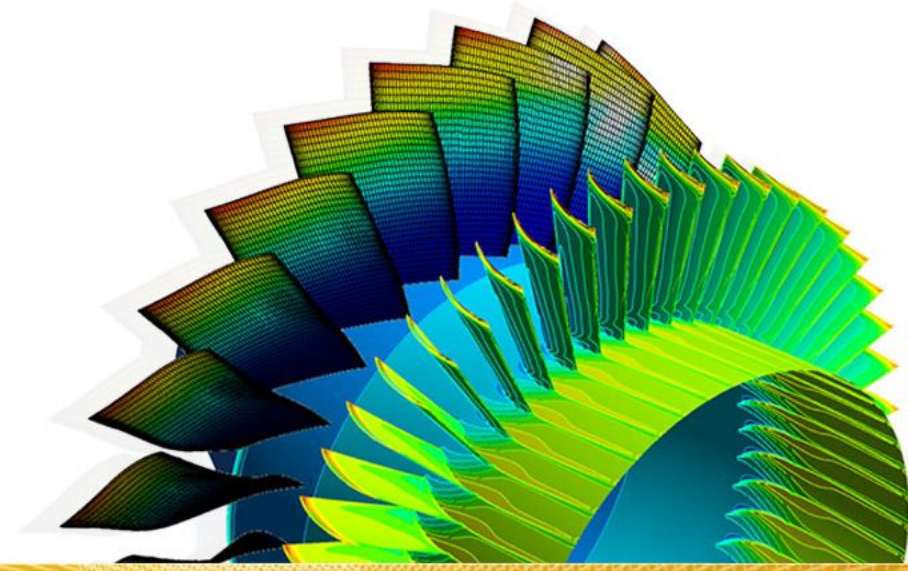
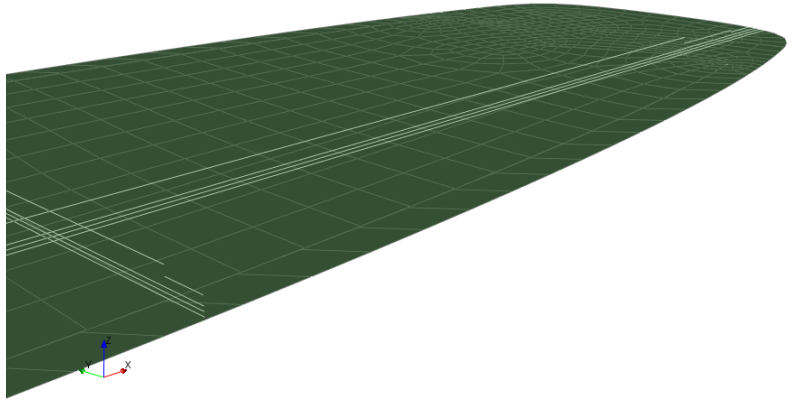
ANSYS Composite PrepPost 19.0

Workshop 03.2 – Kiteboard



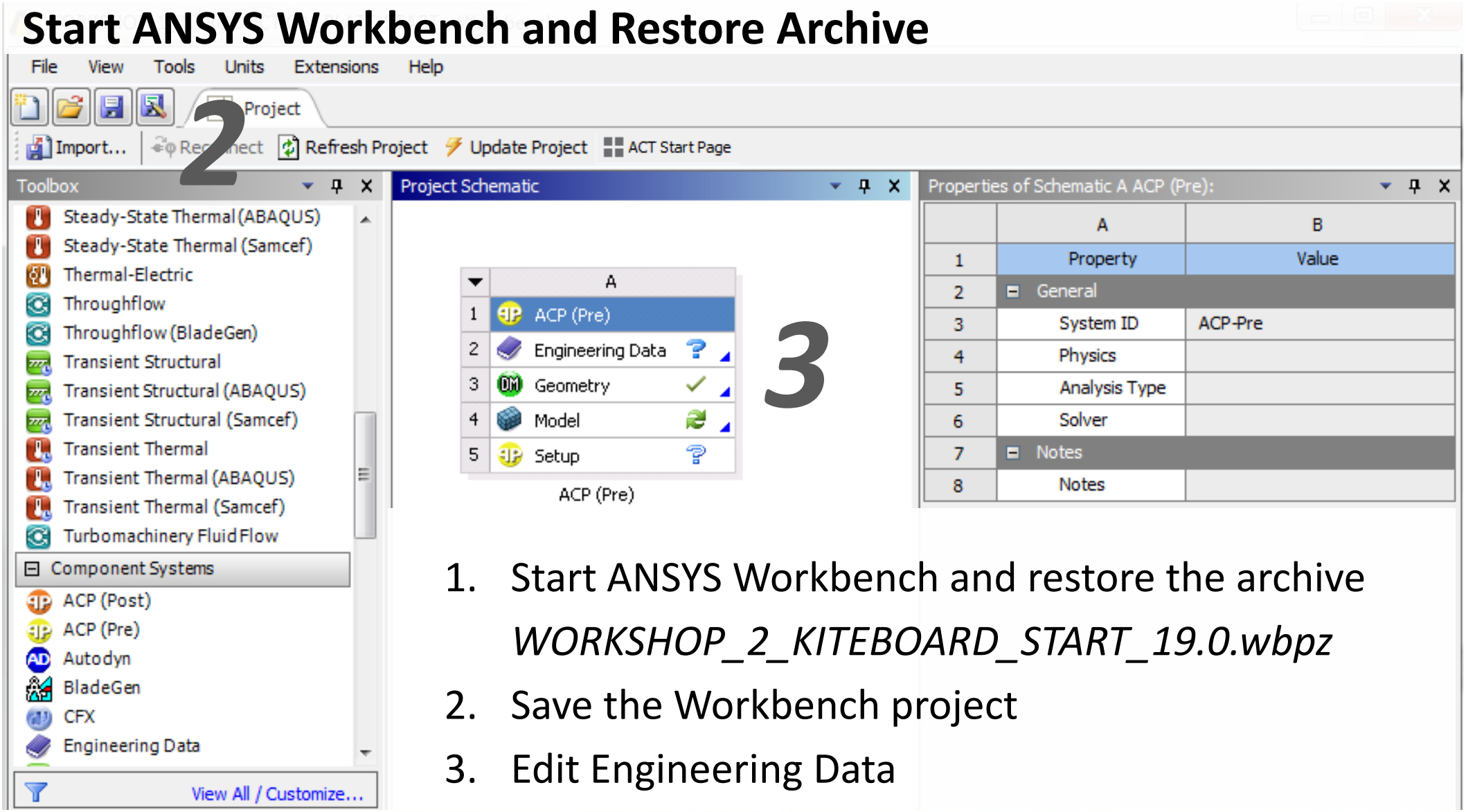
2. Workshop Kiteboard

- In this workshop we will model a kiteboard.
- We will go through the complete process of modeling - solving - postprocessing.
- The bottom plies for the kiteboard are three plies arranged as -45°,0°,45° layup. On top of those layers we will model a sandwich core. On top of the sandwich core we will model a -45°,0°,45° layup.



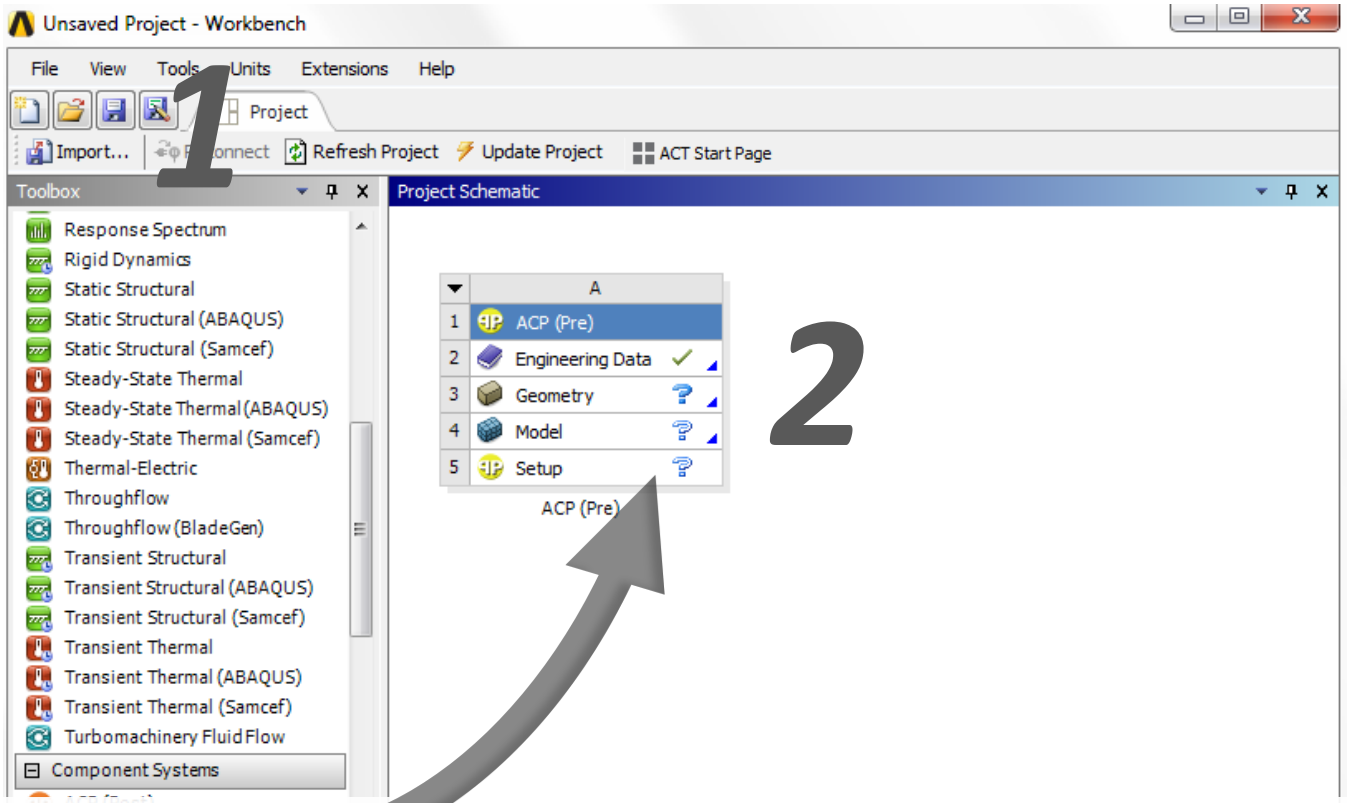
2. Workshop Kiteboard

Start ANSYS Workbench and Restore Archive



1. Start ANSYS Workbench and restore the archive *WORKSHOP_2_KITEBOARD_START_19.0.wbpz*
2. Save the Workbench project
3. Edit Engineering Data

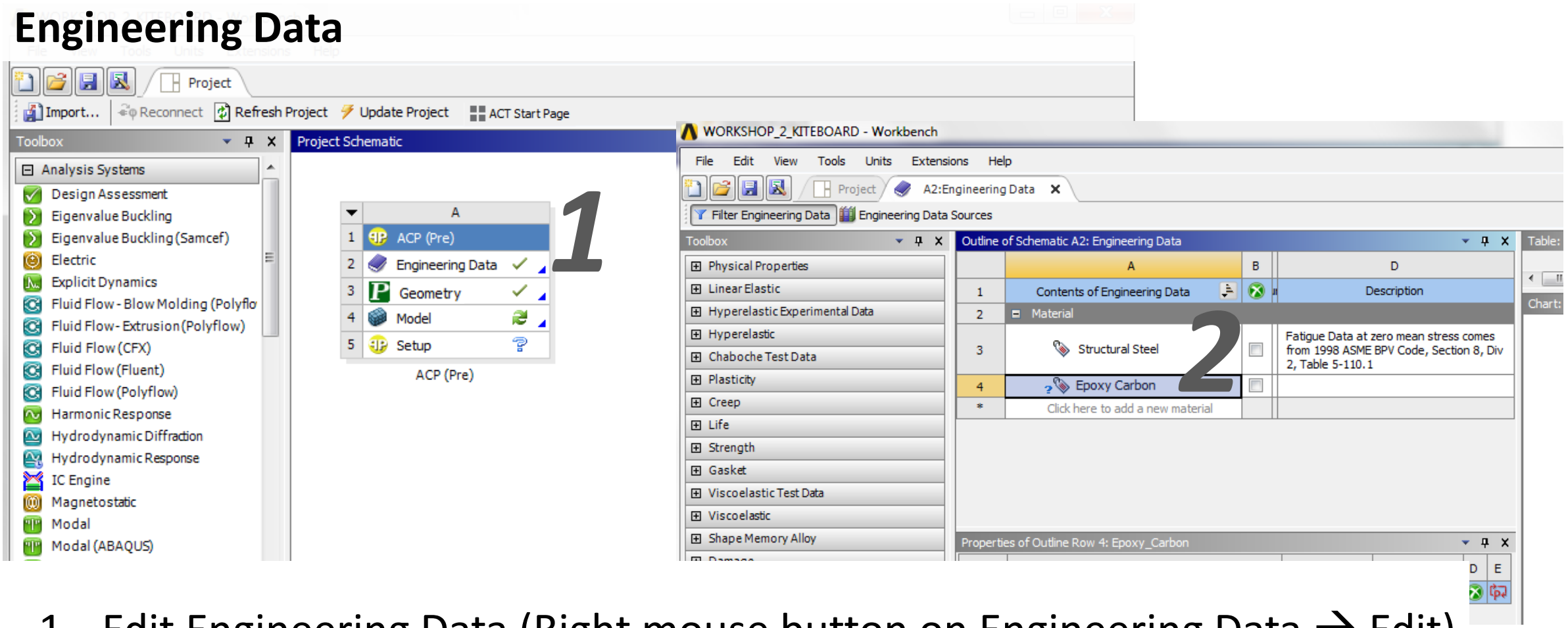
2. Workshop Kiteboard



1. Start ANSYS Workbench and save project
2. Drag and Drop an ACP (Pre) component into the project schematic

2. Workshop Kiteboard

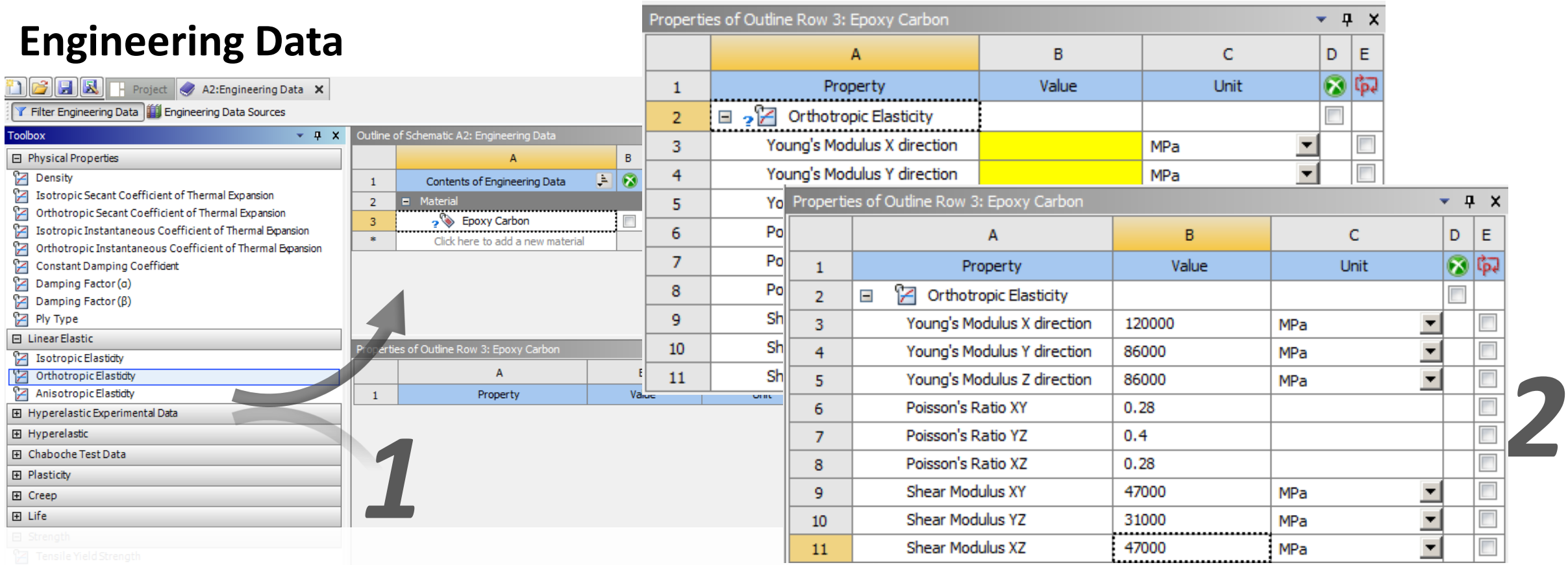
Engineering Data



1. Edit Engineering Data (Right mouse button on Engineering Data → Edit)
2. Add a new Material “Epoxy Carbon”

2. Workshop Kiteboard

Engineering Data

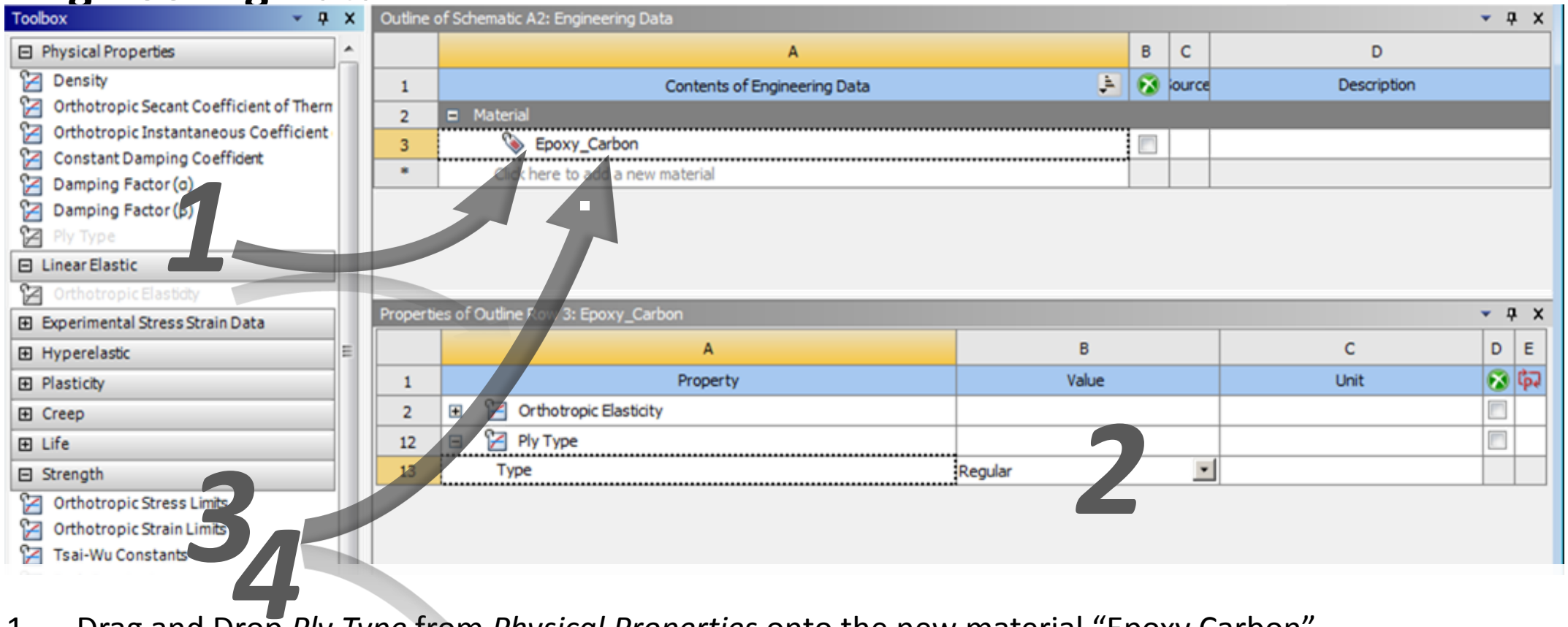


1. Drag and Drop ***Orthotropic Elasticity*** onto the new material "Epoxy Carbon"
2. Define the following material properties

E_x = 1.2e5 MPa	E_y = 8.6e4 MPa	E_z = 8.6e4 MPa
v_{xy} = 0.28	v_{yz} = 0.4	v_{xz} = 0.28
G_{xy} = 4.7e4 MPa	G_{yz} = 3.1e4 MPa	G_{xz} = 4.7e4 MPa

2. Workshop Kiteboard

Engineering Data



1. Drag and Drop *Ply Type* from *Physical Properties* onto the new material “Epoxy Carbon”
2. Define ply type as **regular**
3. Drag and Drop Orthotropic Stress Limits from *Strength* on material “Epoxy Carbon”
4. Drag and Drop Orthotropic Strain Limits from *Strength* on material “Epoxy Carbon”

2. Workshop Kiteboard

Engineering Data

Properties of Outline Row 3: Epoxy Carbon

	A	B	C	D	E
1	Property	Value	Unit		
2	⊞ Orthotropic Elasticity				
12	⊟ Orthotropic Stress Limits				
13	Tensile X direction	2274	MPa		
14	Tensile Y direction	30	MPa		
15	Tensile Z direction	30	MPa		
16	Compressive X direction	-1100	MPa		
17	Compressive Y direction	-100	MPa		
18	Compressive Z direction	-100	MPa		
19	Shear XY	60	MPa		
20	Shear YZ	30	MPa		
21	Shear XZ	60	MPa		

1. Define stress limits for the material

$X_t = 2274\ MPa$ $X_c = -1100\ MPa$

$Y_t = 30\ MPa$ $Y_c = -100\ MPa$

$Z_t = 30\ MPa$ $Z_c = -100\ MPa$

$S_{xy} = 60\ MPa$ $S_{yz} = 30\ MPa$

$S_{xz} = 60\ MPa$

2. Workshop Kiteboard

Engineering Data

Properties of Outline Row 3: Epoxy Carbon

	A	B	C	D	E
1	Property	Value	Unit		
2	⊞ Orthotropic Elasticity				
12	⊞ Orthotropic Stress Limits				
22	⊟ Orthotropic Strain Limits				
23	Tensile X direction	0.017			
24	Tensile Y direction	0.003			
25	Tensile Z direction	0.003			
26	Compressive X direction	-0.011			
27	Compressive Y direction	-0.019			
28	Compressive Z direction	-0.019			
29	Shear XY	0.012			
30	Shear YZ	0			
31	Shear XZ	0			

1. Define strain limits for the material

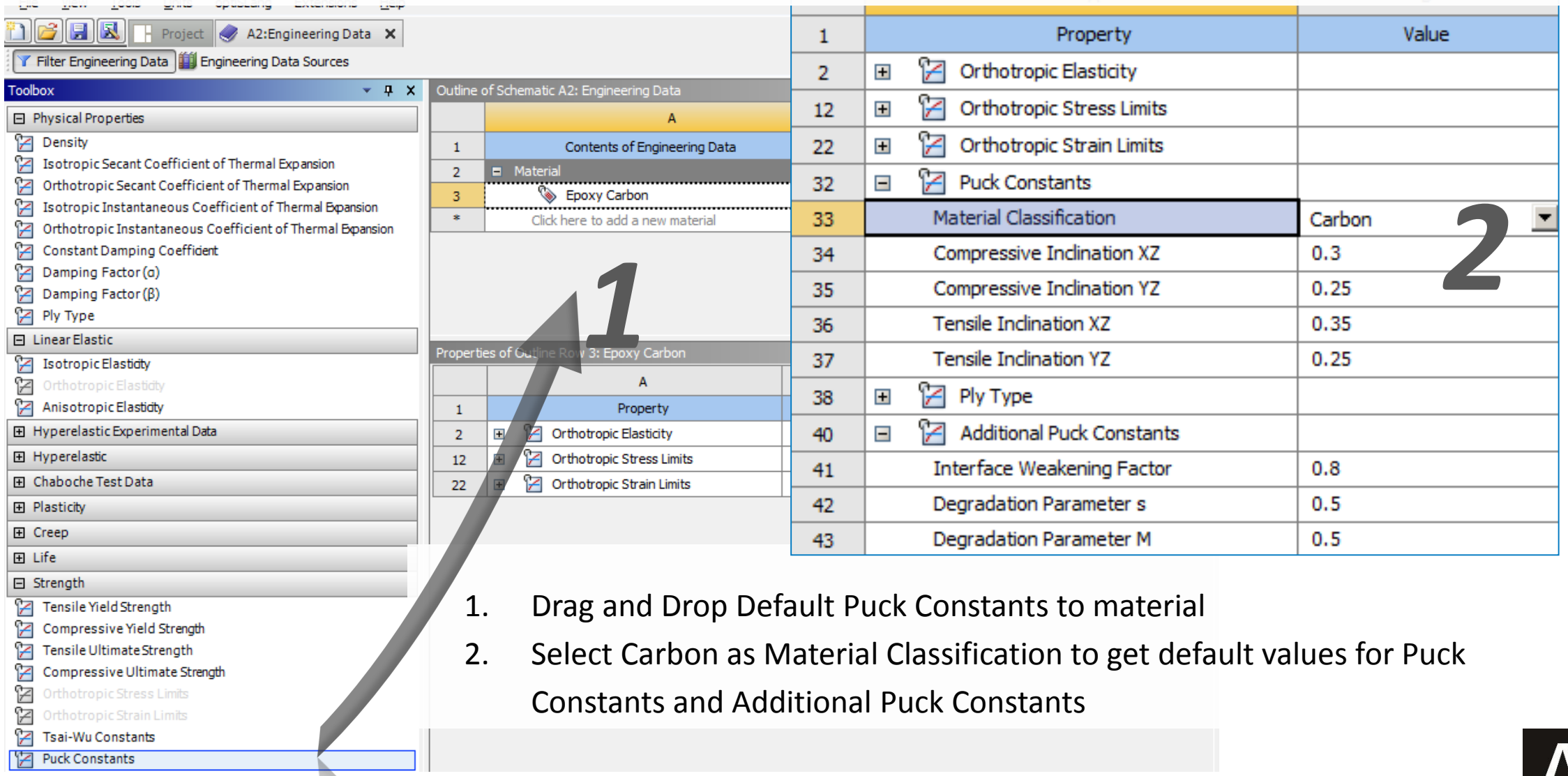
$e_{xt} = 0.017$ $e_{Xc} = -0.011$ $e_{Sxy} = 0.012$

$e_{yt} = 0.003$ $e_{yc} = -0.019$ $e_{Syz} = 0$

$e_{zt} = 0.003$ $e_{zc} = -0.019$ $e_{Sxz} = 0$

2. Workshop Kiteboard

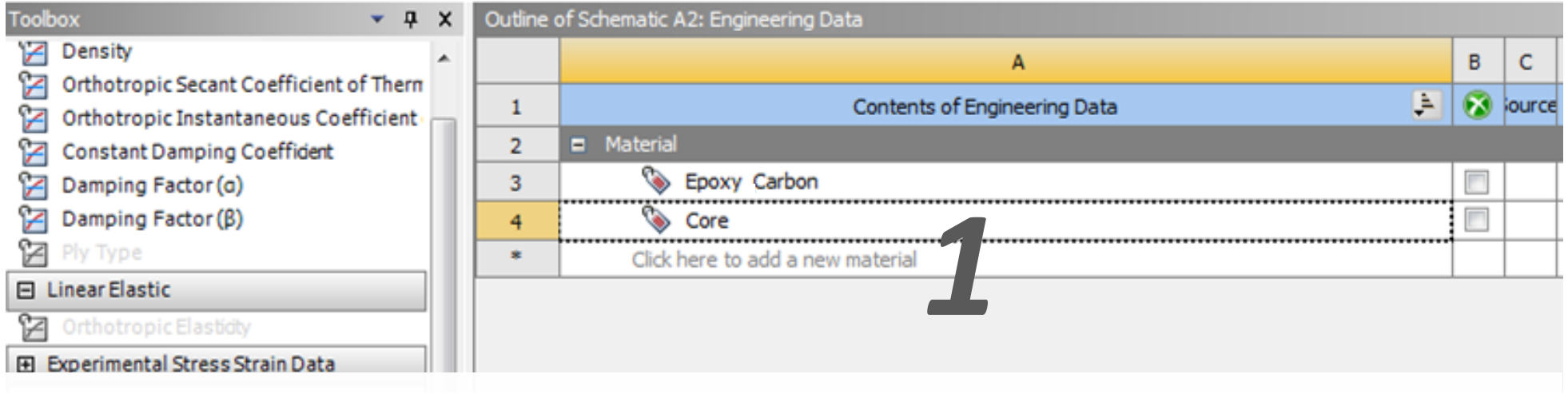
Engineering Data



1. Drag and Drop Default Puck Constants to material
2. Select Carbon as Material Classification to get default values for Puck Constants and Additional Puck Constants

