Reinforced Ply Criteria'. The 'Failure' dialog box (labeled 2) is also open, showing 'Failure.1' with 'Data Scope' set to 'All Elements' and 'Component' set to 'Inverse Reserve Factors'. To the right, a 3D model of a cylindrical vessel is shown with a failure plot. A color scale legend for 'Failure.1' indicates values from 0 to 1.125, with a maximum of 1.8734 and a minimum of 0.016279. The plot shows a critical spot near the vessel cap.

1. Add Max Strain and Max Stress in the Failure Criteria Definition

2. Insert Failure in the solution, the maximum IRF element wise. The critical spot is near the vessel cap (need to introduce a fillet in the geometry to reduce stress concentration)