2. Workshop Kiteboard

Engineering Data



1. Define a new material with orthotropic elastic properties for the core

   $E_x = E_y = E_z = 60$ MPa *(Core material properties are isotropic but to define*

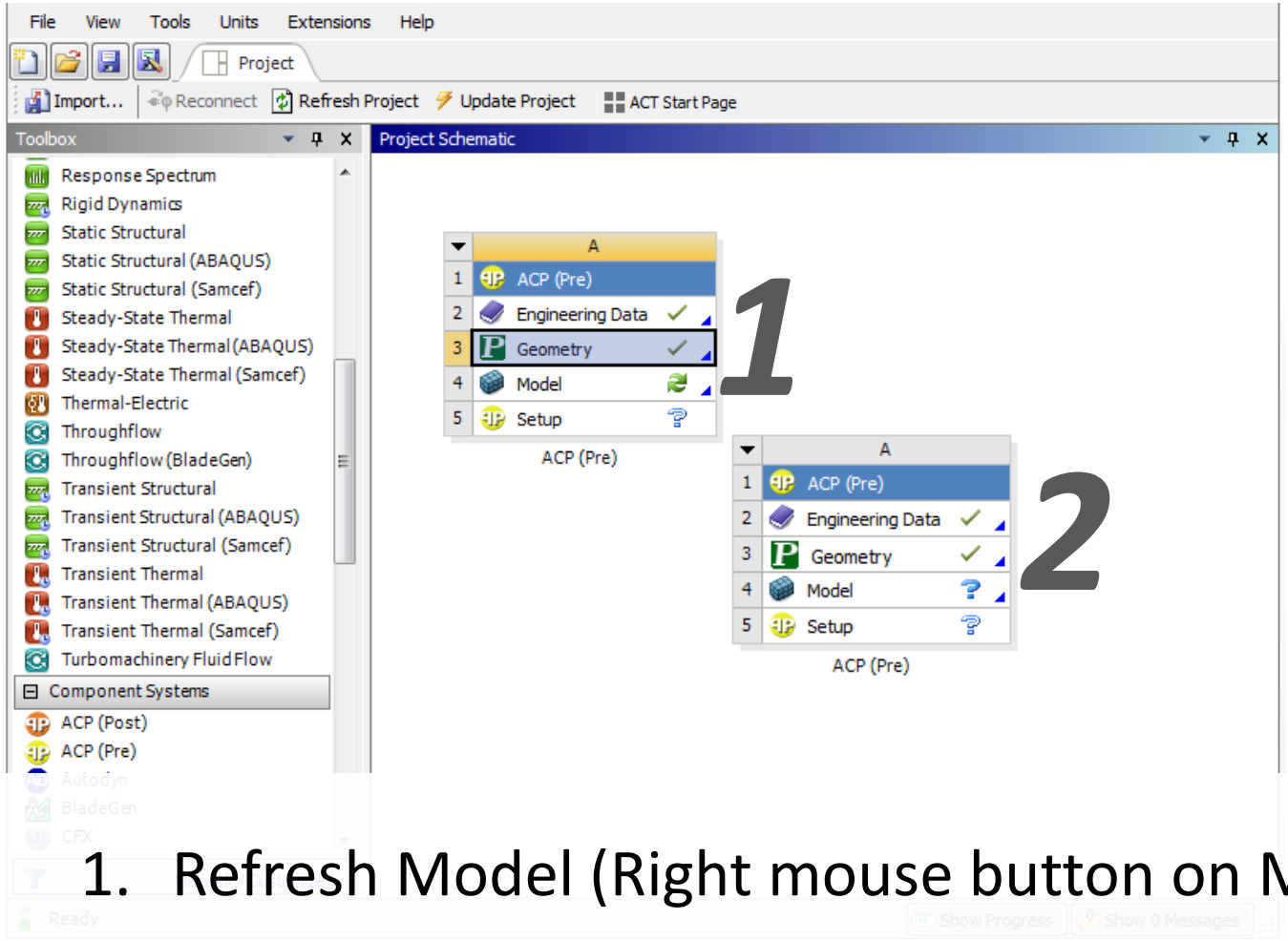
   $v_{xy} = v_{yz} = v_{xz} = 0.35$ *the correct stress limits we define the core material*

   $G_{xy} = G_{yz} = G_{xz} = 23$ MPa *as orthotropic)*

2. Specify the Ply Type as *Orthotropic Homogenous Core*
3. Define Stress Limits *$Z_t = 1.1$ MPa, $S_{xz} = 0.8$ MPa, $S_{yz} = 0.8$ MPa*

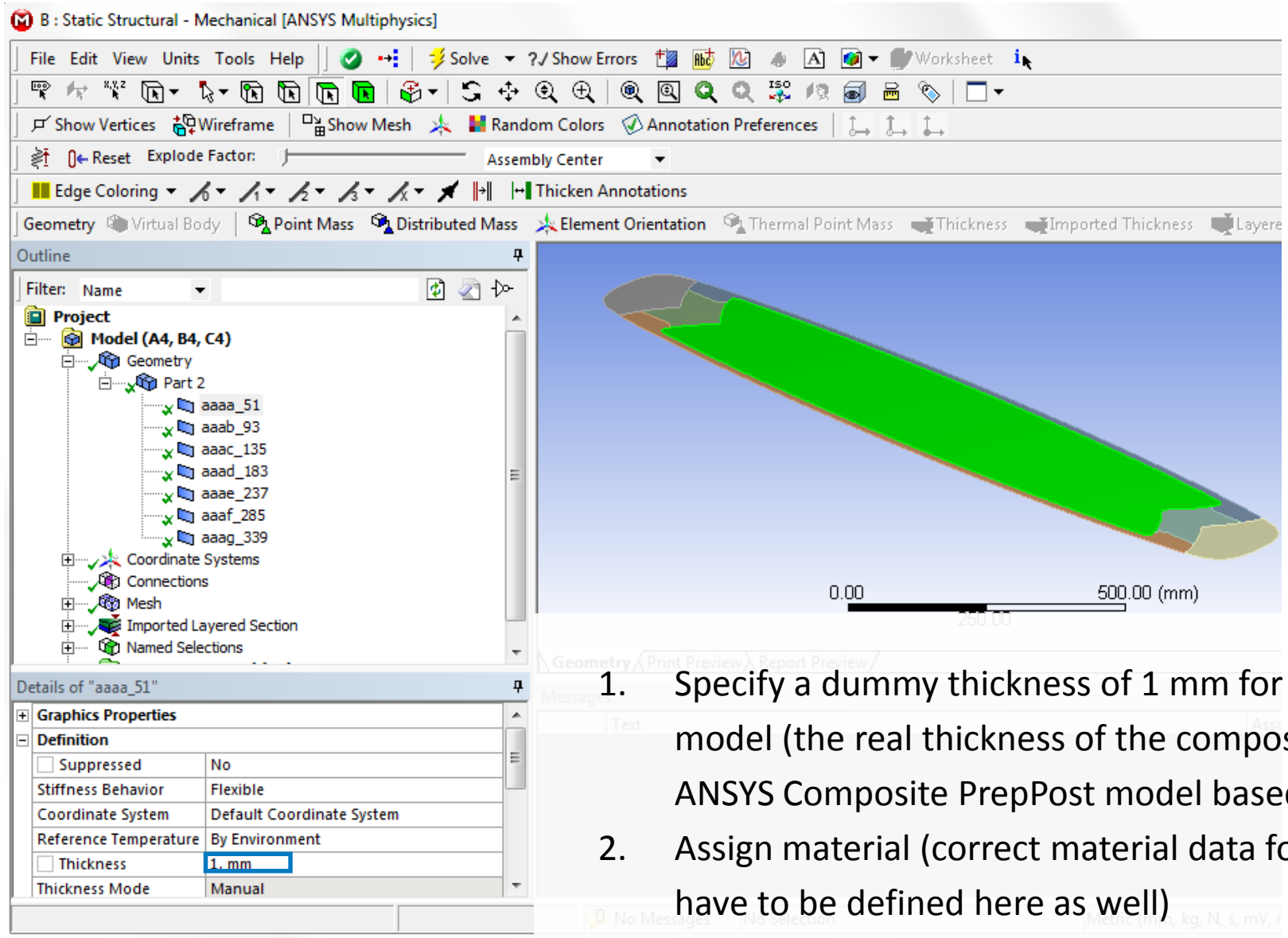
   (the remaining stress limits can be defined as zero)
4. Return to Project

2. Workshop Kiteboard



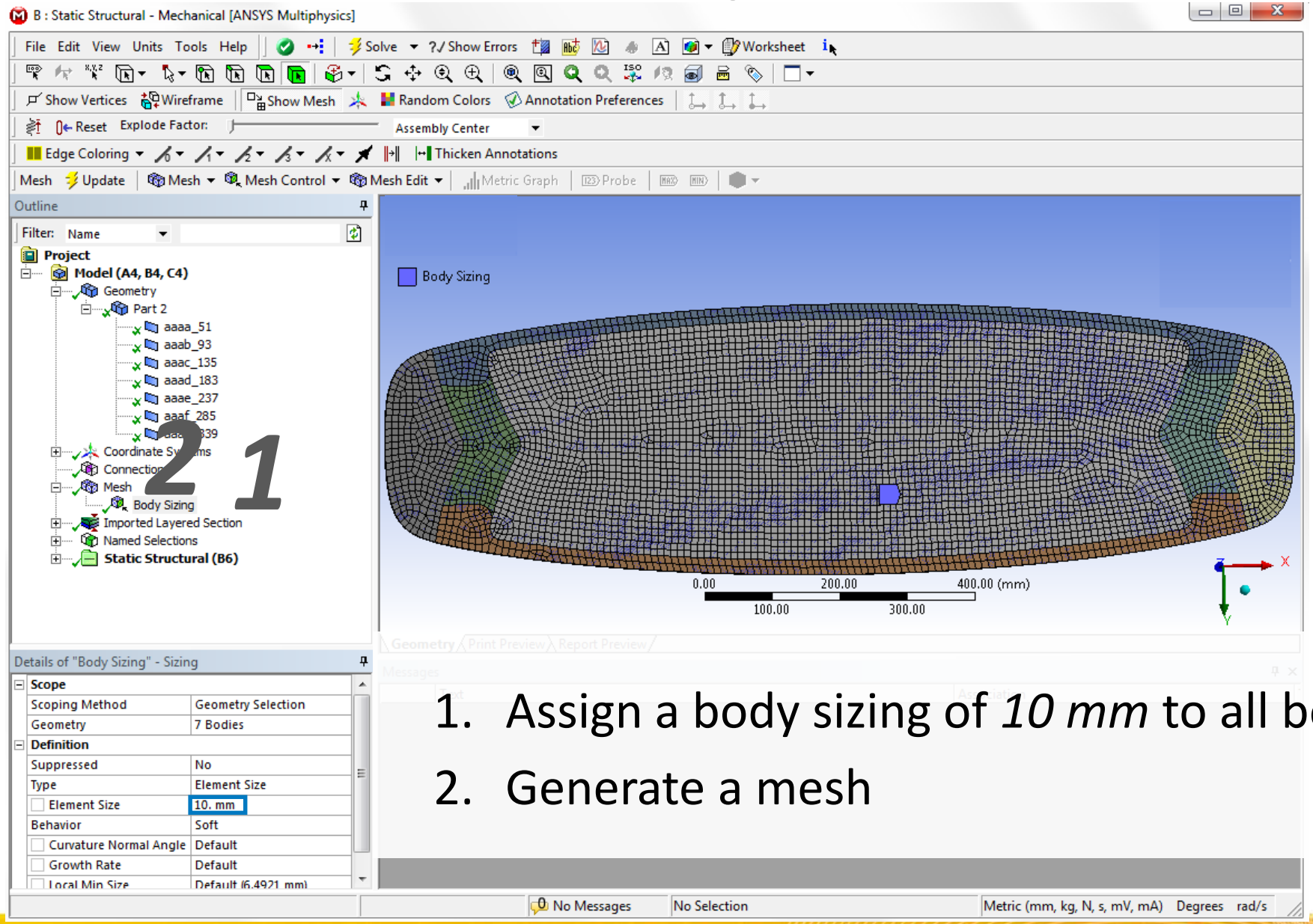
1. Refresh Model (Right mouse button on Model → Refresh)
2. Open ANSYS Mechanical (Right mouse button on Model → Edit)

2. Workshop Kiteboard



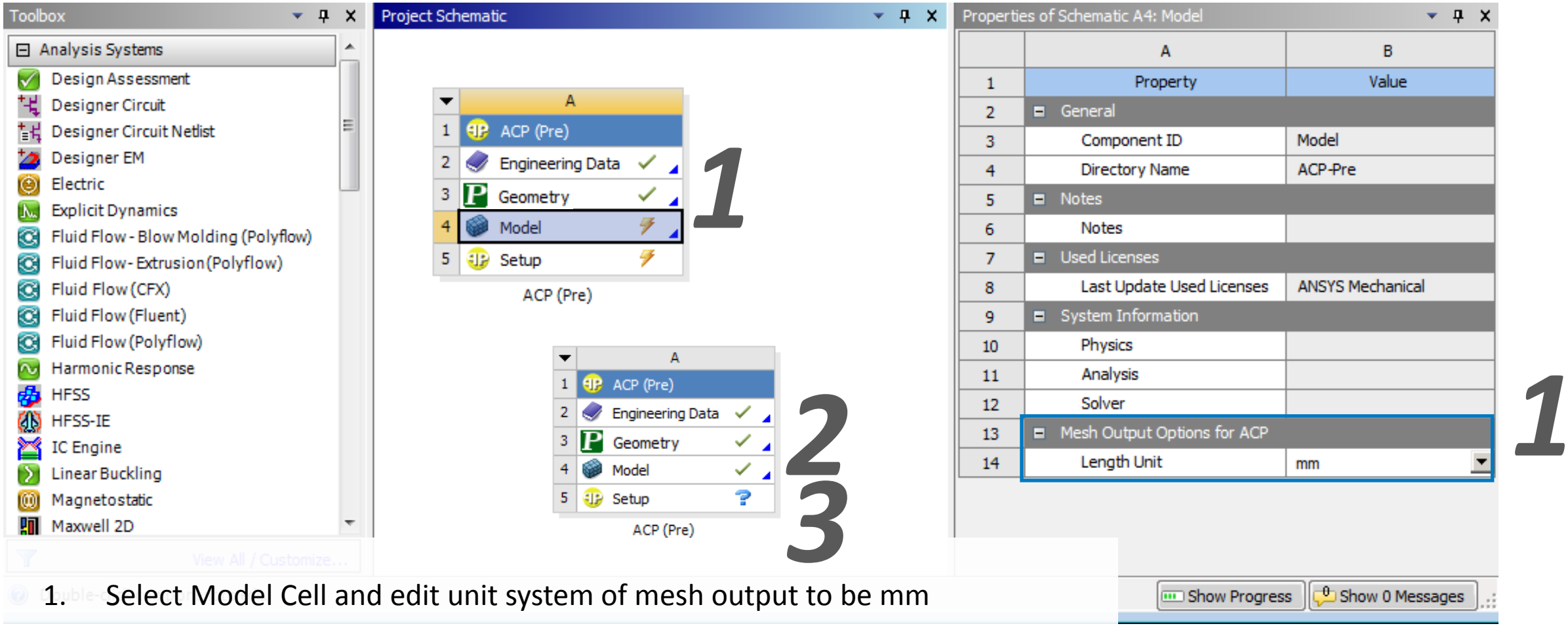
1. Specify a dummy thickness of 1 mm for all composite parts of the model (the real thickness of the composite parts will be defined in the ANSYS Composite PrepPost model based on the composite layup)
2. Assign material (correct material data for non composite materials have to be defined here as well)

2. Workshop Kiteboard



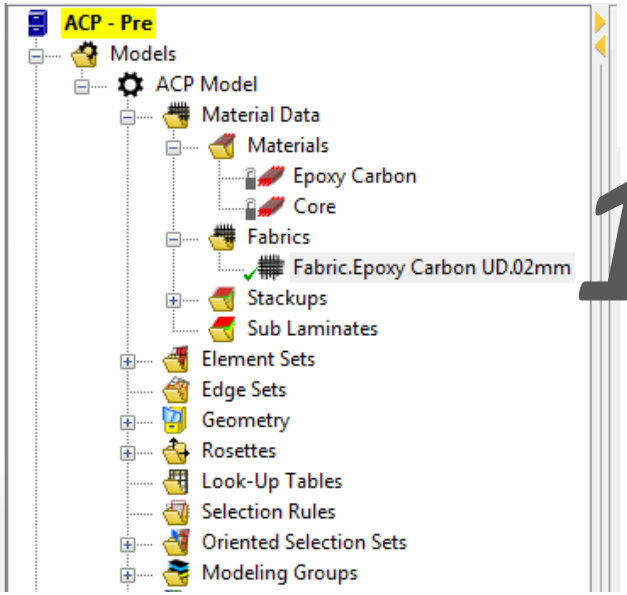
1. Assign a body sizing of *10 mm* to all bodies
2. Generate a mesh

2. Workshop Kiteboard

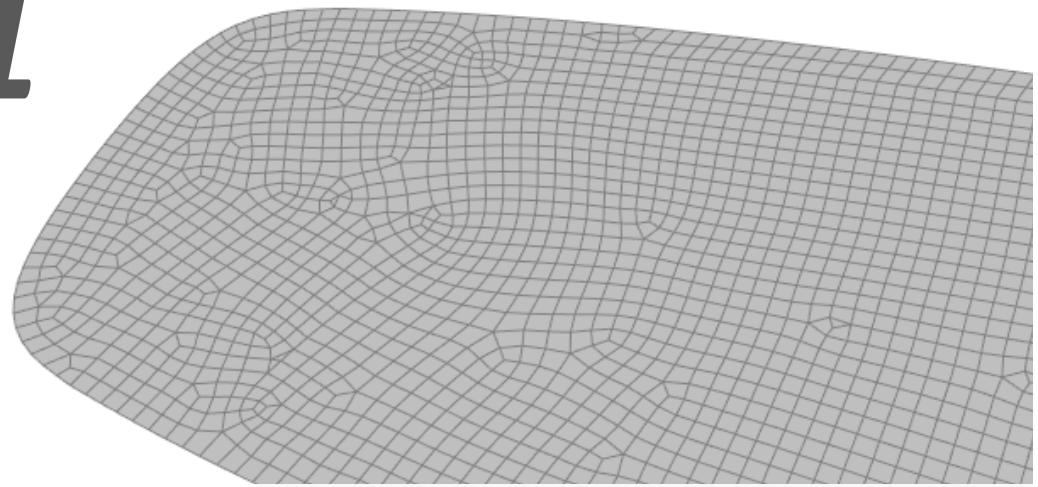


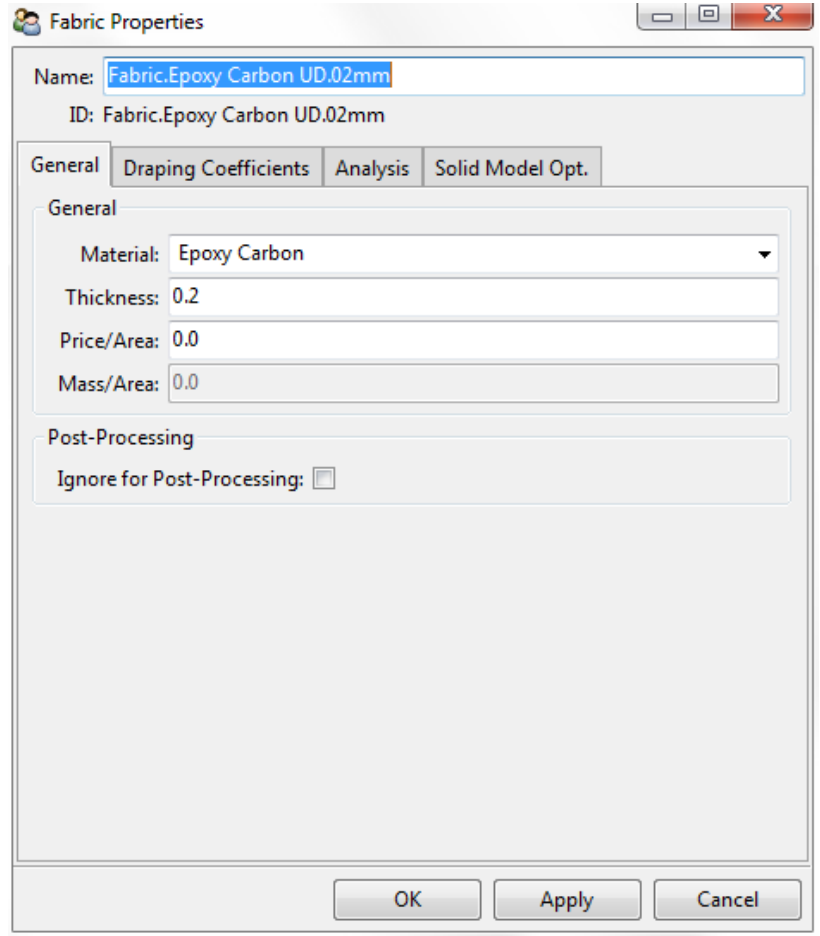
1. Select Model Cell and edit unit system of mesh output to be mm
2. Update Model
3. Refresh Setup and open ANSYS Composite PrepPost
   (Right mouse button on Setup → Edit)

2. Workshop Kiteboard



1

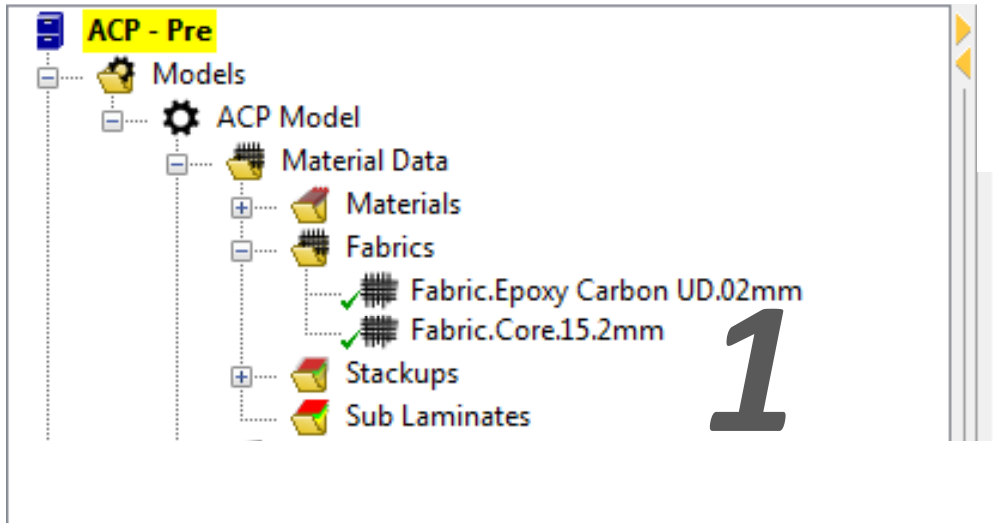


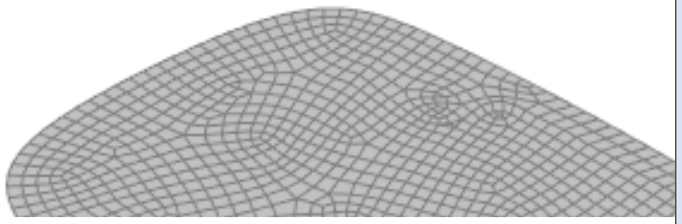


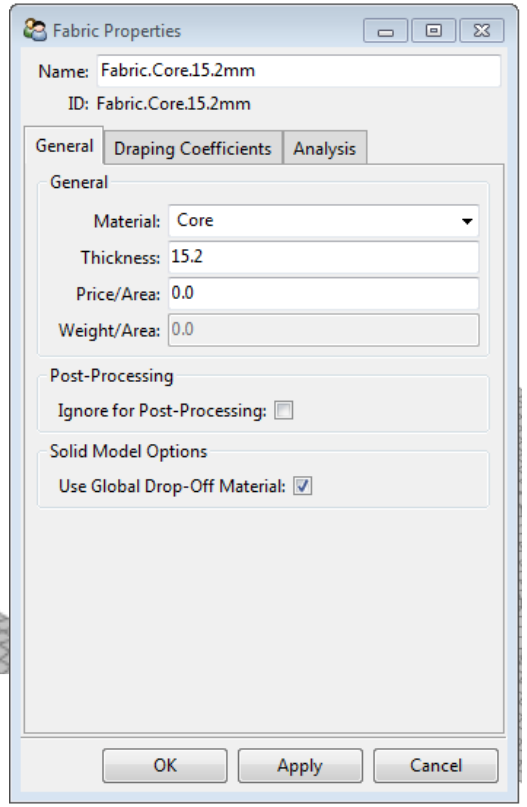
1. Right mouse button on Fabrics → Create Fabrics opens Fabric Properties window
2. Select the material for the fabric (Epoxy Carbon) and rename the Fabric for better identification
3. Define a thickness for the single layer (t = 0.2 mm)

2. Workshop Kiteboard

Fabrics





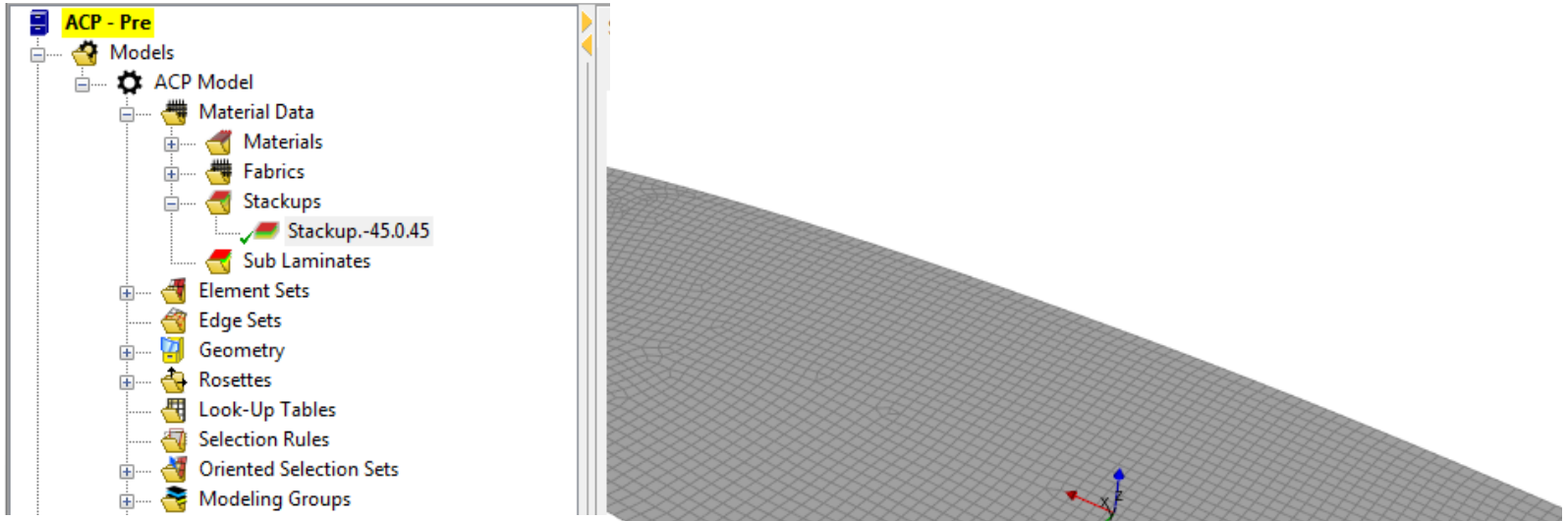


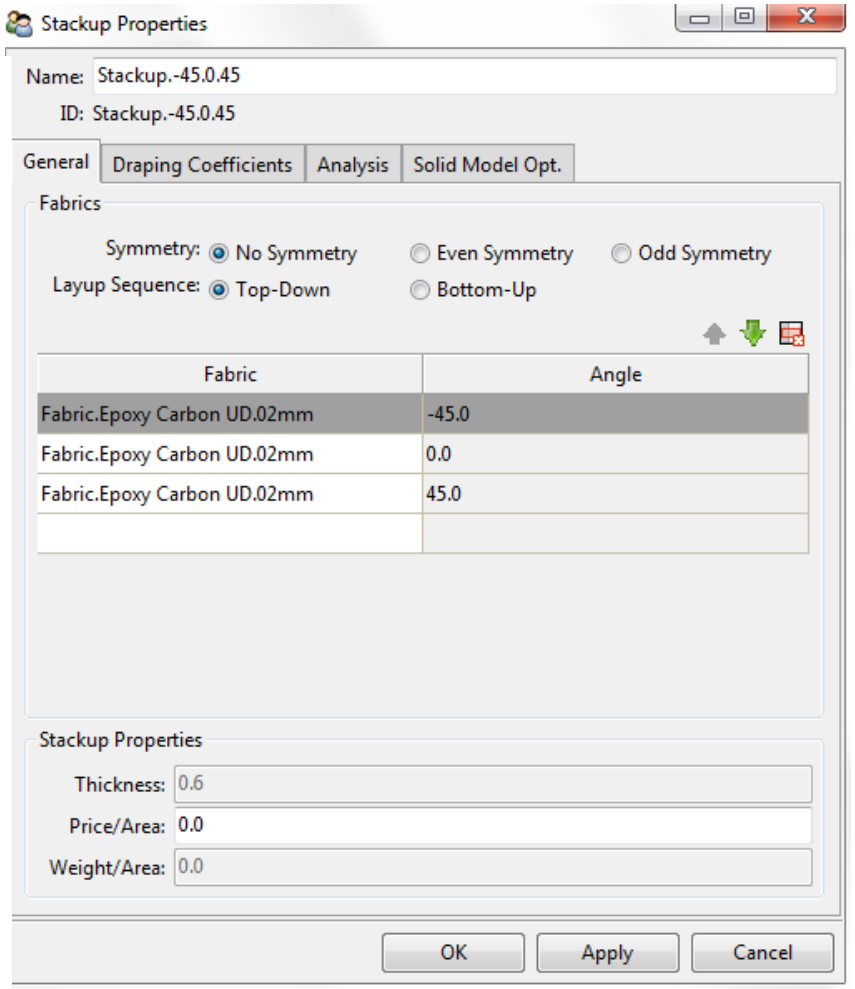
1. In addition to the UD Fabric define a core fabric, specify a thickness value of 15.2 mm, this value will change later with an imported core geometry

2. Workshop Kiteboard

- We will use a (-45°,0°,45°) stackup for the bottom and top plies.
- A stackup is a predefined combination of multiple fabrics and can be ordered from a manufacturer.
- Using stackups reduces the number of layers we have to lay down significantly since the complete stackup is placed as one material.
- The downside of using stackups is that we can not modify the stackup structure itself anymore. Ply angles of the fabrics used in the stackup are predefined.

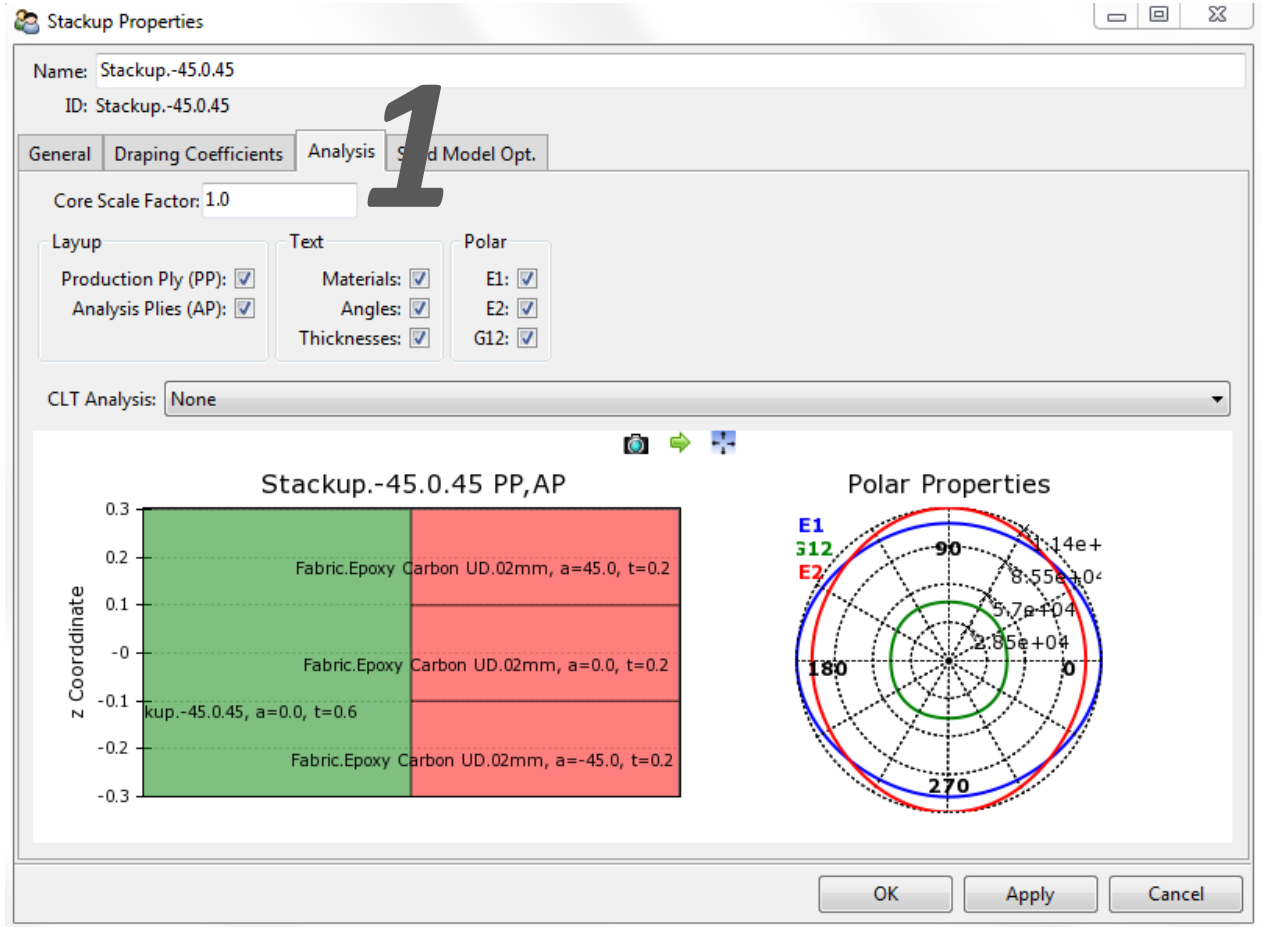
2. Workshop Kiteboard





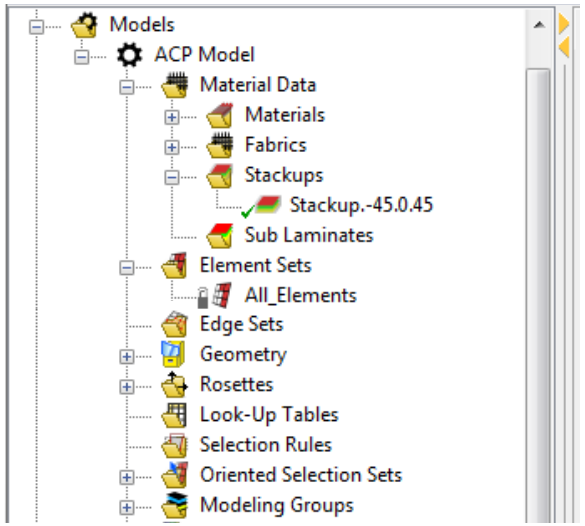
1. Right mouse button on Stackups → Create Stackup opens Stackup Properties window
2. Create a Stackup using three Epoxy Carbon UD Layer (-45°, 0°, 45°)
3. Rename the Stackup for better identification

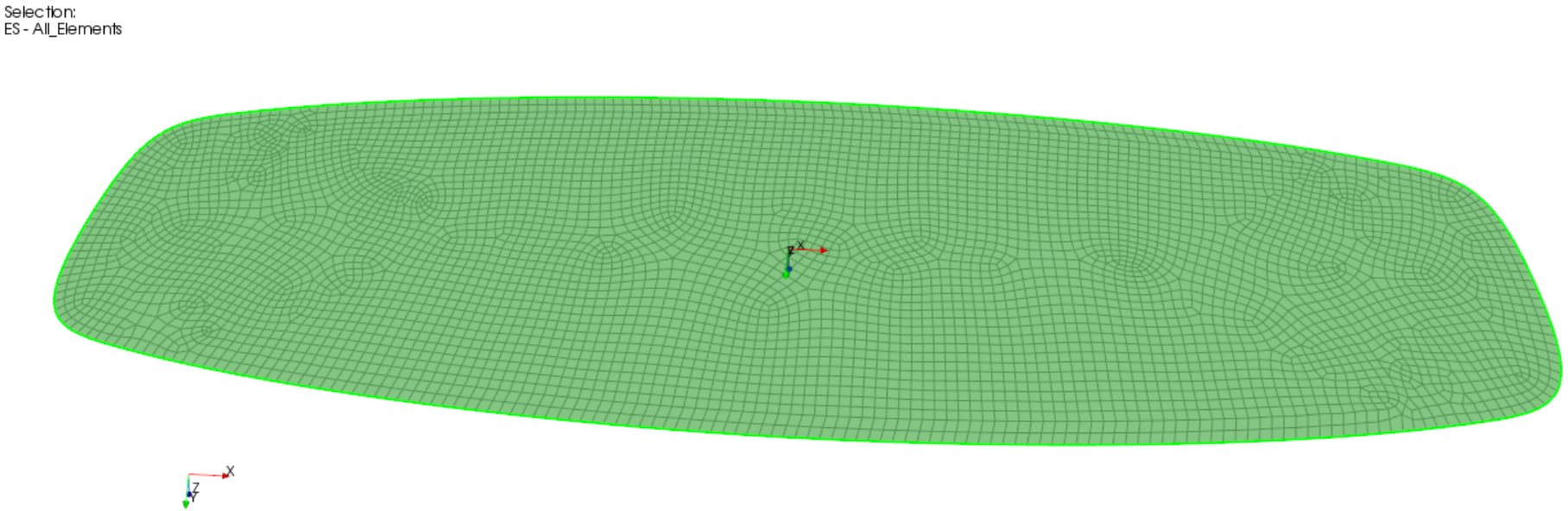
2. Workshop Kiteboard



1. Switching to the Analysis tab in the Stackup Properties window allows to check the definition of the Stackup and gives the mechanical properties in a polar plot

2. Workshop Kiteboard



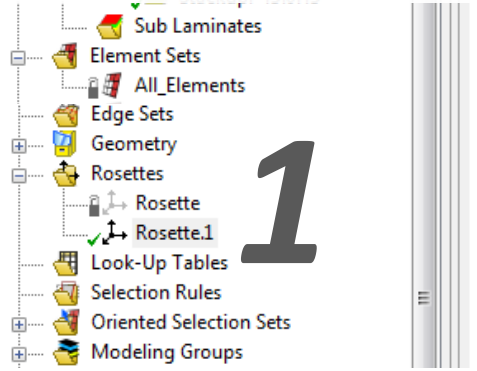


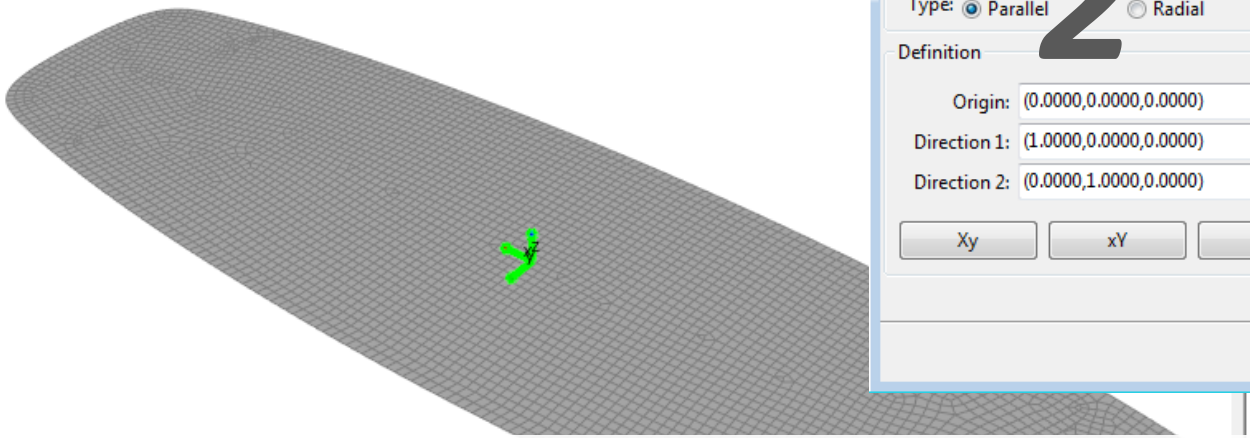
1. Check the Element Sets in the model. The Element Sets have been defined based on the Named Selections

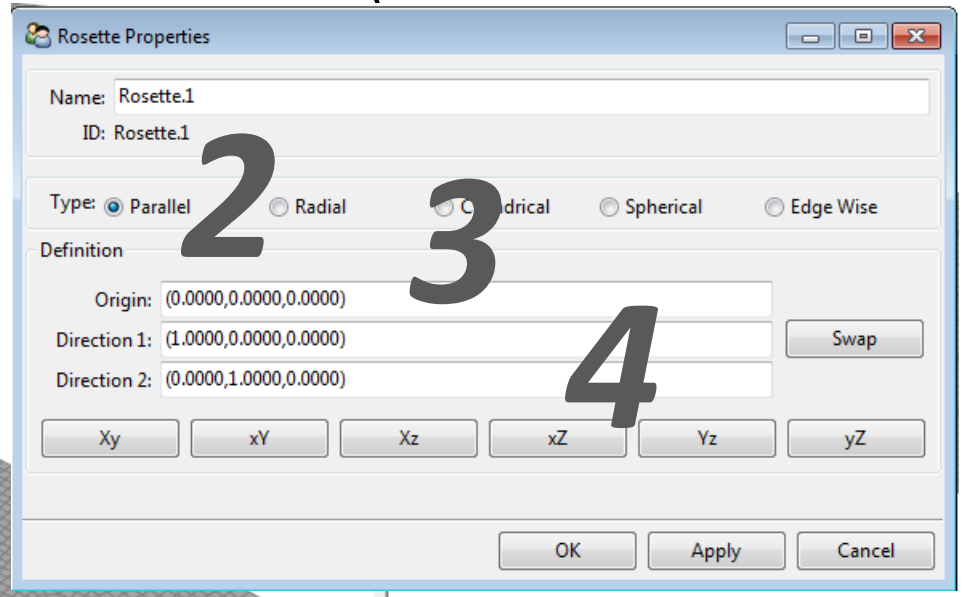
2. Workshop Kiteboard

Rosettes

1. Create a Rosette by Right mouse button on Rosettes → Create Rosette (The x-Axis of a Rosette defines the 0° fiber direction of the layup)
2. Chose Parallel Rosette





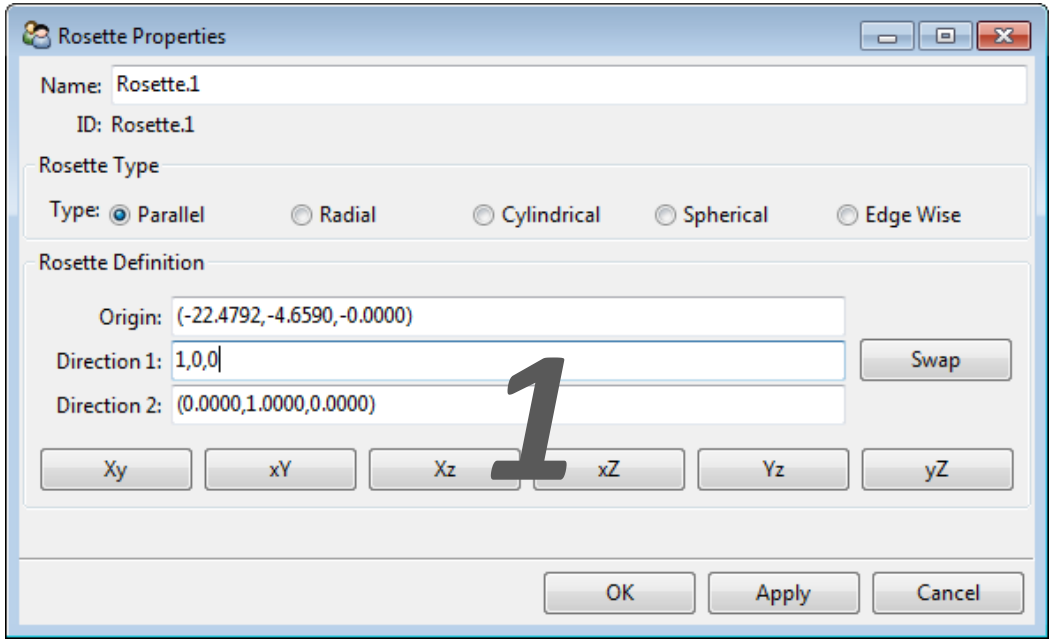


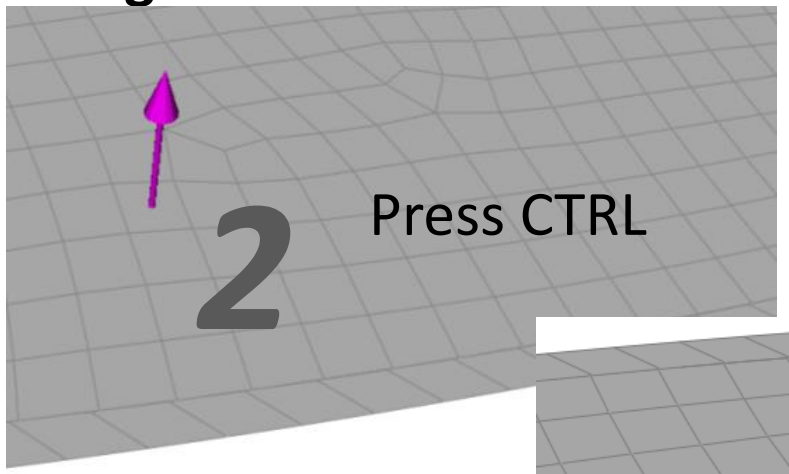
3. Define the origin of the rosette by coordinates or by clicking into the coordinate definition area and then selecting an element or a node of the model
4. Define directions 1 (1,0,0) and 2 (0,1,0) by defining a vector or by clicking in the vector definition area and selecting two elements (nodes) while pressing CTRL. The direction between both elements (nodes) describes the direction (see next slide)

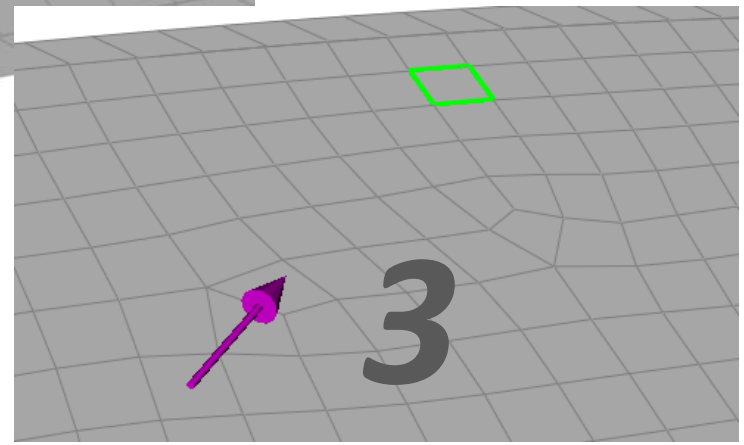
2. Workshop Kiteboard

Defining Directions for the Rosette by Picking Elements

1. Click into the Direction 1 definition area







2. Select one element as source
3. Press CTRL and select another element as target

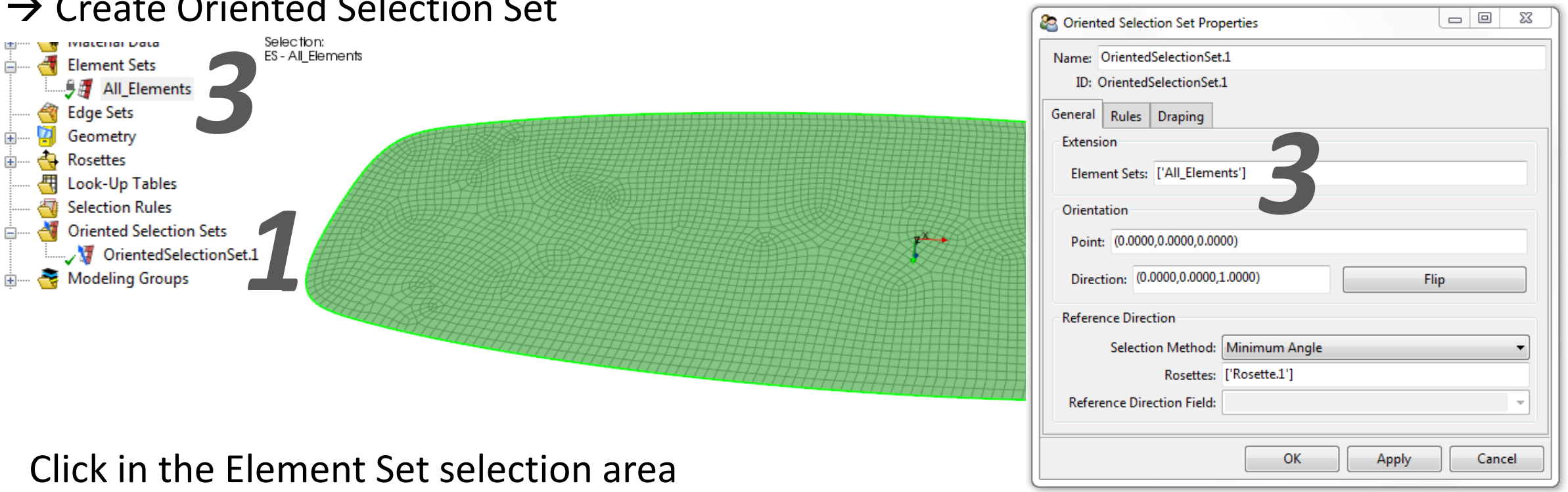
•This works whenever direction are defined in ANSYS Composite PrepPost. Similar to this you can define origins by selecting a node or an element and define normal directions simply by selecting an element.

2. Workshop Kiteboard

Oriented Selection Sets

1. Create an Oriented Selection Set by Right mouse button on Oriented Selection Sets

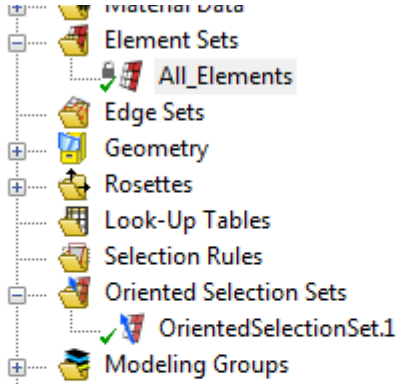
   → Create Oriented Selection Set

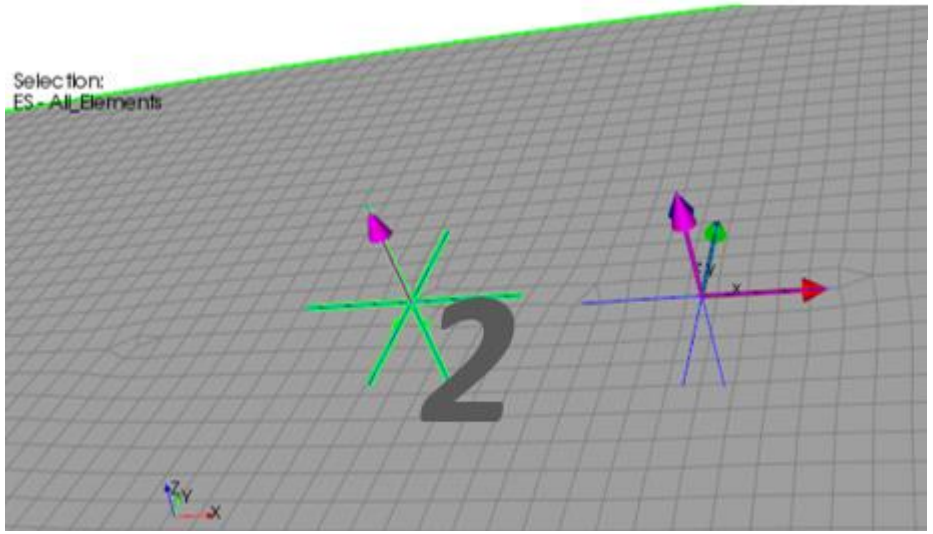


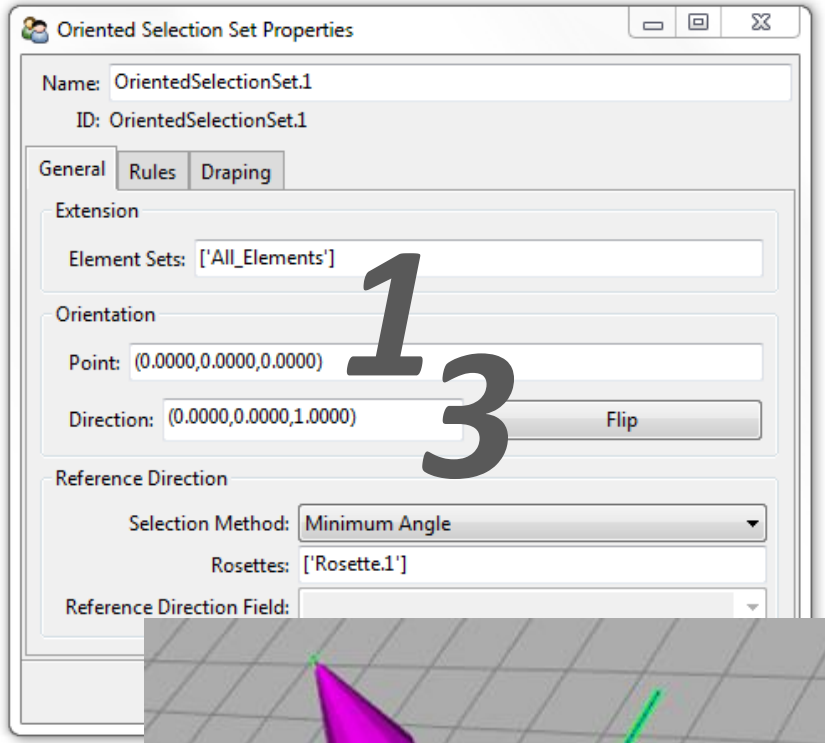
2. Click in the Element Set selection area
3. Select the Element Set All_Elements

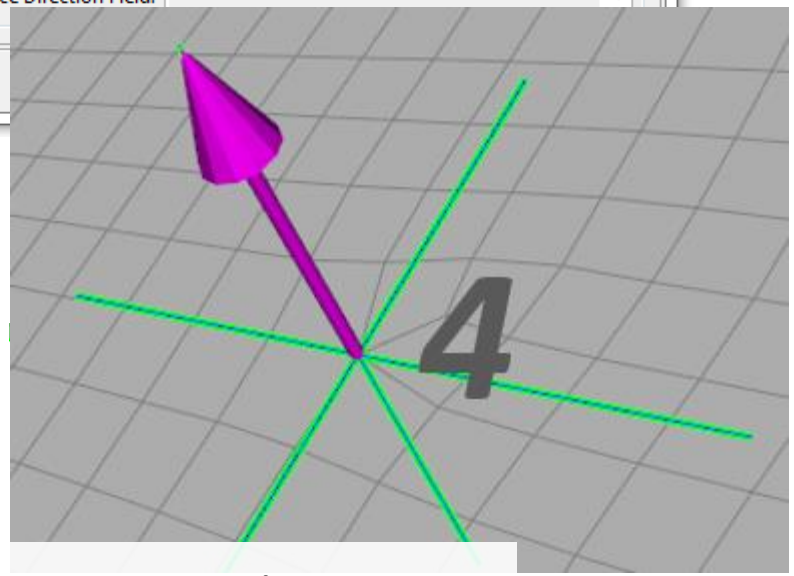
2. Workshop Kiteboard

Oriented Selection Sets





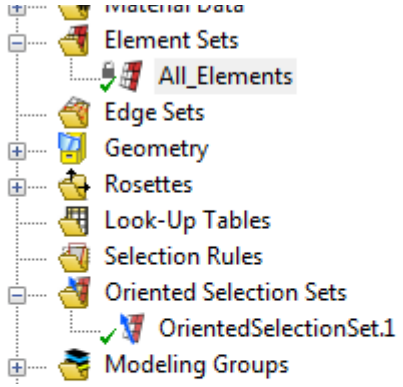


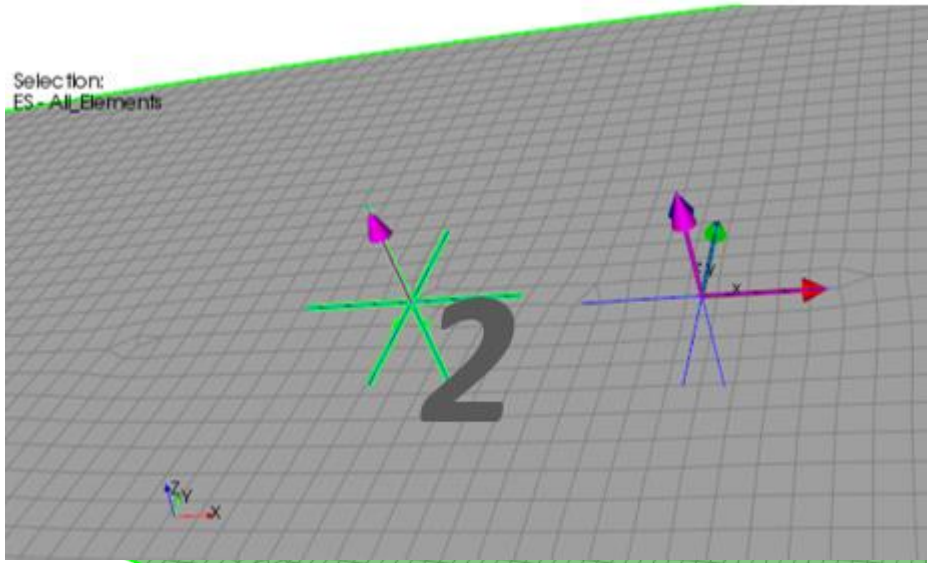


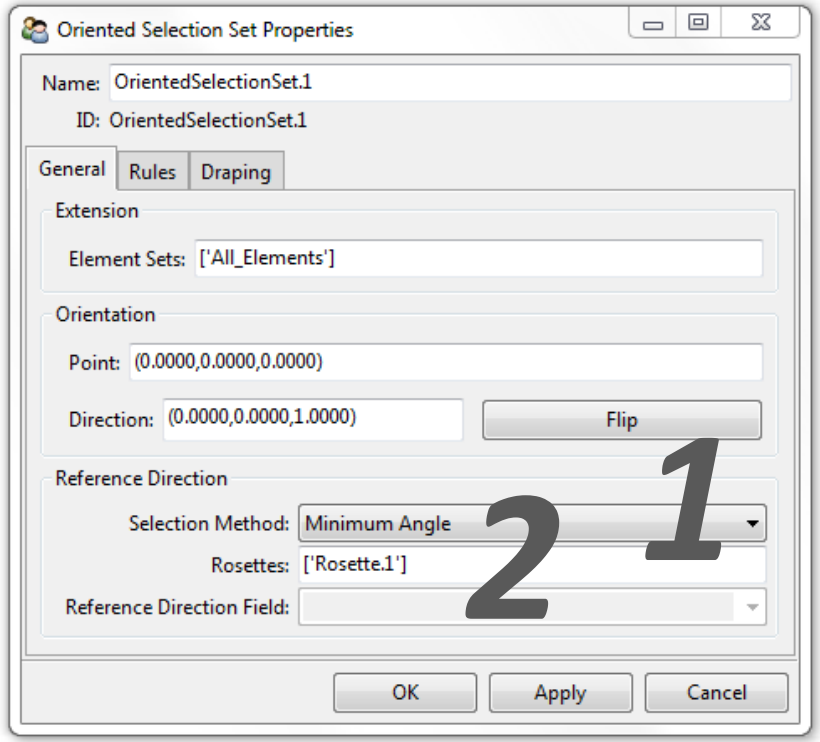
1. Click in the Orientation Point definition area
2. Select a point in the middle of the Kiteboard (For the plane kiteboard the position of the orientation point is not important, the importance of the position will be explained later)
3. Click in the Orientation Direction definition area
4. Select an element to use the element normal as orientation direction (the orientation direction defines the layup direction)

2. Workshop Kiteboard

Oriented Selection Sets





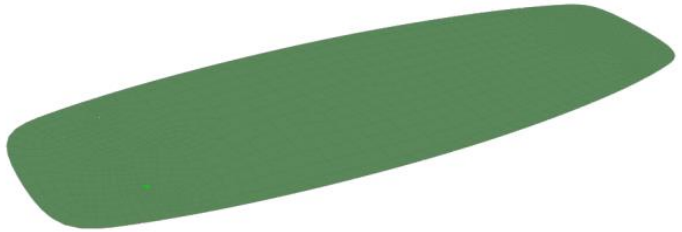


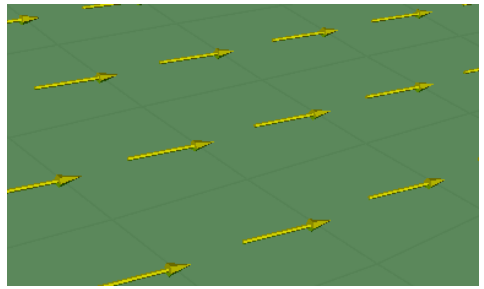
1. Select Minimum Angle as Selection Method

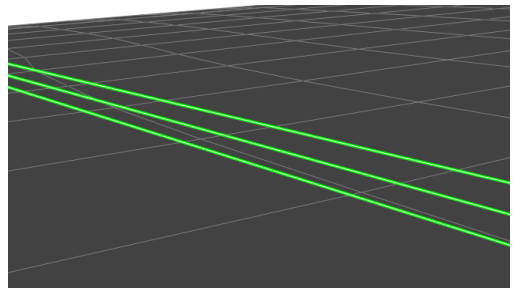
   (The selection method defines how the reference direction should be defined when multiple rosettes are selected, the different options will be explained later)
2. Select the Rosette generated before

2. Workshop Kiteboard

- Oriented Selection Sets define the basis for the layup definition. They contain the following important details for a composite layup:
  - The area which is later used to apply layers on (Selection Set)
  - The direction defining the 0° fiber direction. All defined angles are based on this direction (Rosette)
  - The direction in which layers are applied (Orientation Direction)



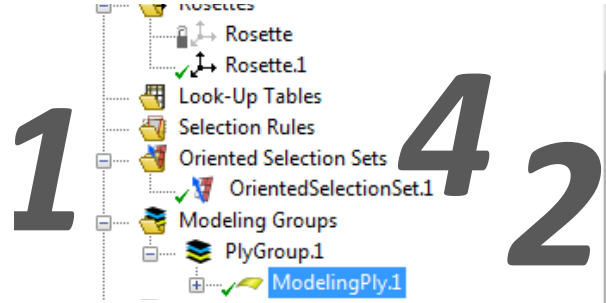


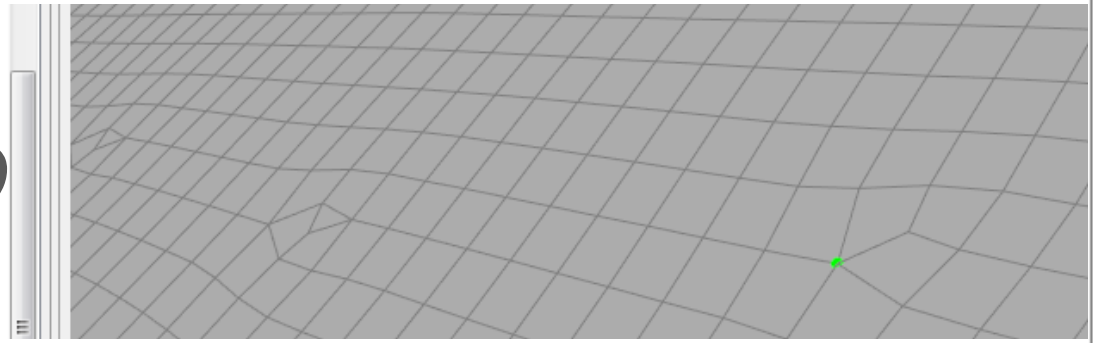


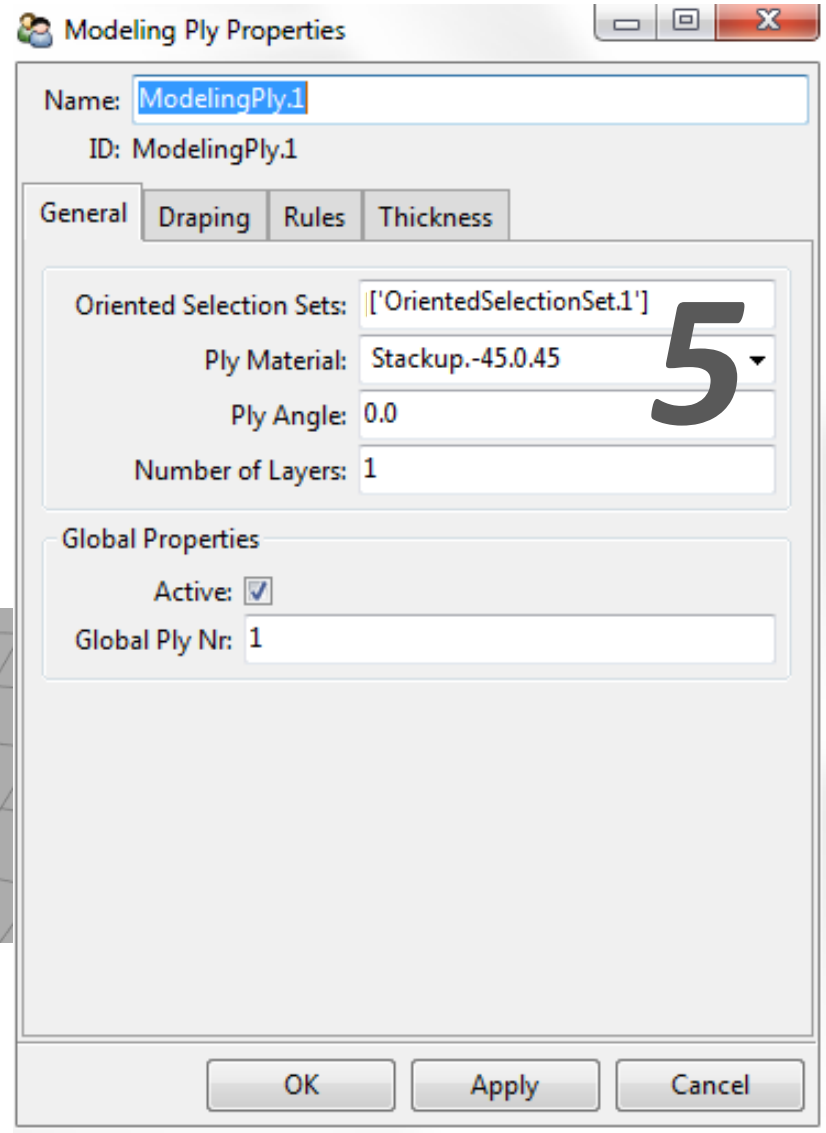
2. Workshop Kiteboard

Create a Ply

1. Create a Group by Right mouse button on
   Modeling Groups → Create Modeling Group
   (Ply Groups are used to organize the composite layup for the design)
2. Right mouse button on the Ply Group → Create Ply
   opens the Modeling Ply Properties







3. Click in the Oriented Selection Sets selection area
4. Select the oriented selection set created before
5. Select the created Stackup as Ply Material (you can
   select single Fabrics, Stackups and Sublaminates here)

2. Workshop Kiteboard

- **Modeling Plies**

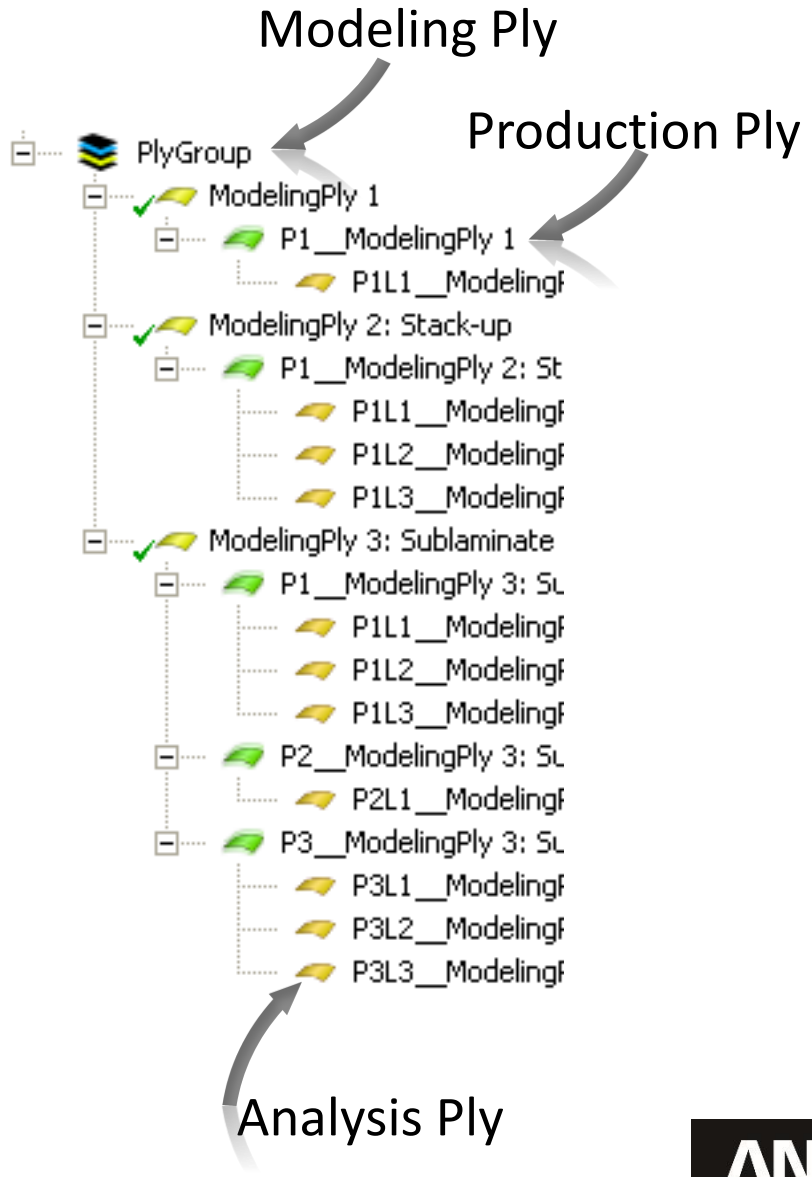
  Plies are defined at the Modeling Ply level in ANSYS Composite PrepPost. Fabric selection, oriented selection sets and draping is defined one the Modeling Ply level.

- **Production Plies**

  The production ply level describes plies as they are used in manufacturing. Stackups will be seen at the production ply level.

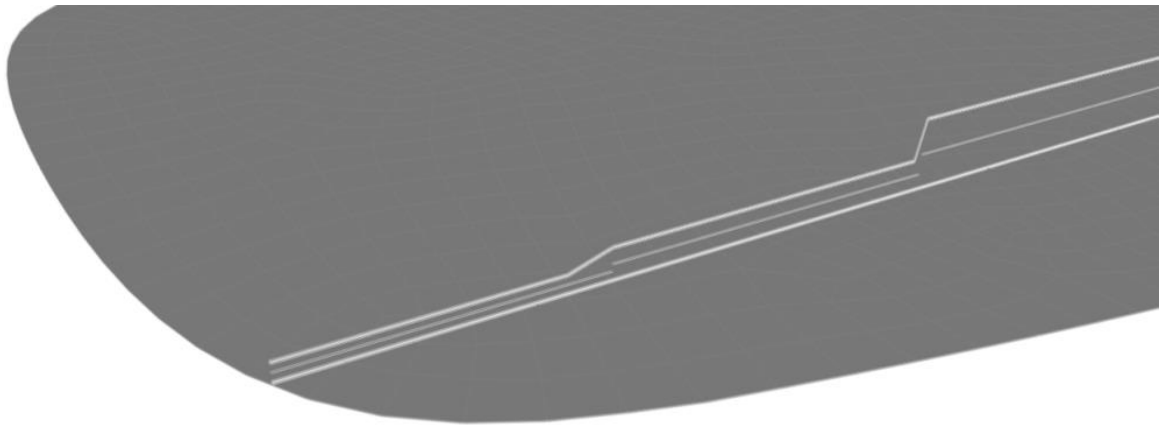
- **Analysis Plies**

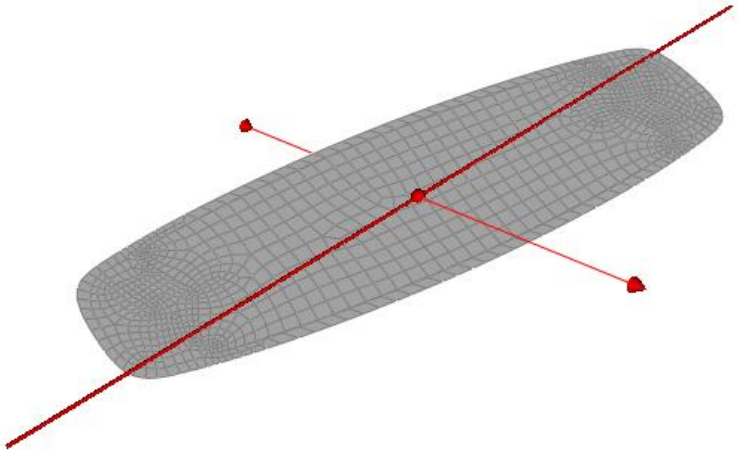
  At this level all analysis plies are shown as used in the simulation and available for postprocessing.



2. Workshop Kiteboard

- Section Cuts are used to check the layup we defined
- Section cuts are showing the layers in a section cut plane
- The location and direction of the section cut plane can either be specified by defining an origin and a normal direction or interactively by dragging and rotating

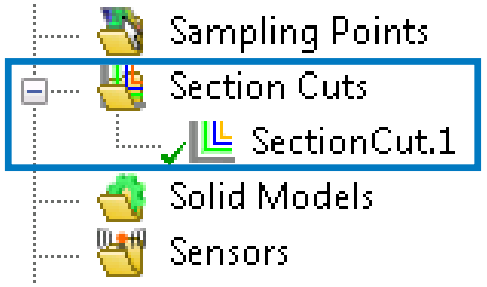


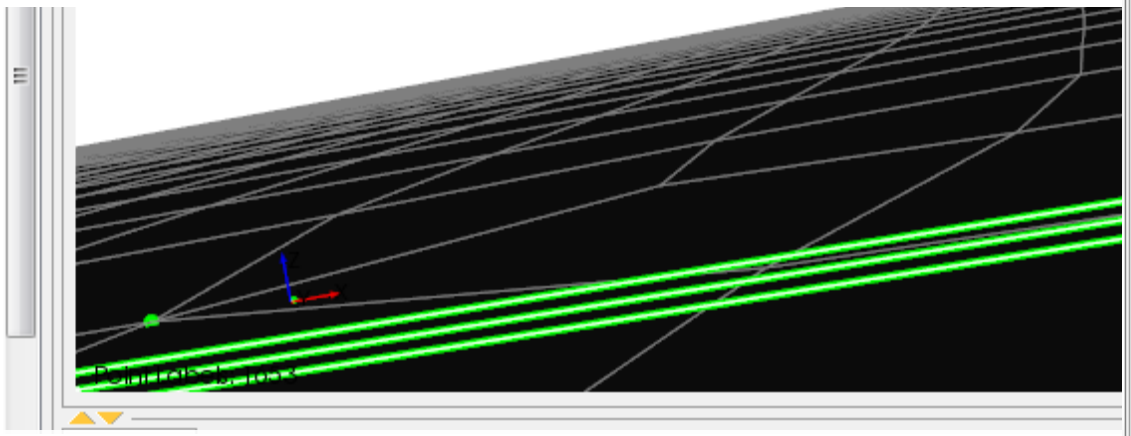


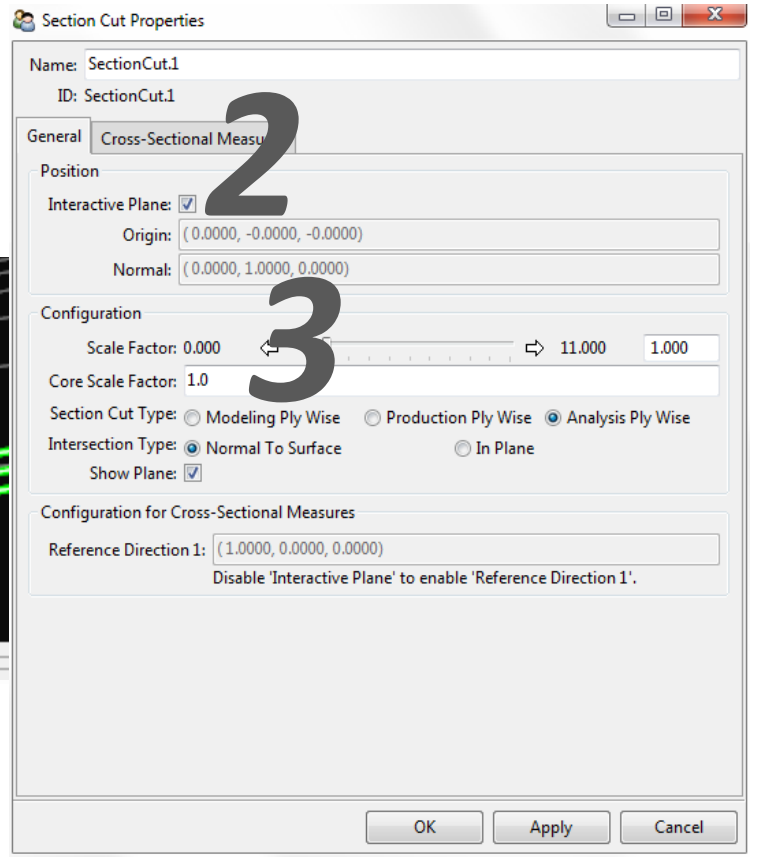
2. Workshop Kiteboard

Insert Section Cuts and Define Location and Orientation by Input

1. Right mouse button on Section Cuts → Create Section Cut
   opens the Section Cut Properties window



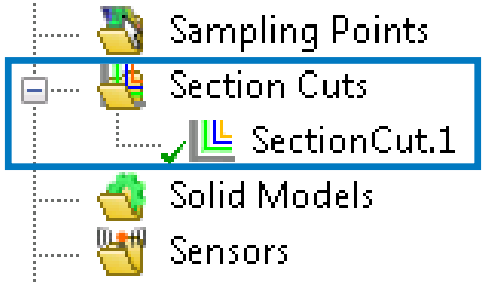


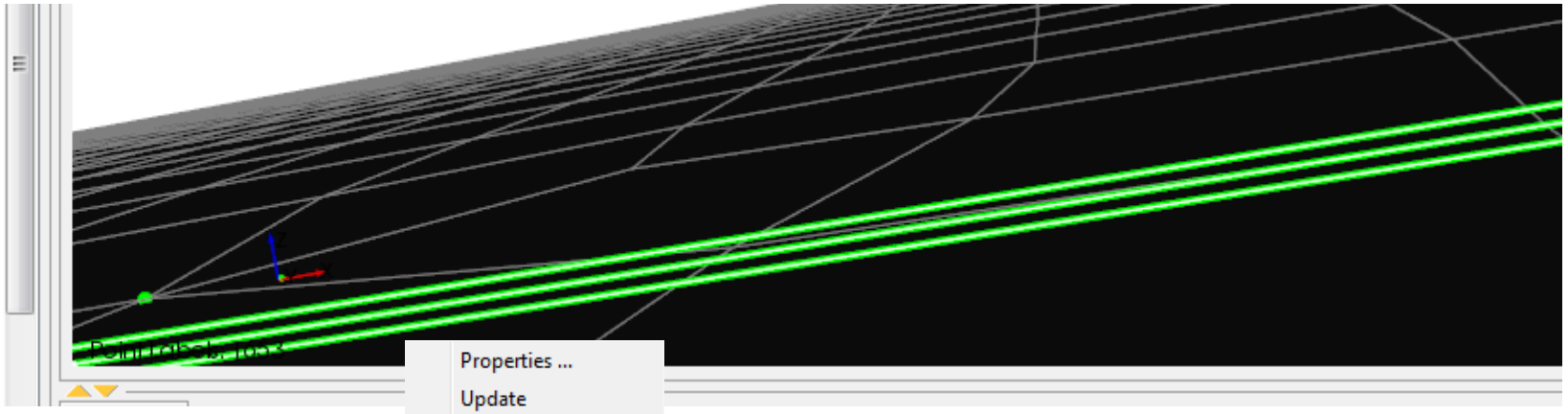


2. Unselect Interactive Plane and type in Origin (0,0,0) and Normal (0,1,0)
3. Select Analysis Ply Wise and define a core scale factor of 1.0
4. You can also scale the section thicknesses in the view by scaling all plies and/or just core plies

2. Workshop Kiteboard

Section Cuts

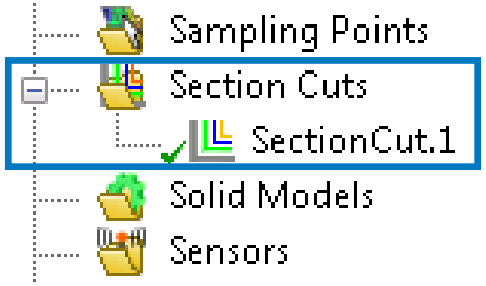


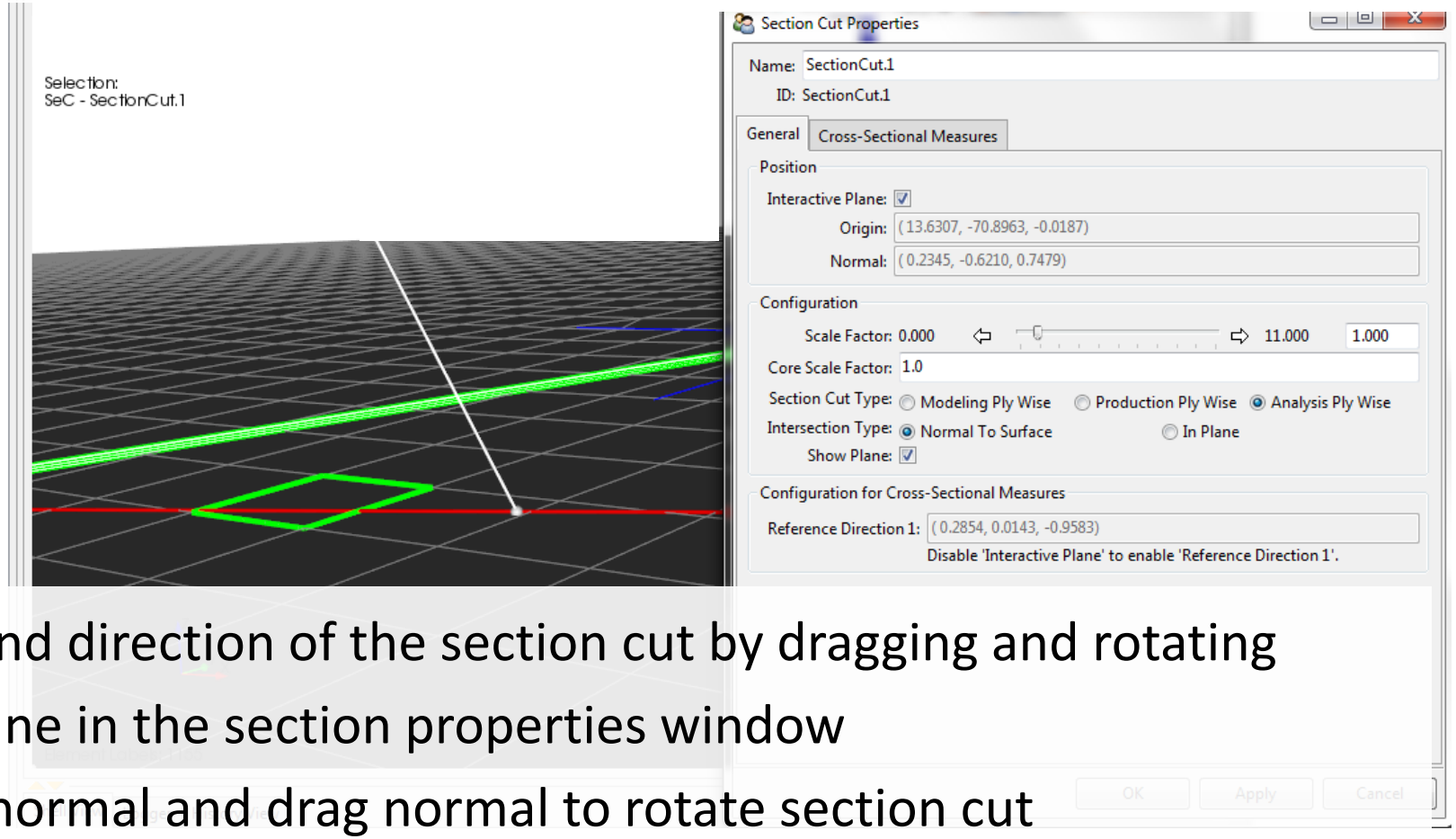


1. Check the Layup using Section Cuts
2. Multiple Section Cuts can be used
3. Use Show / Hide in Pop Up Menu

2. Workshop Kiteboard

Interactive Section Cuts

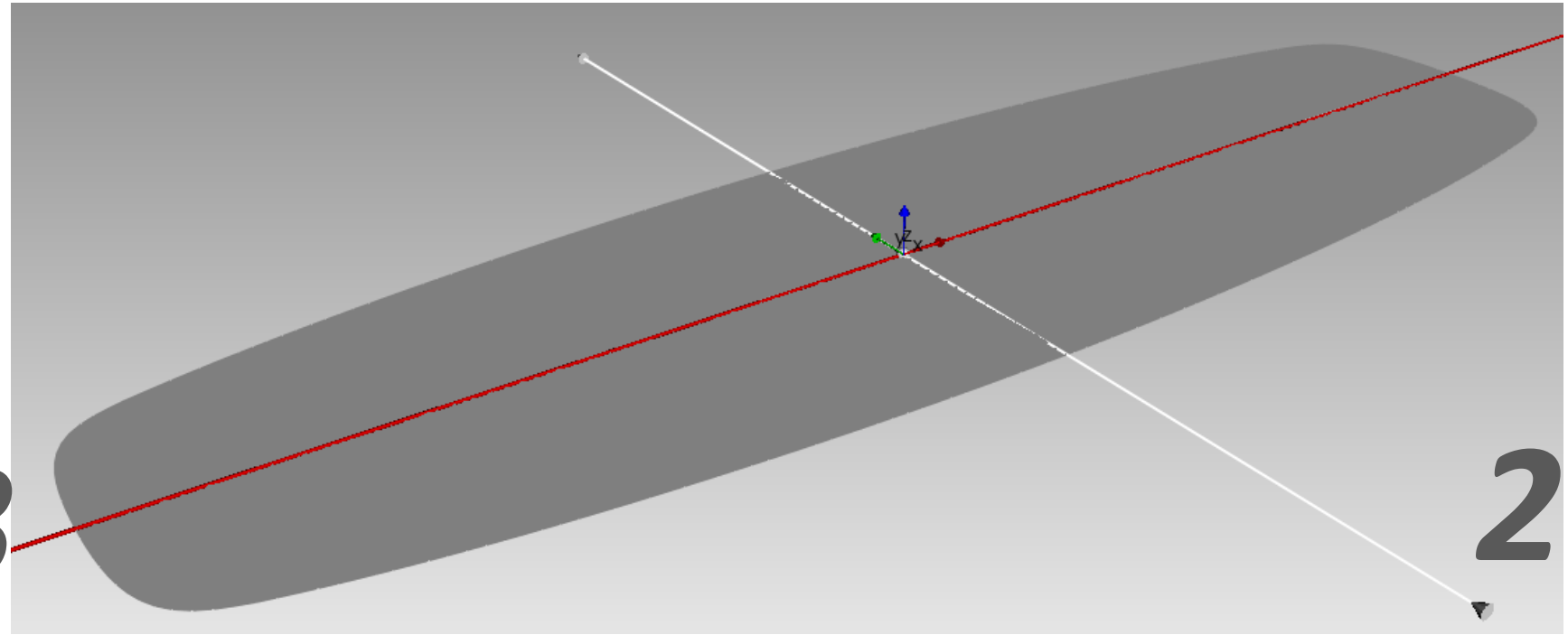




1. To modify location and direction of the section cut by dragging and rotating select interactive plane in the section properties window
2. Click on section cut normal and drag normal to rotate section cut
3. Click on section cut plane and drag plane in normal direction to modify the location of the plane

2. Workshop Kiteboard

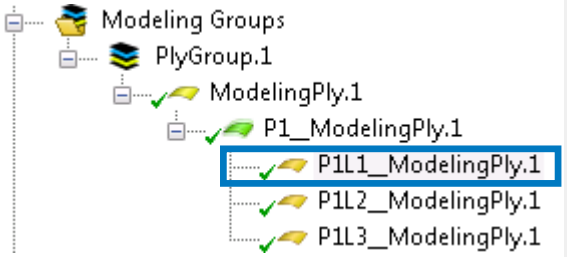
Interactive Section Cuts

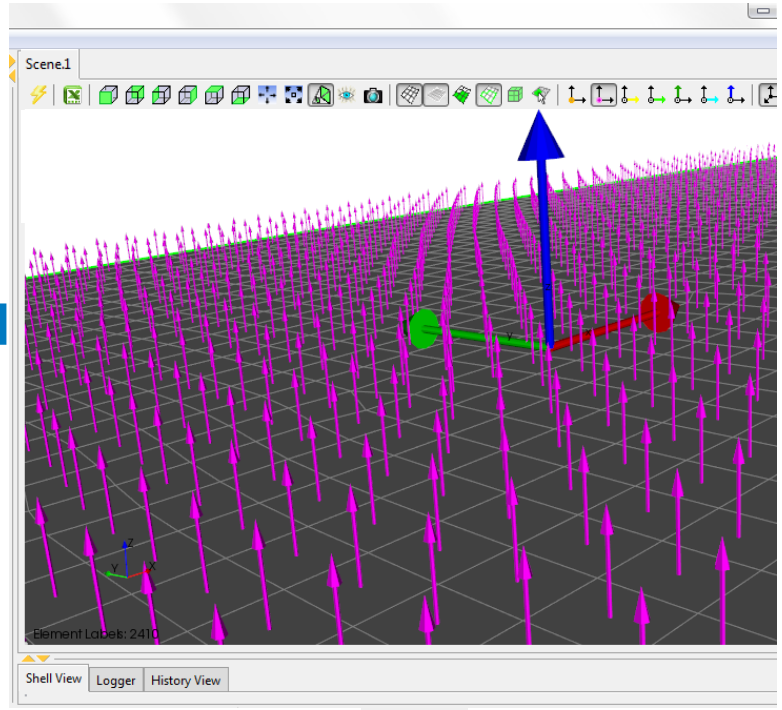


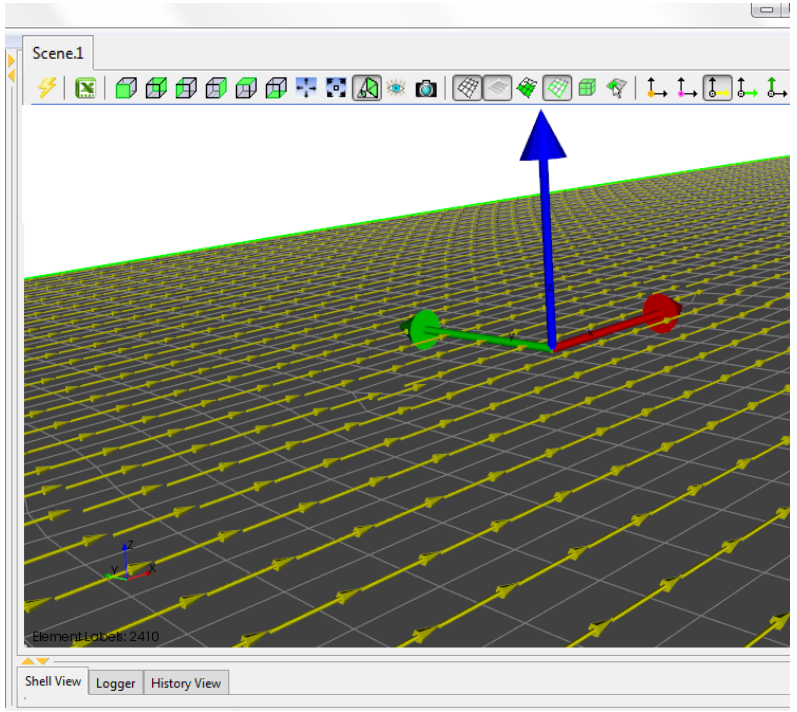
1. To modify location and direction of the section cut by dragging and rotating select interactive plane in the section properties window
2. Click on section cut normal and drag normal to rotate section cut
3. Click on section cut plane and drag plane in normal direction to modify the location of the plane

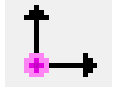
2. Workshop Kiteboard

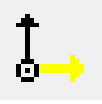
- Show Layup Direction and Reference Direction





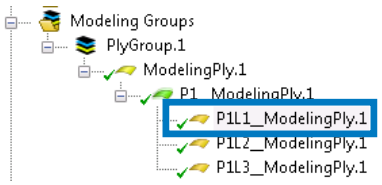


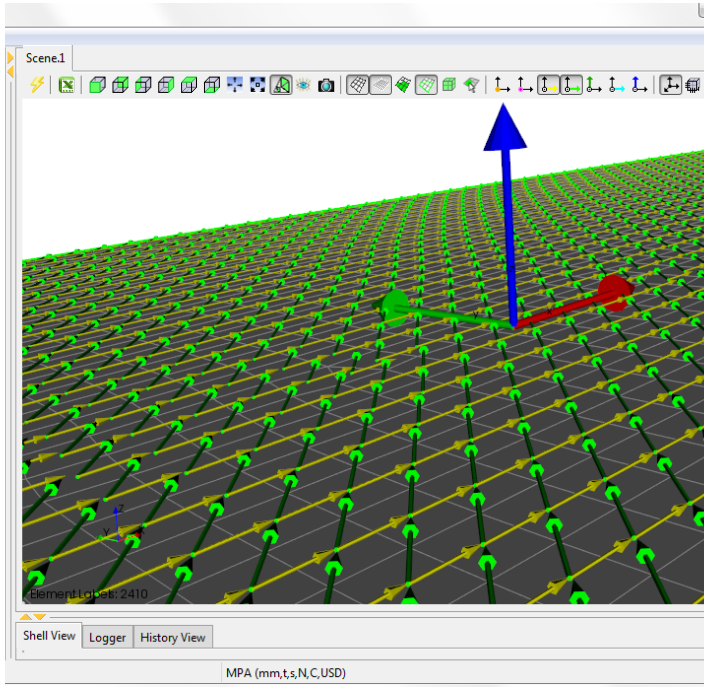
- Highlight Ply and Click on the Show Orientations icon in the scene toolbar 
- If orientations for the ply are not shown reselect the ply in the tree

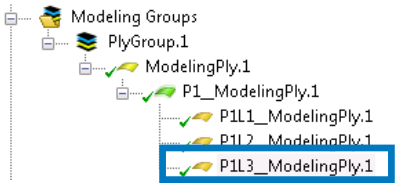
- Click on the Show Reference Direction icon in the scene toolbar 
- If reference directions for the ply are not shown reselect the ply in the tree

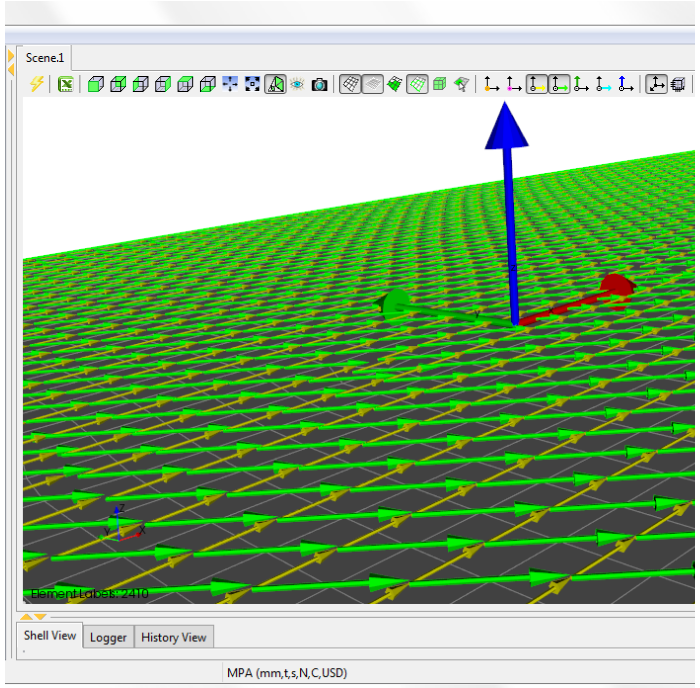
2. Workshop Kiteboard

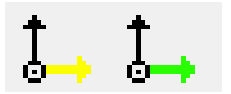
- Show Fiber Direction







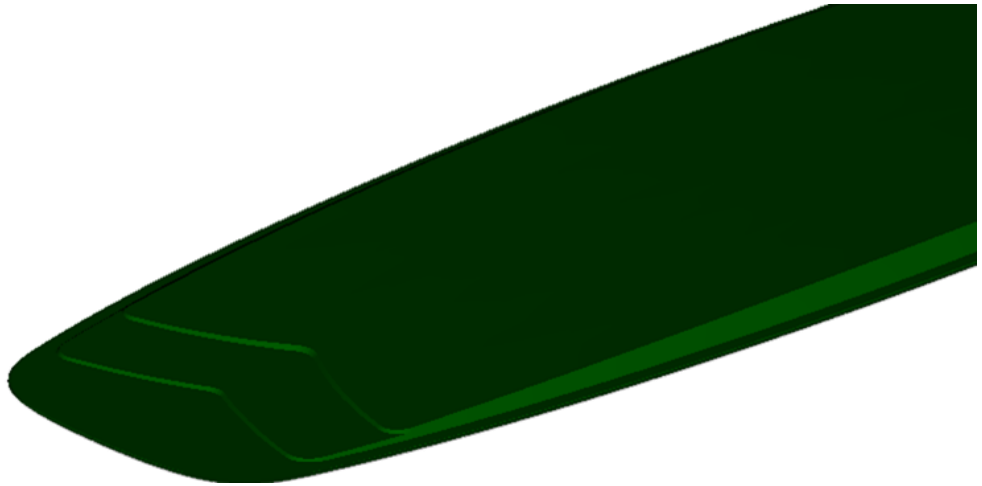


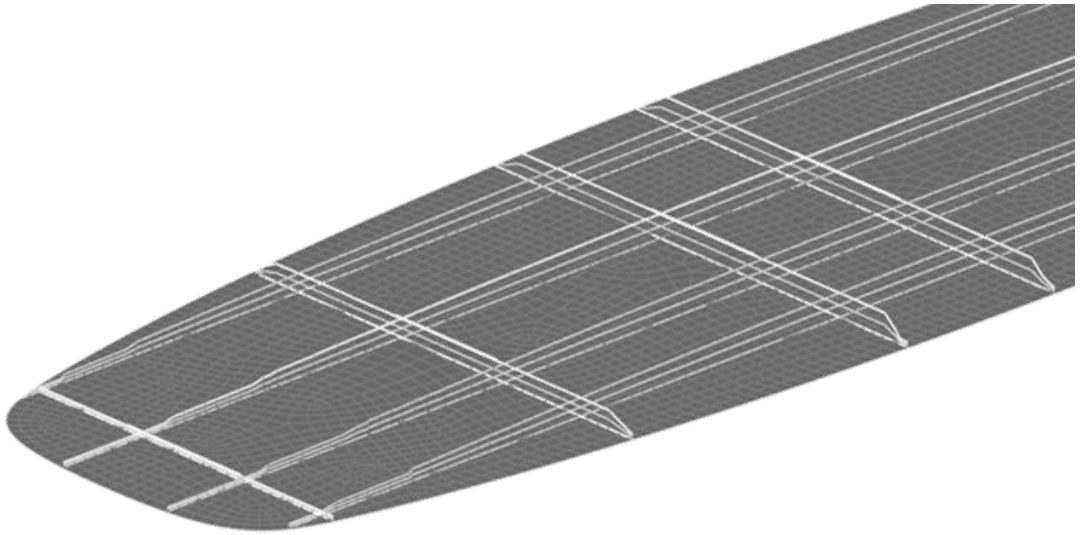
- Click on the Show Fiber Directions icon in the scene toolbar. 
- If fiber directions for the ply are not shown reselect the ply in the tree

2. Workshop Kiteboard

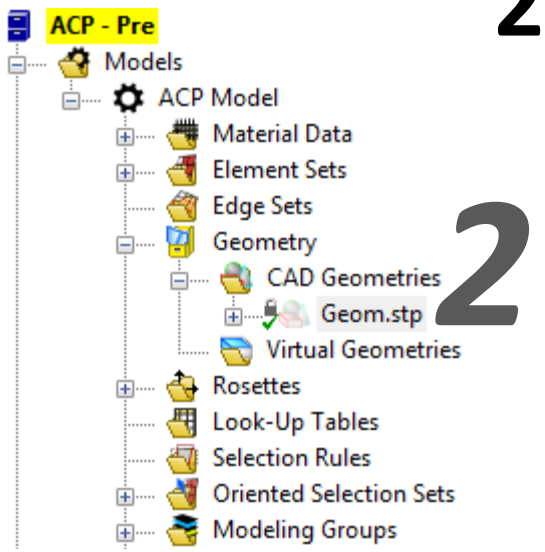
- **Varying Core Thickness**

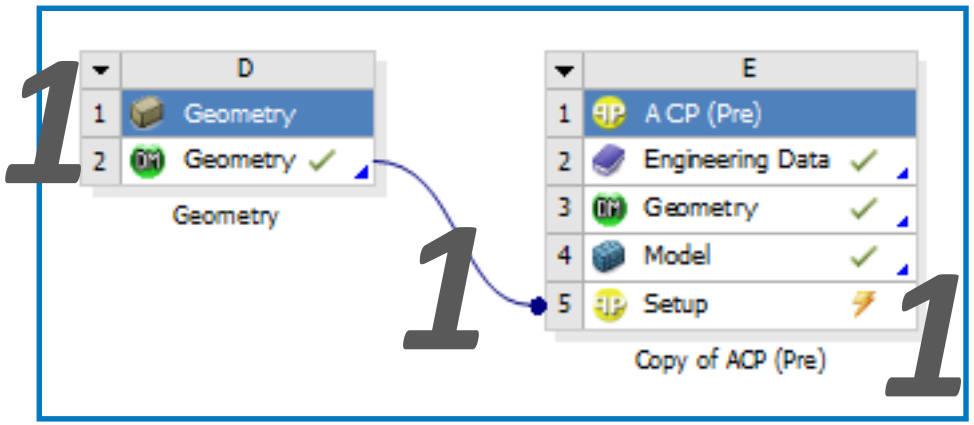
Core Thickness can be defined using CAD data in ANSYS Composite PrepPost





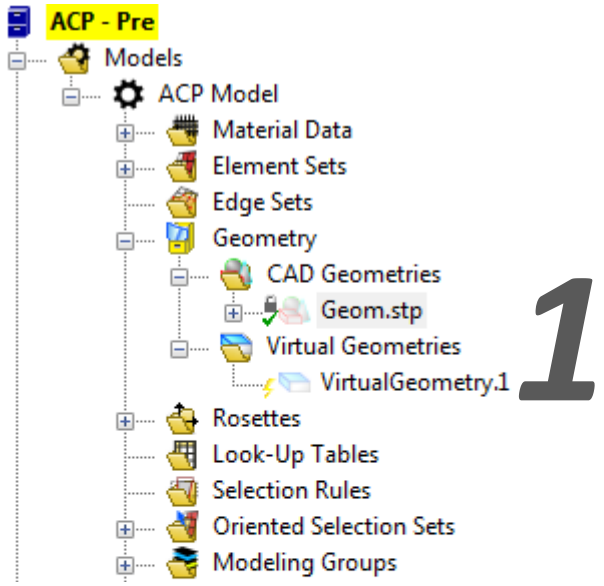
2. Workshop Kiteboard

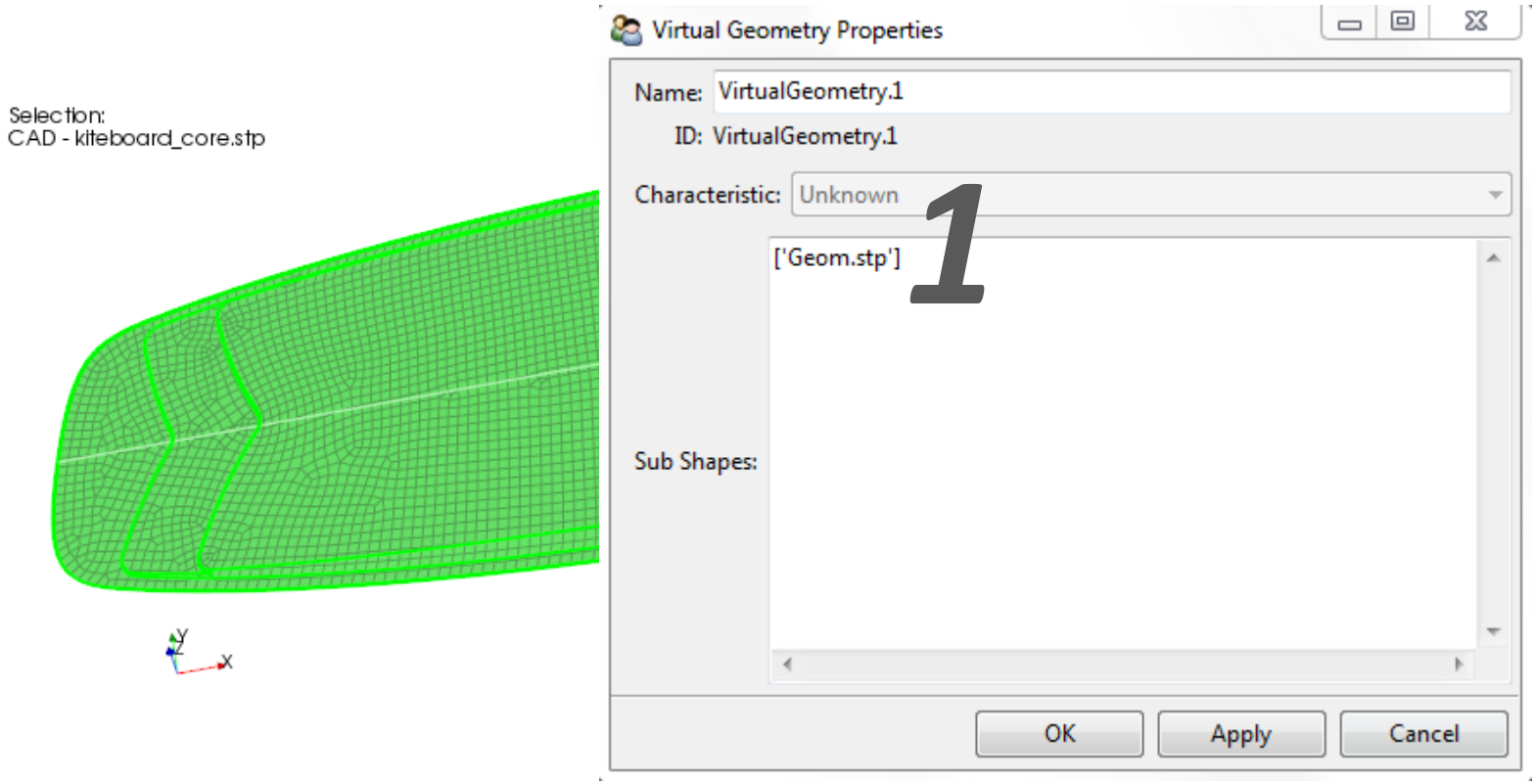




1. Add Geometry Object on **Project Schematic**, update *ACP setup* to import kiteboard_core.stp
2. Check geometry object in tree of **ACP**

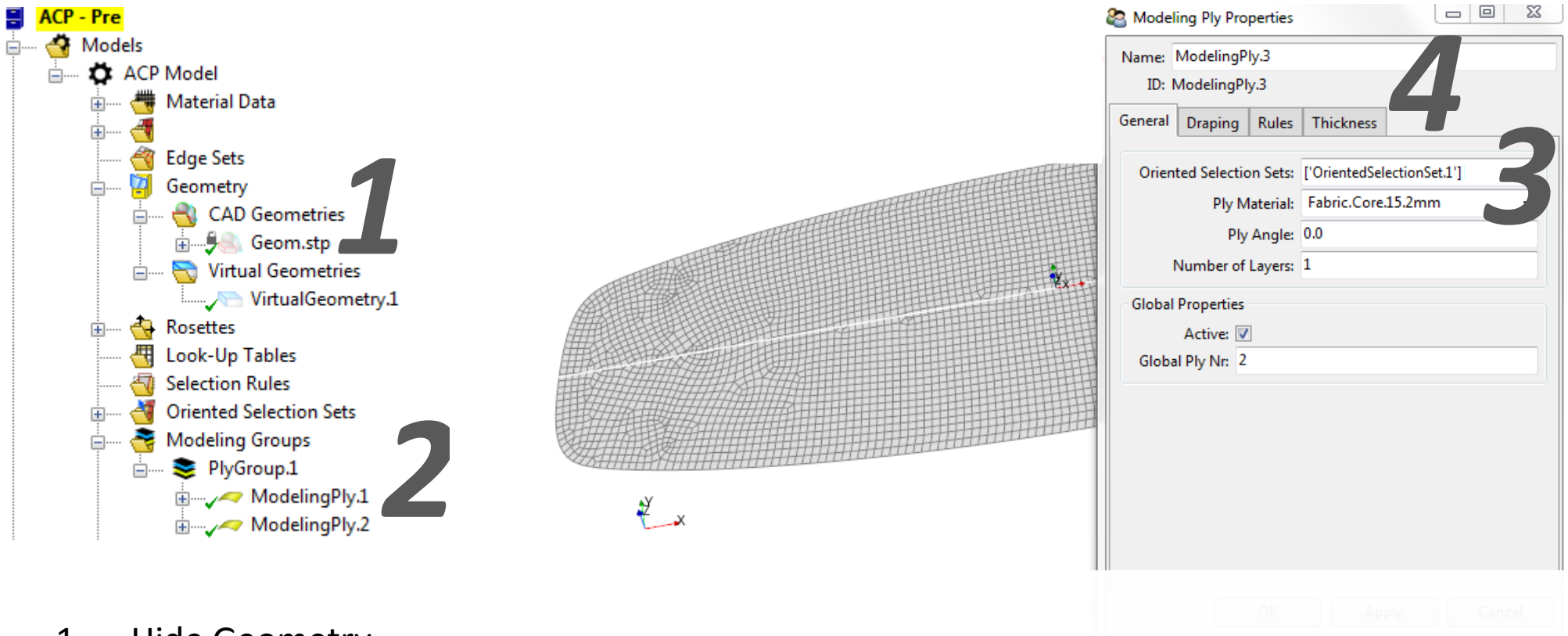
2. Workshop Kiteboard





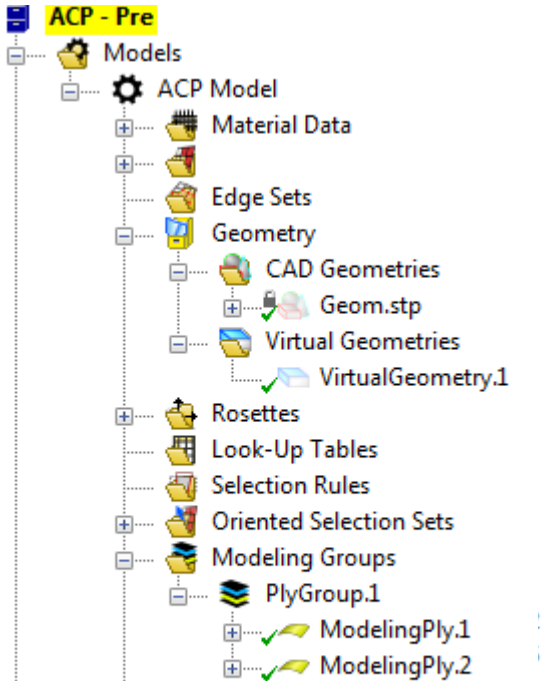
1. In the ACP tree create a Virtual Geometry selecting the previous CAD Geometry as Sub Shape

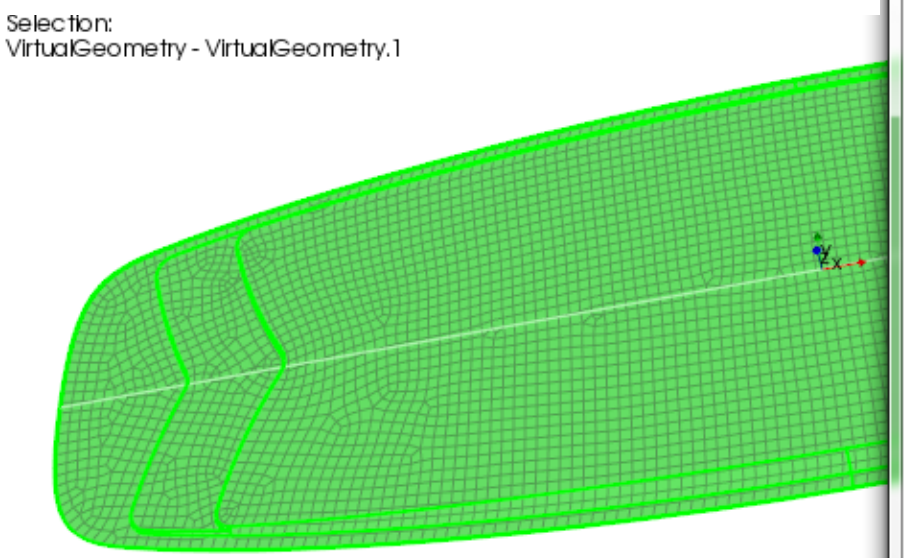
2. Workshop Kiteboard

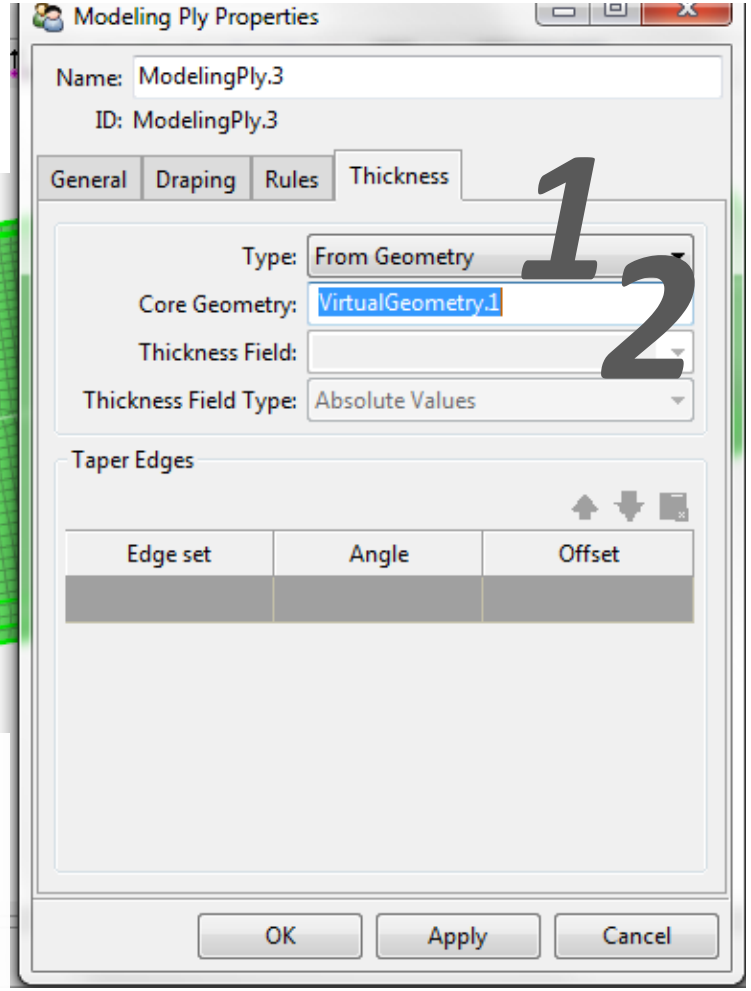


1. Hide Geometry
2. Select the first Stackup Modeling Ply in the Modeling Groups section → Right mouse button → Create Ply after
3. Select Oriented Selection Set containing all elements and the defined core fabrics as ply material
4. Switch to tab Thickness

2. Workshop Kiteboard





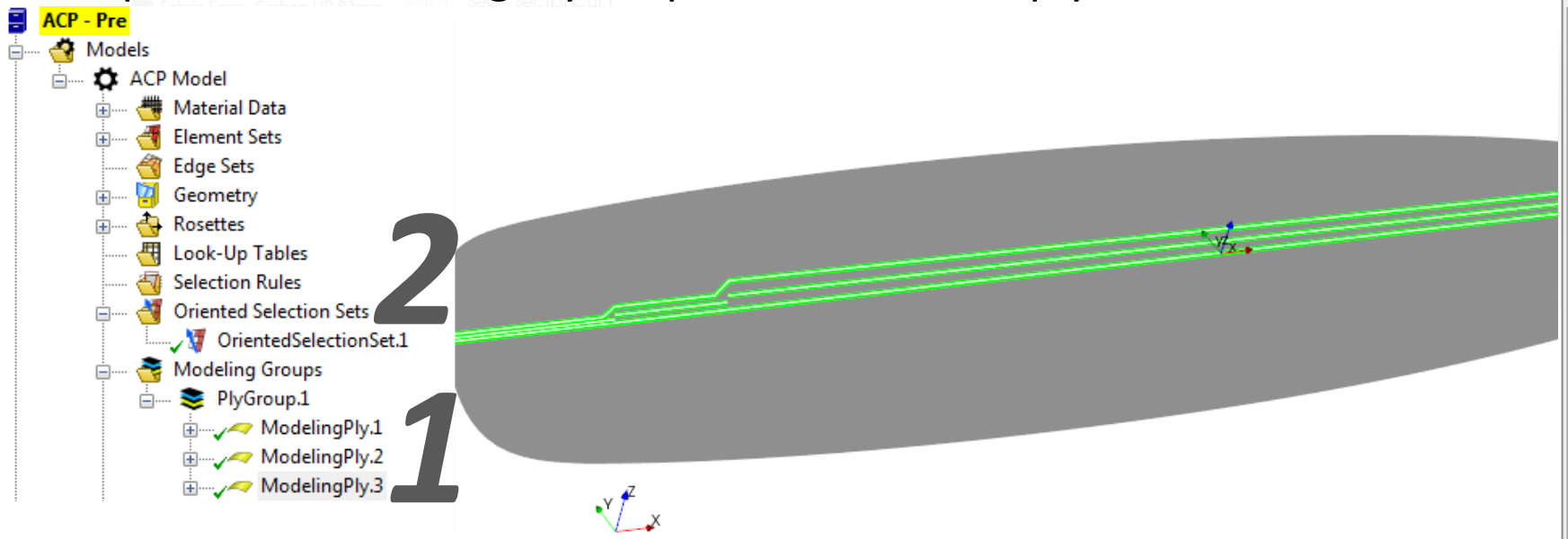


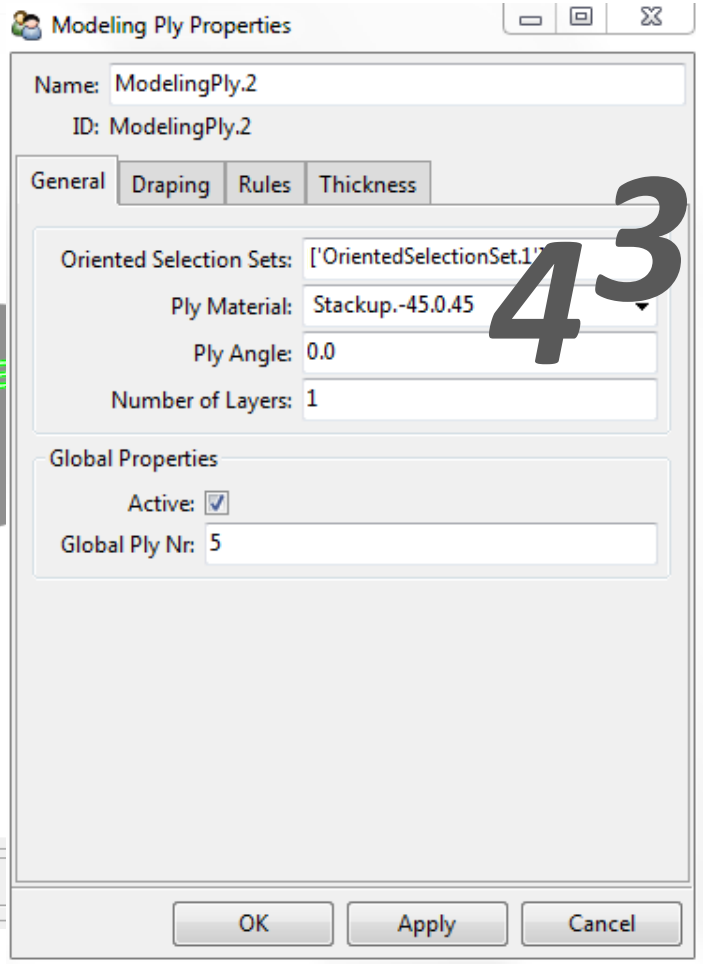
1. Switch Type to From Geometry
2. Select the Virtual Geometry created before as Core Geometry
3. Update the model to see the new layer in the section cut

2. Workshop Kiteboard

Create the Top Layer

1. Right mouse button on the existing Ply Group → Create Ply opens the Modeling Ply Properties for a new ply



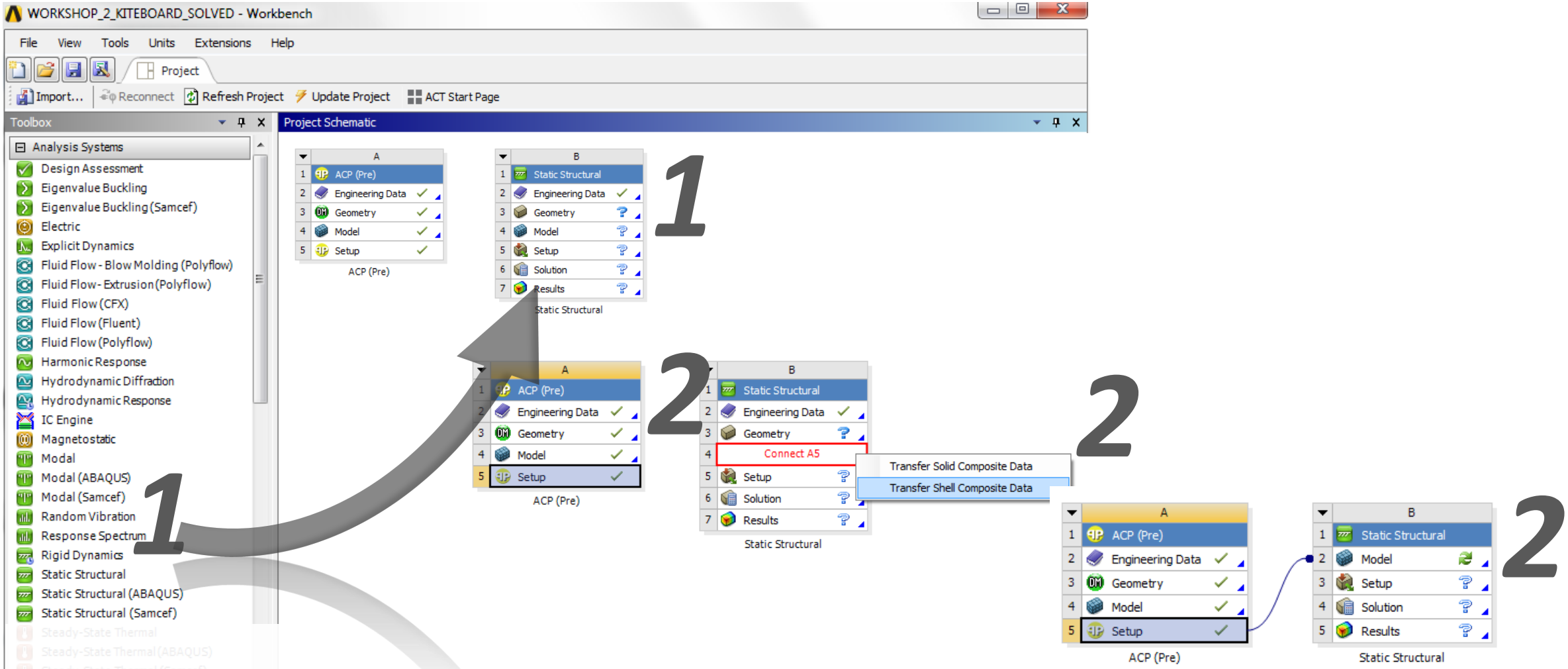


2. Click in the Oriented Selection Sets selection area
3. Select the Oriented Selection Set created at the beginning, this step defines the top layers covering the core material
4. Select the Stackup as Ply Material and update the model

2. Workshop Kiteboard

- We have completed the composite layup definition. The composite preprocessing part of the analysis is therefore completed.
- As a next step we have to add boundary conditions and loads to our model.

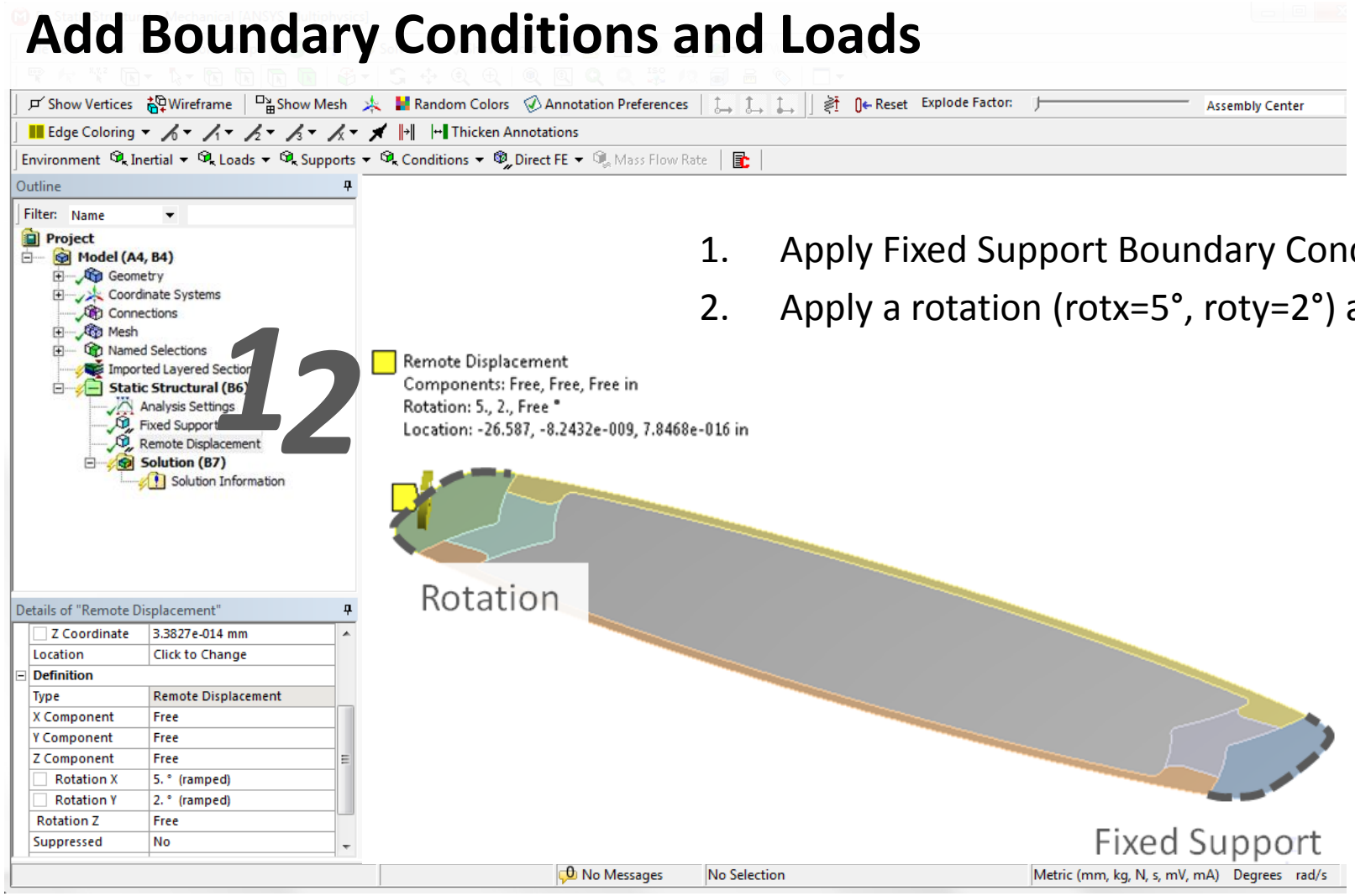
2. Workshop Kiteboard



1. Drag and Drop a Static Structural Analysis onto the project page
2. Link *Setup* of ACP (Pre) to *Model* of Static Structural (select Transfer Shell Composite Data), refresh and edit *Model* of Static Structural

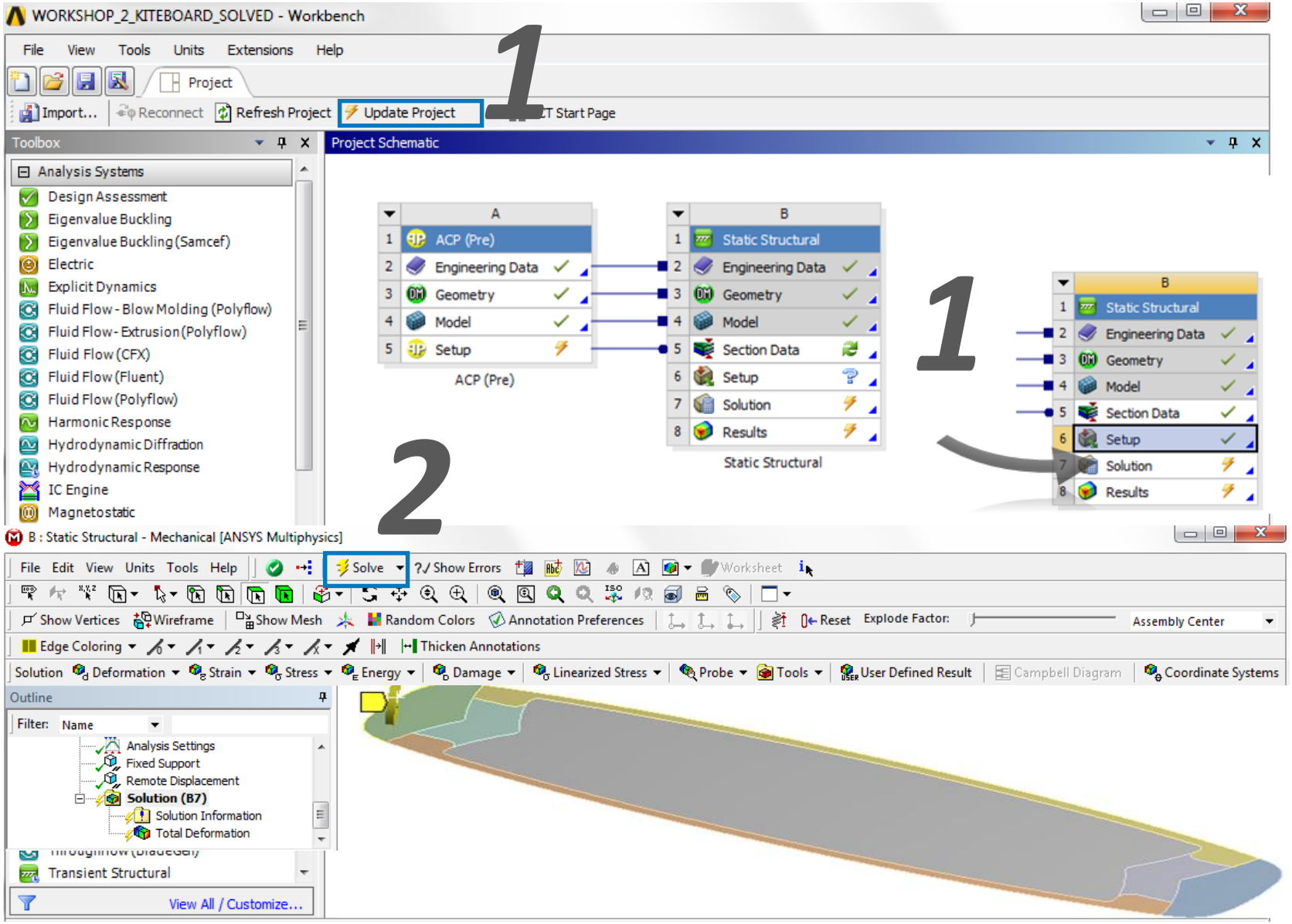
2. Workshop Kiteboard

Add Boundary Conditions and Loads



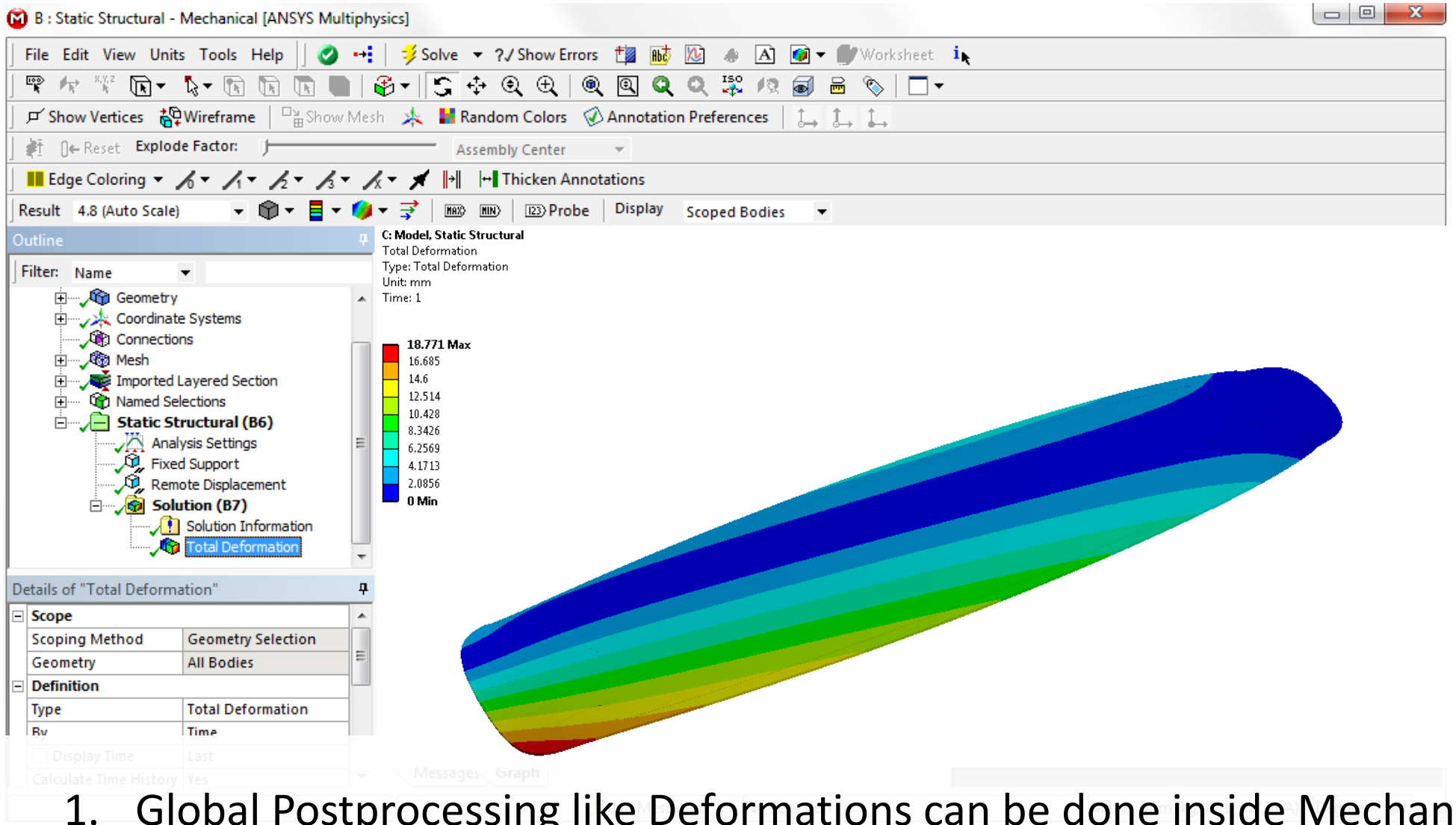
1. Apply Fixed Support Boundary Conditions at one end of the Kiteboard
2. Apply a rotation (rotx=5°, roty=2°) at the other end

2. Workshop Kiteboard



1. Update Setup of the Static Structural Analysis (or use the Update Project selection)
2. Solve the Model

2. Workshop Kiteboard

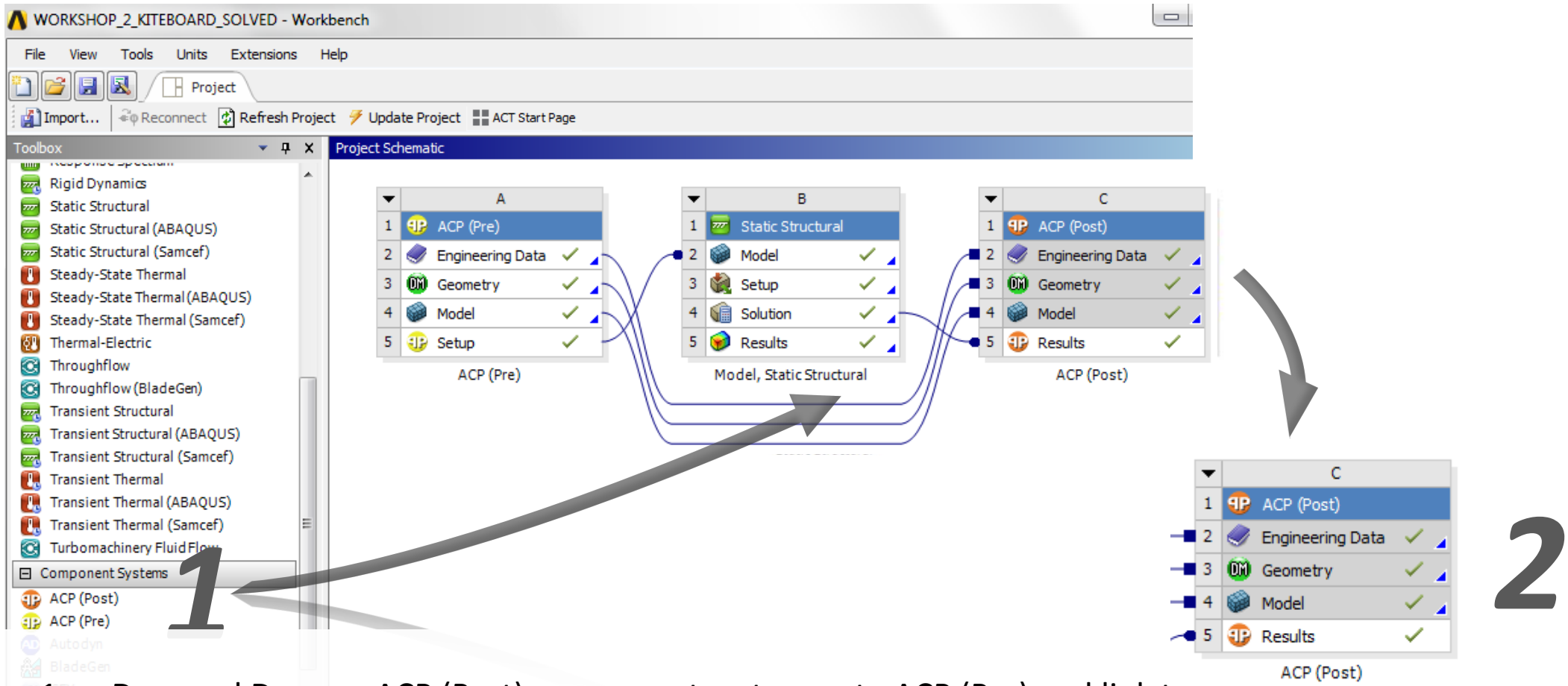


1. Global Postprocessing like Deformations can be done inside Mechanical

2. Workshop Kiteboard

- As a next step we will evaluate the composite specific results in ANSYS Composite PrepPost.
- We will see how to apply different failure criteria and evaluate the design based on failure criteria and stresses within the layer.

2. Workshop Kiteboard



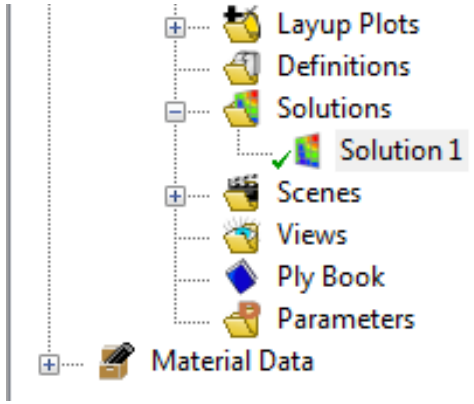
1. Drag and Drop an ACP (Post) component system onto ACP (Pre) and link the Solution of Static Structural to the Results of ACP (Post)
2. Open the results in ANSYS Composite PrepPost (Right mouse button on the ACP (Post) Results → Edit

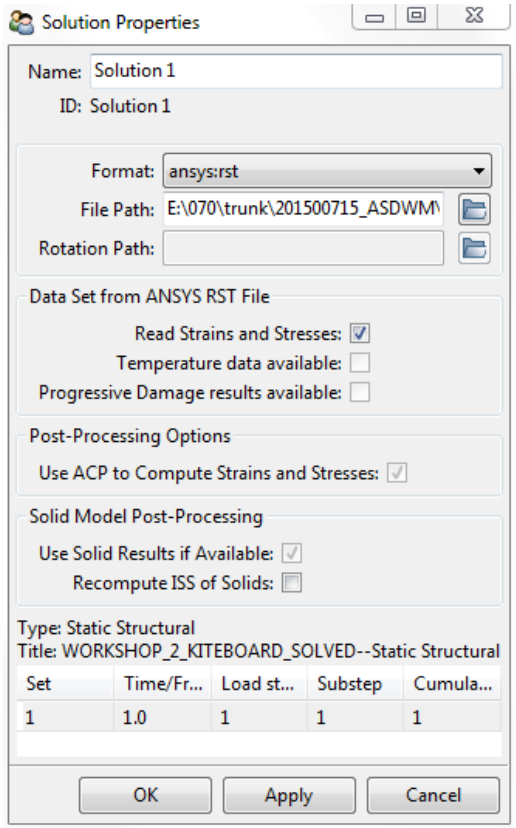
2. Workshop Kiteboard

- **Solutions**

  Before evaluating results the user needs to define the result file to postprocess.

  See **Solution.1** Object under **Solutions** in tree, for the automatically inserted object.





2. Workshop Kiteboard

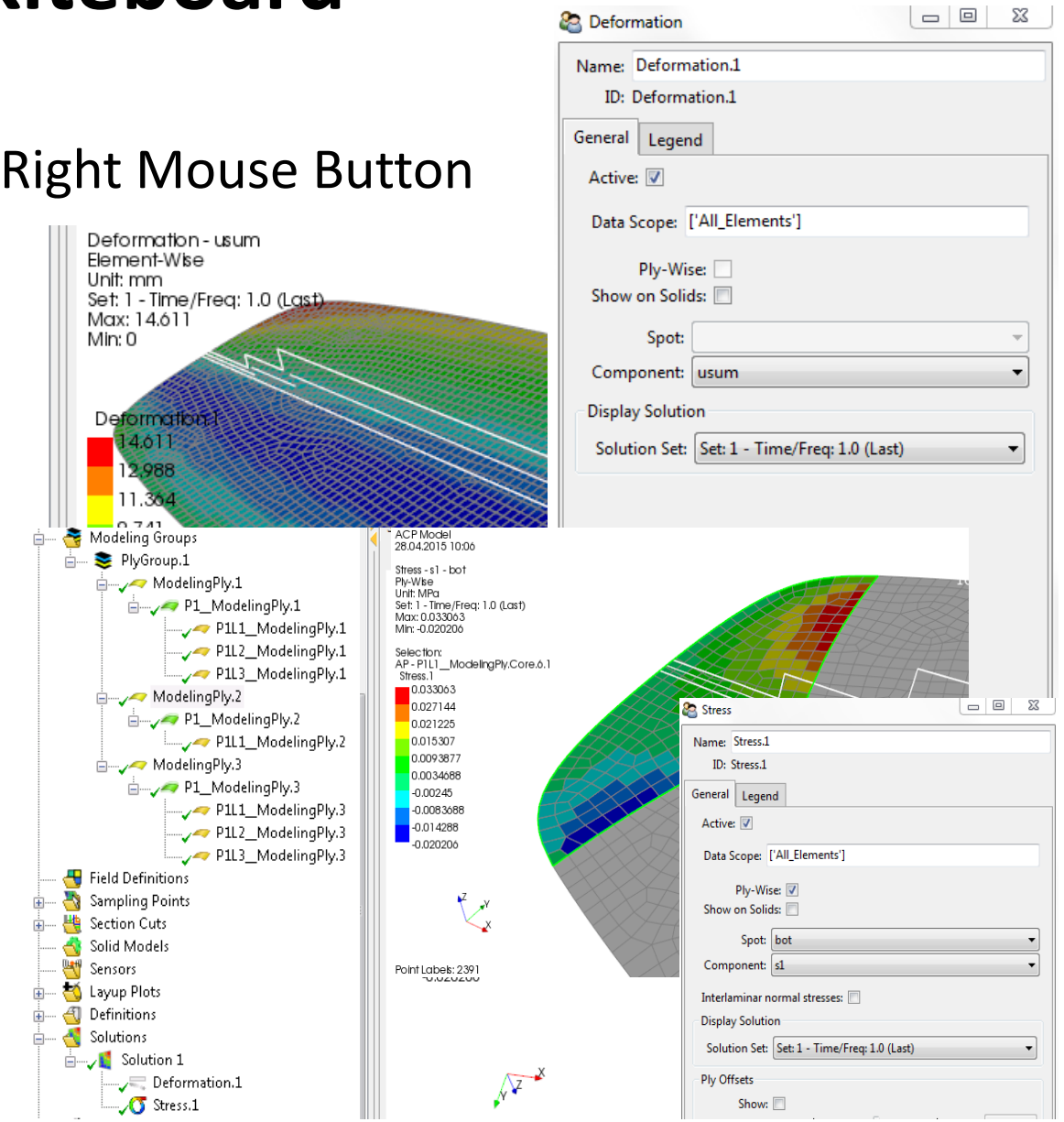
Solutions

After evaluation of Solution Object, use Right Mouse Button to insert result object:

- Create Deformation
- Create Stress

Others available

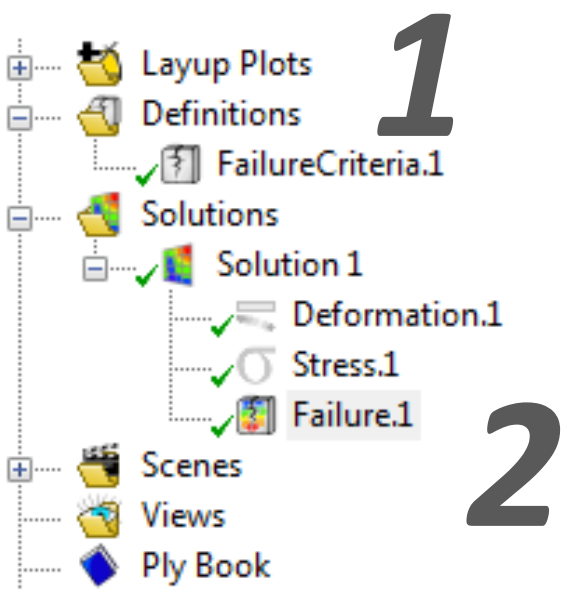
- Create Strain
- Create Failure
- Create Temperature

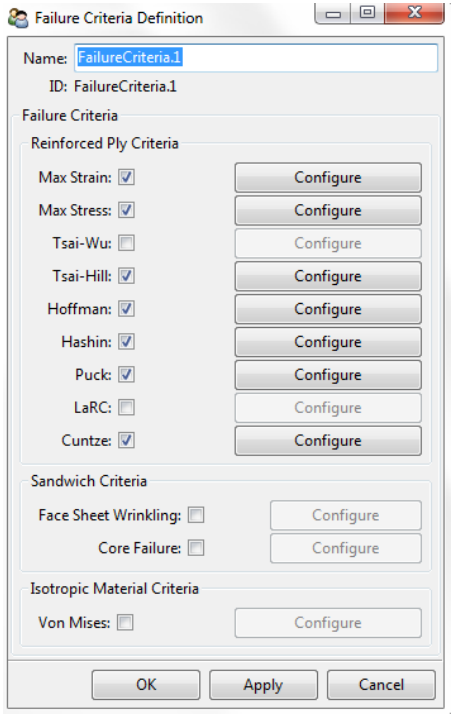


2. Workshop Kiteboard

Definitions

1. Before evaluating Failure results the user needs to define the type of Failure Criteria's which should be evaluated.
2. Failure Criteria need to be added under Definitions to be available for postprocessing under Solution.

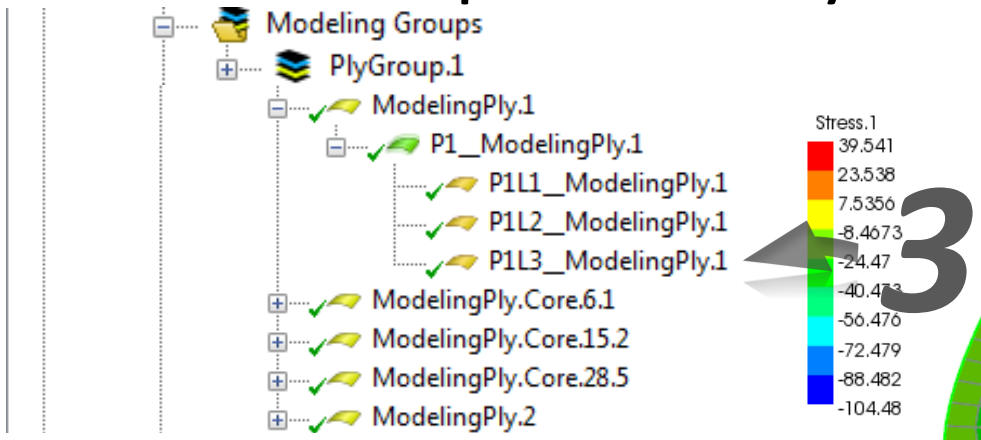


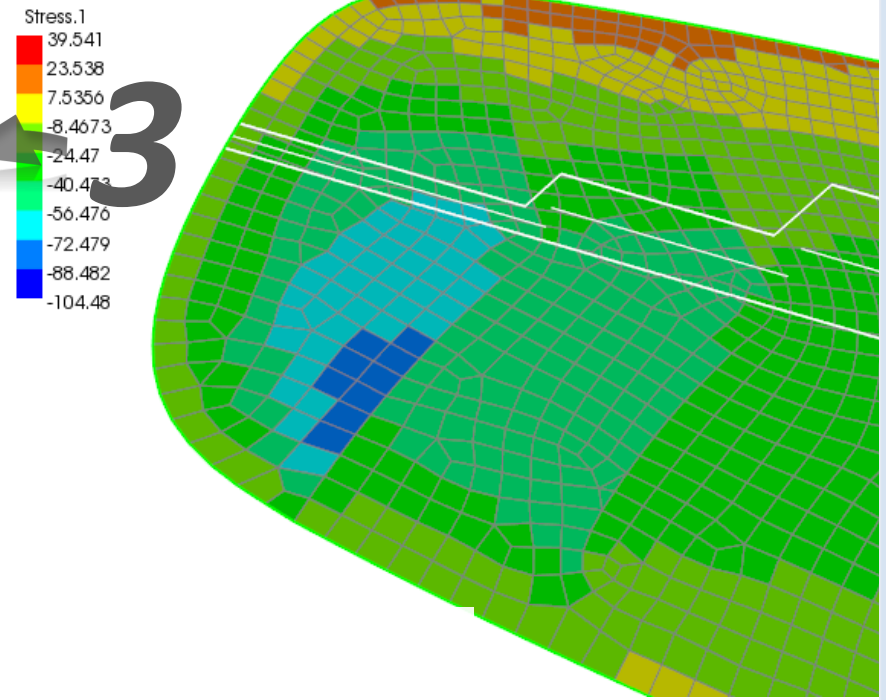


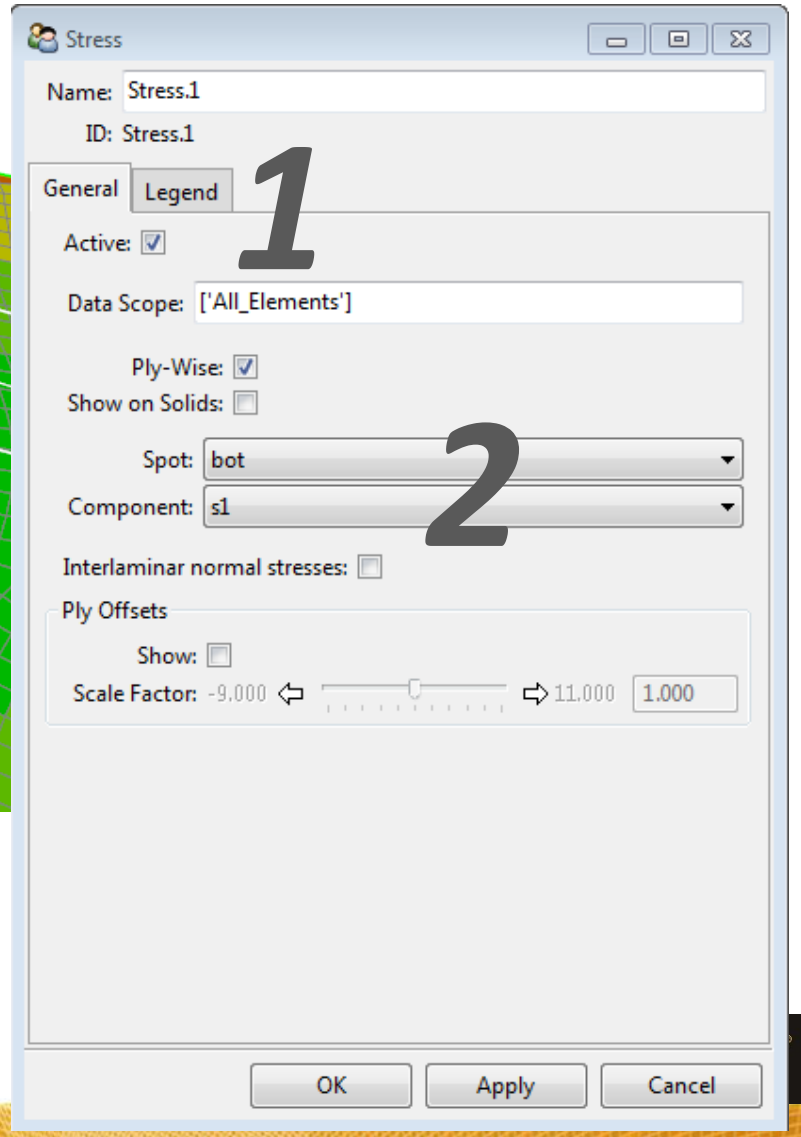
1. Select the Failure Criteria

2. Workshop Kiteboard

Edit stress plot's to layer by layer:



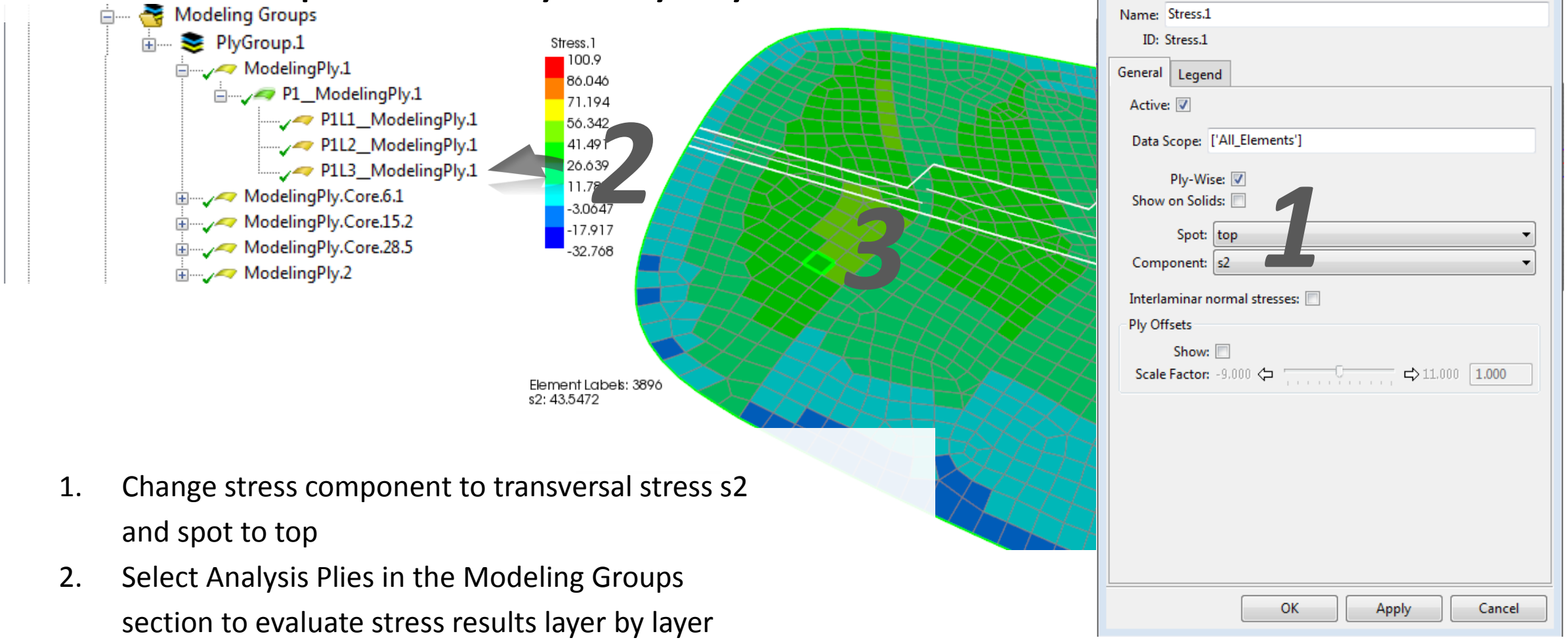




1. Select *Ply Wise* Postprocessing
2. Select type of stress and offset setting
3. Select Analysis Plies in the Modeling Groups section to evaluate stress results layer by layer

2. Workshop Kiteboard

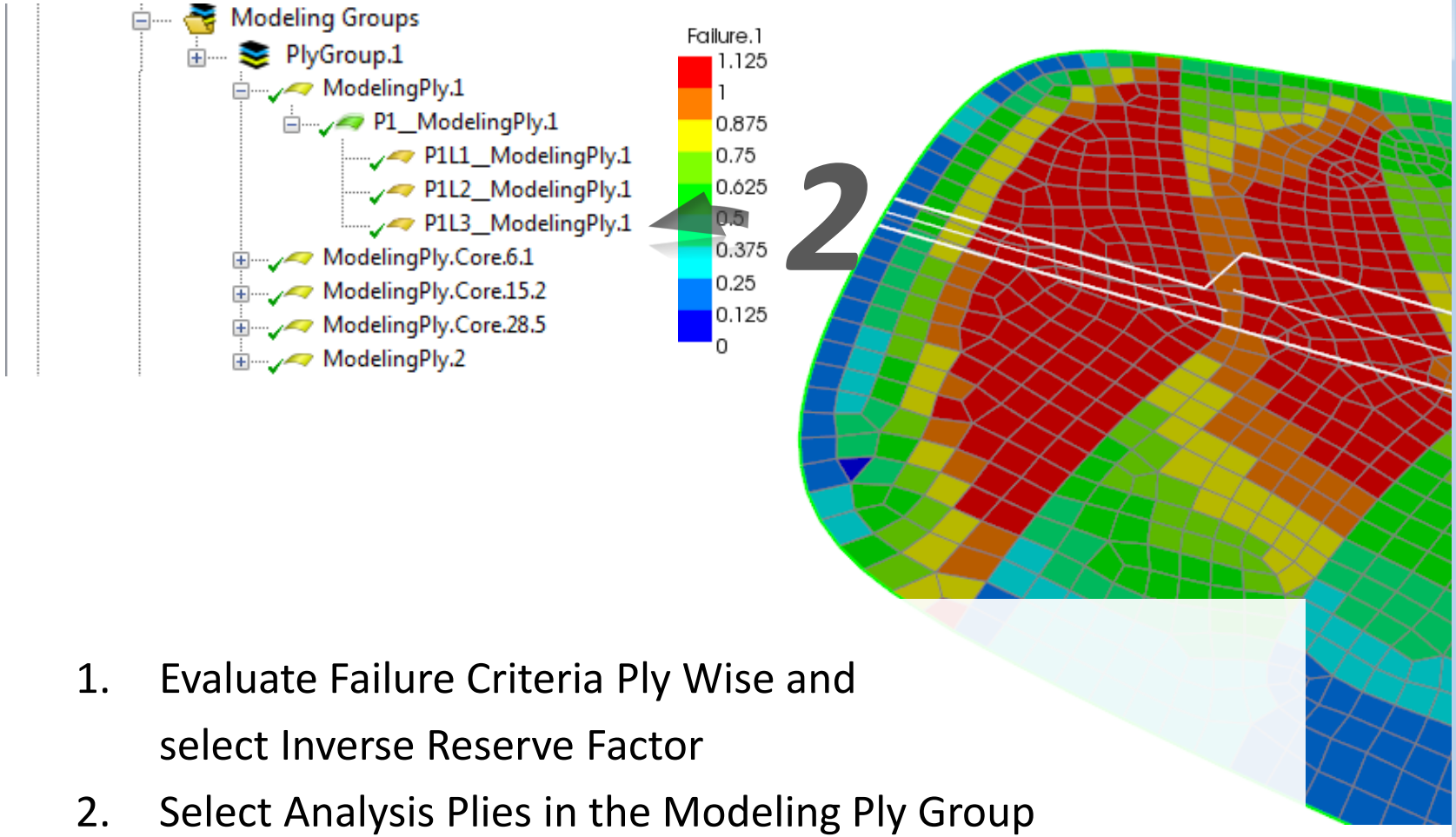
Edit stress plot's to layer by layer:

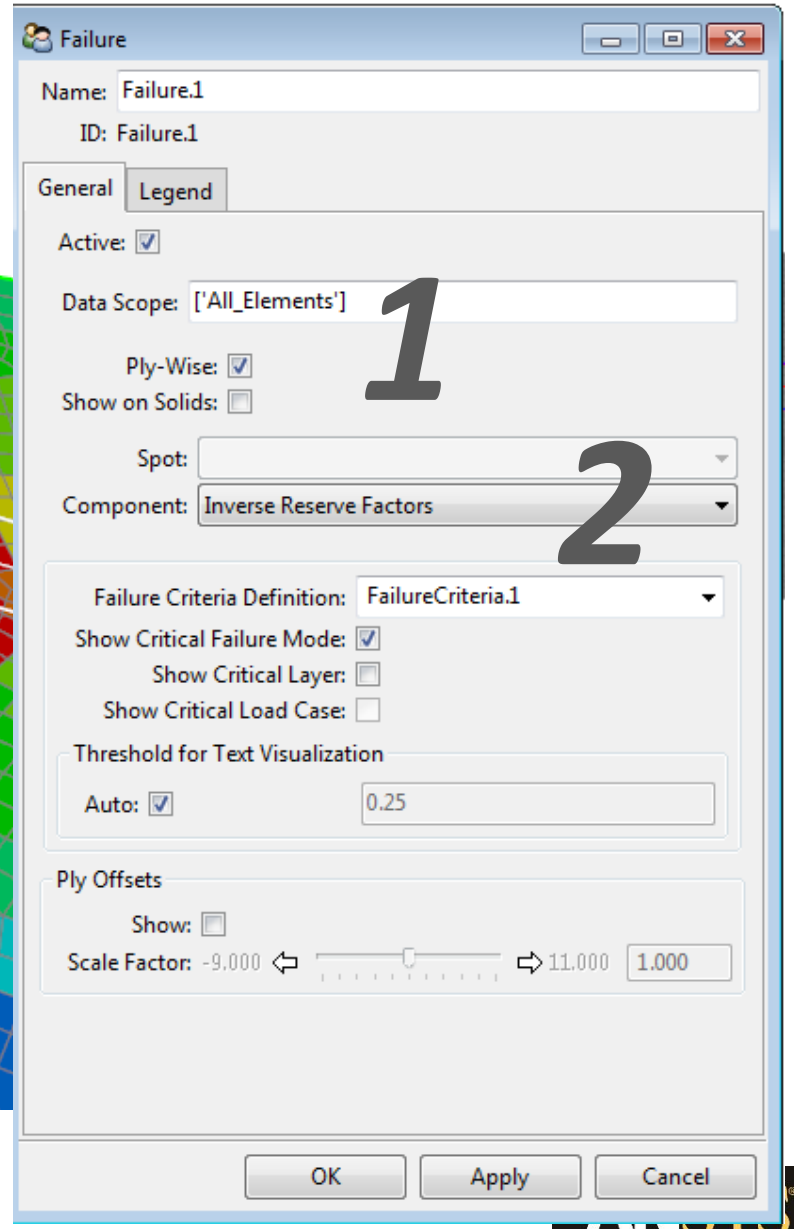


1. Change stress component to transversal stress s2 and spot to top
2. Select Analysis Plies in the Modeling Groups section to evaluate stress results layer by layer
3. Get exact value for a specific element by selecting the element

2. Workshop Kiteboard

Evaluate Failure Criteria

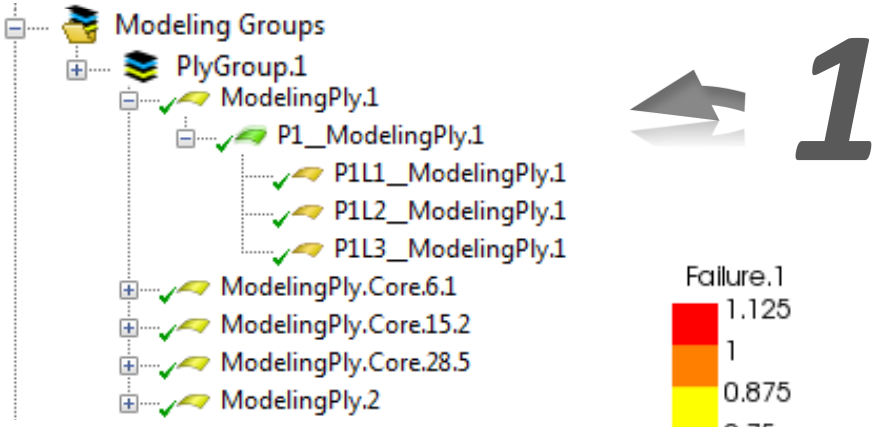




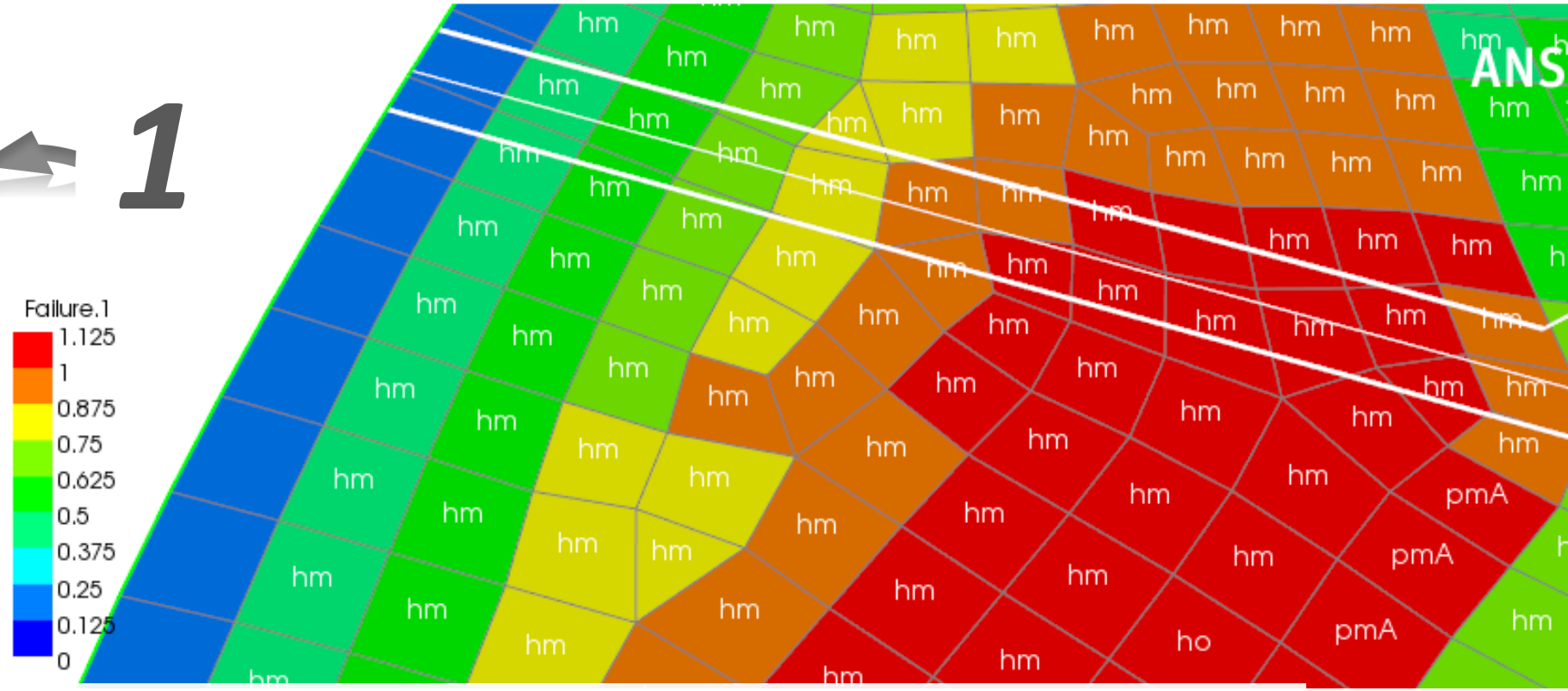
1. Evaluate Failure Criteria Ply Wise and select Inverse Reserve Factor
2. Select Analysis Plies in the Modeling Ply Group section to evaluate failure criteria layer by layer

2. Workshop Kiteboard

Evaluate Failure Criteria

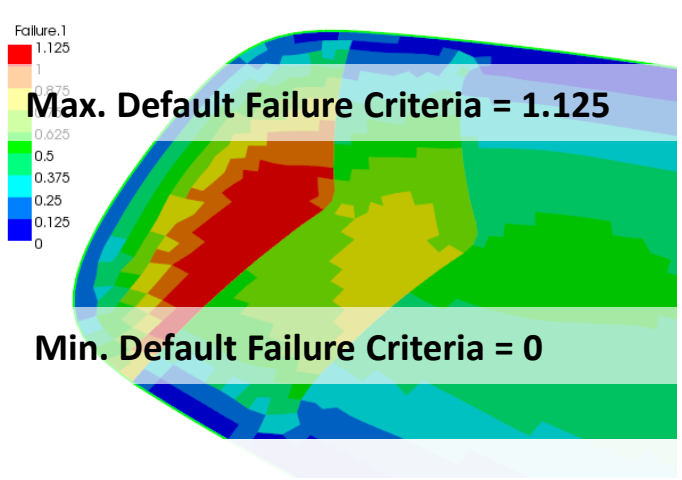


1

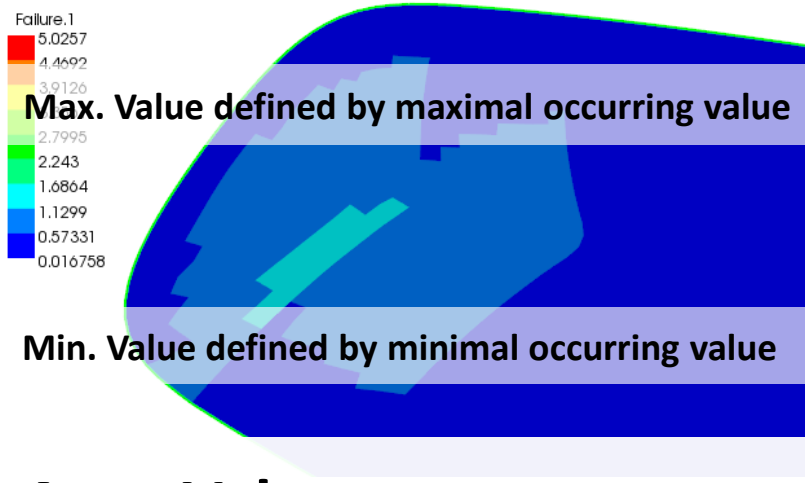


1. Zoom into critical area of the top ply
2. Switch Annotation on to highlight critical failure criteria in contour plot (Toggle Text Plot) (Avoid switching annotation on when complete model is displayed)

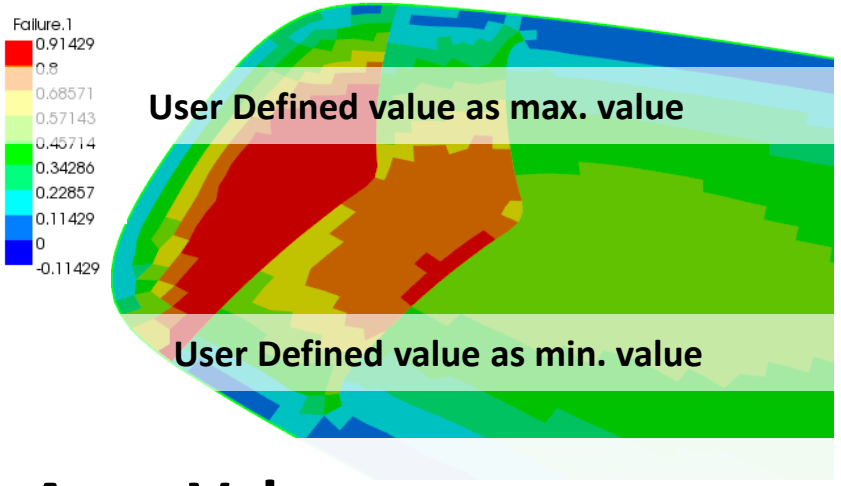
2. Workshop Kiteboard



Use Defaults



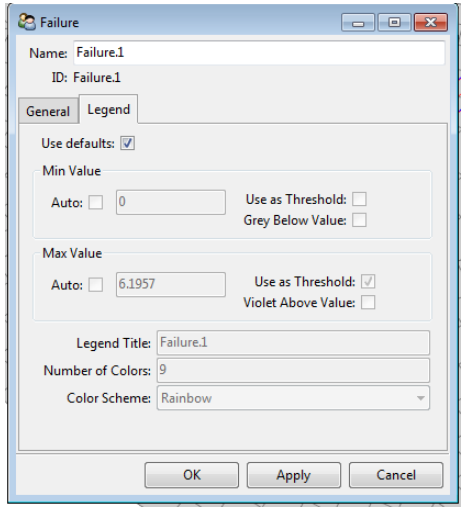
Auto Values



Auto Values

- **Contour Settings**

  Contour Settings are modified in the Legend tab of the Plots under Solution. Default maximum and minimum values are not the same for all results (stresses, strains, failure criteria …).

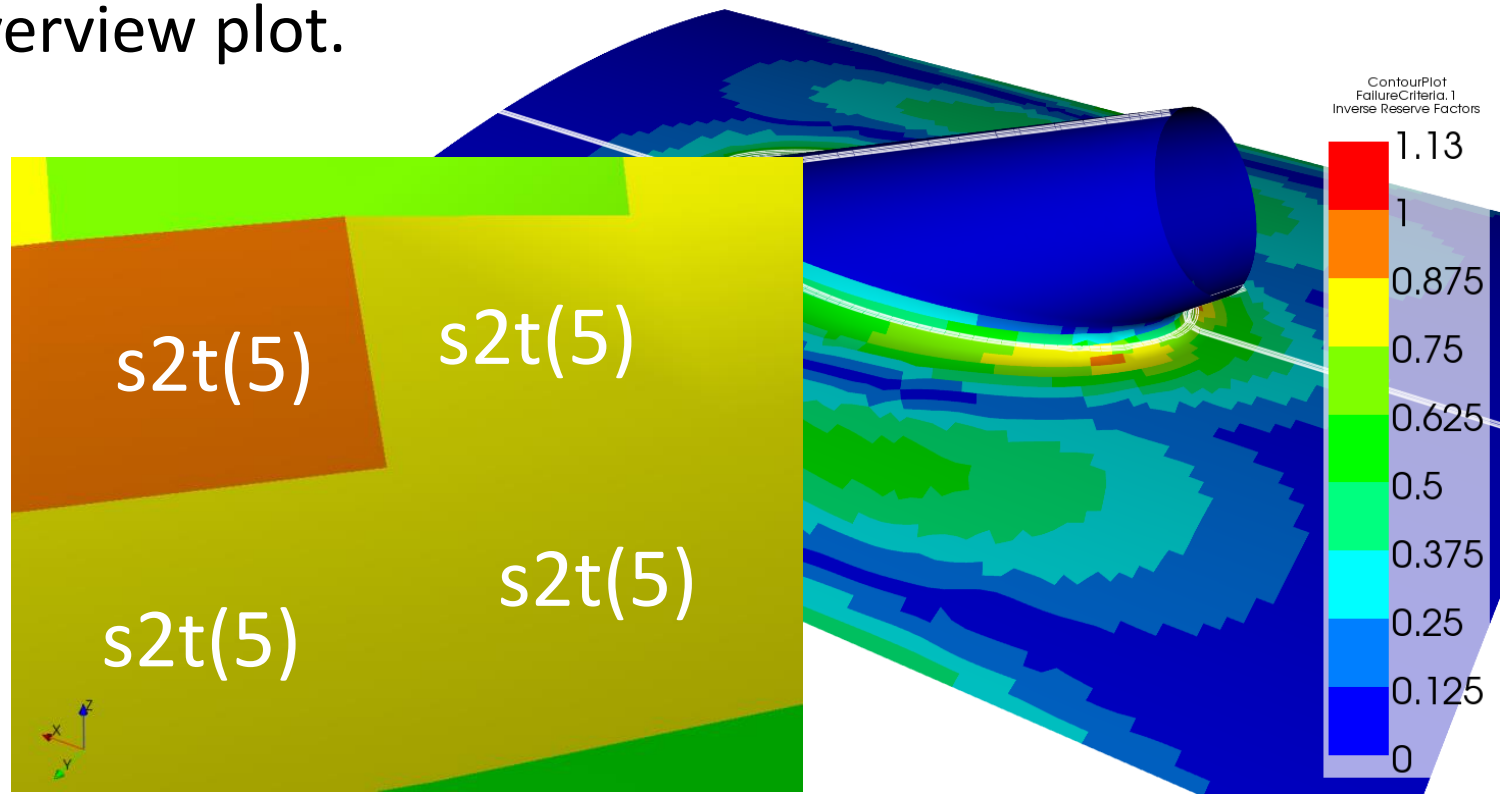


2. Workshop Kiteboard

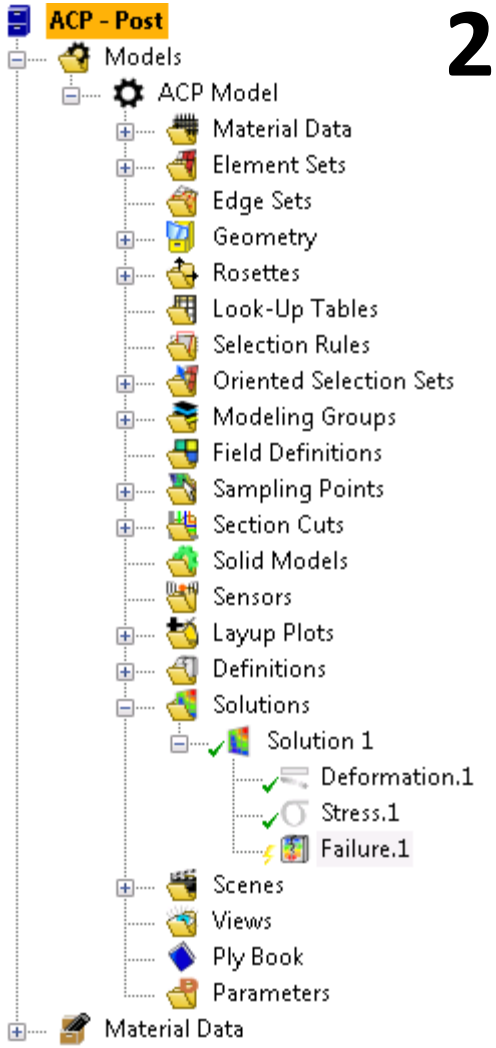
- **All in One Overview Plot**

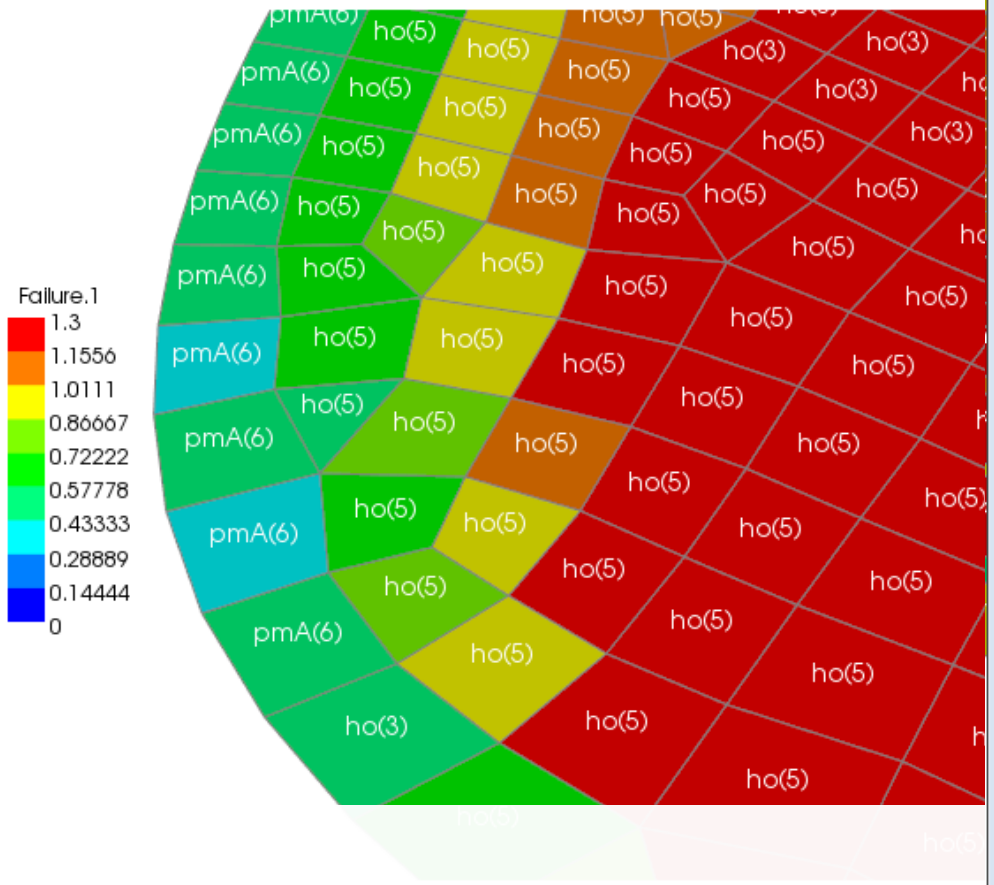
  In addition to the layerwise postprocessing failure criteria can be evaluated as all in one overview plot.

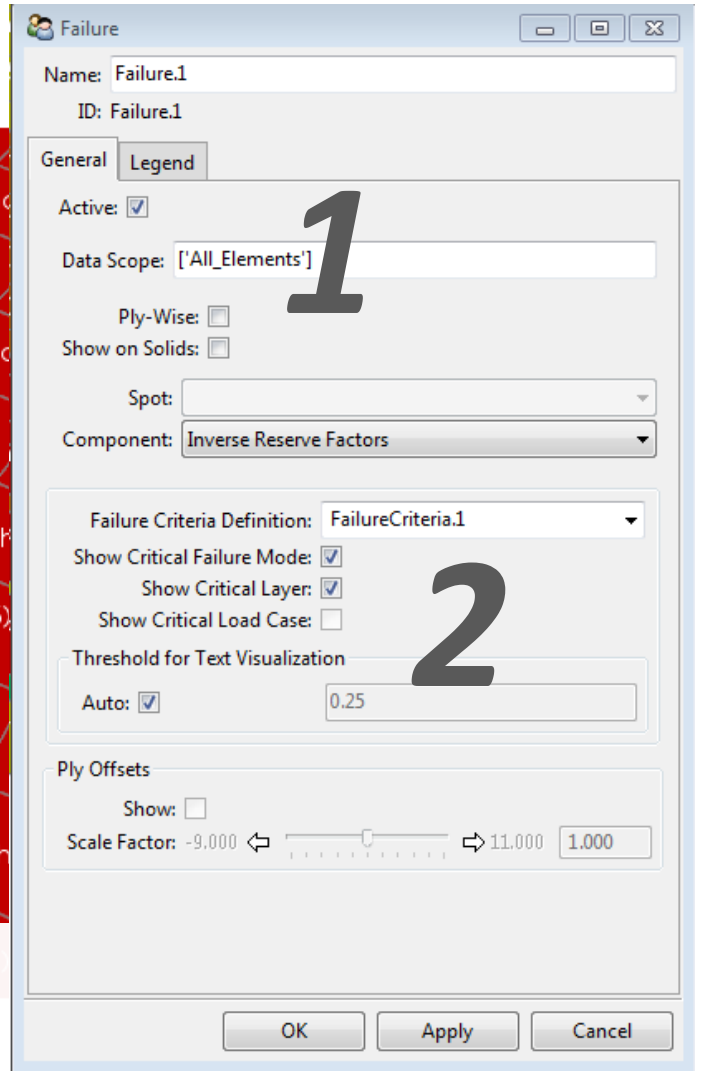
- **In One View**
  - Failure Criteria
  - Failure Mode
  - Critical Layer
  - Critical Loadstep



2. Workshop Kiteboard







1. Unselect *Ply Wise* evaluation
2. Select Show Failure Mode Plot and Critical Failure Mode as well as Critical Layer
3. Highlight Annotation (*ho(5)* indicates failure of layer number 4 according to Hoffman criterion). See all failure criteria terms on next slides

2. Workshop Kiteboard

Failure Criteria Terms:

- e = strain s = stress
- 1 = material 1 direction
- 2 = material 2 direction
- 3 = out-of-plane normal direction
- 12 = in-plane shear
- 13 and 23 = out-of-plane shear terms
- I = principal I direction
- II = principal II direction
- III = principal III direction
- t = tension, c = compression

2. Workshop Kiteboard

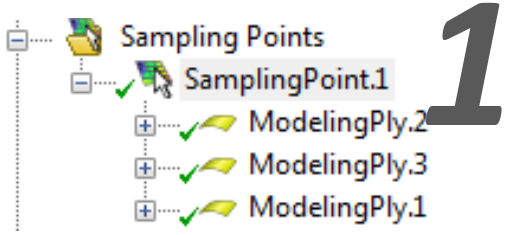
Failure Modes:

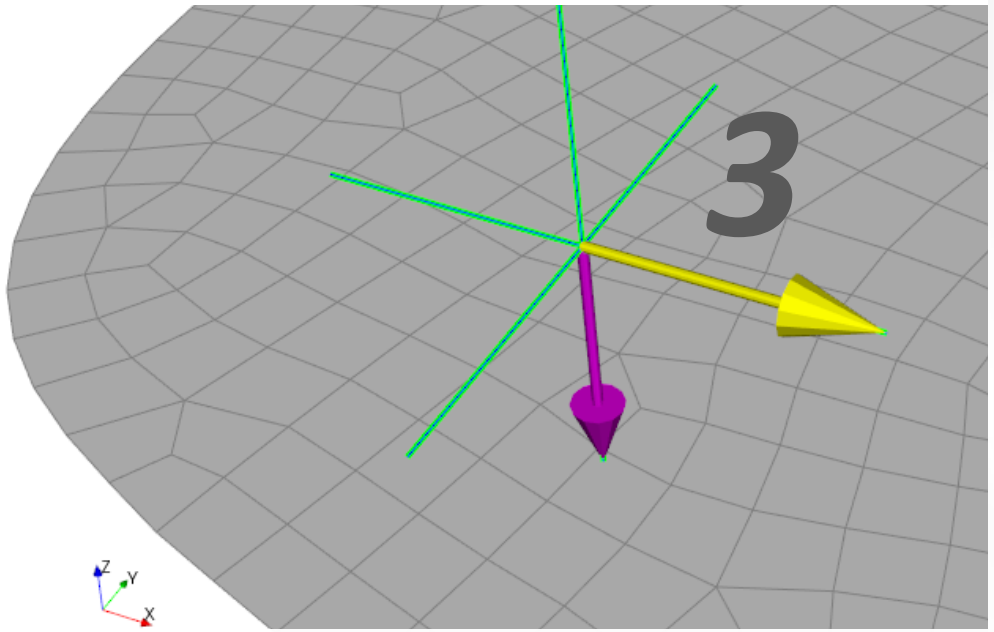
Maximum Strain Failure modes	e1t, e1c, e2t, e2c, e12
Maximum Stress	s1t, s1c, s2t, s2c, s3t, s3c, s12, s23, s13
Tsai-Wu 2D and 3D	tw
Tsai-Hill 2D and 3D	th
Hashin	hf (fiber failure) hm (matrix failure) hd (delamination failure)
Puck (simplified, 2D and 3D)	pf (fiber failure) pmA (matrix tension failure) pmB (matrix compression failure) pmC (matrix shear failure) pd (delamination)

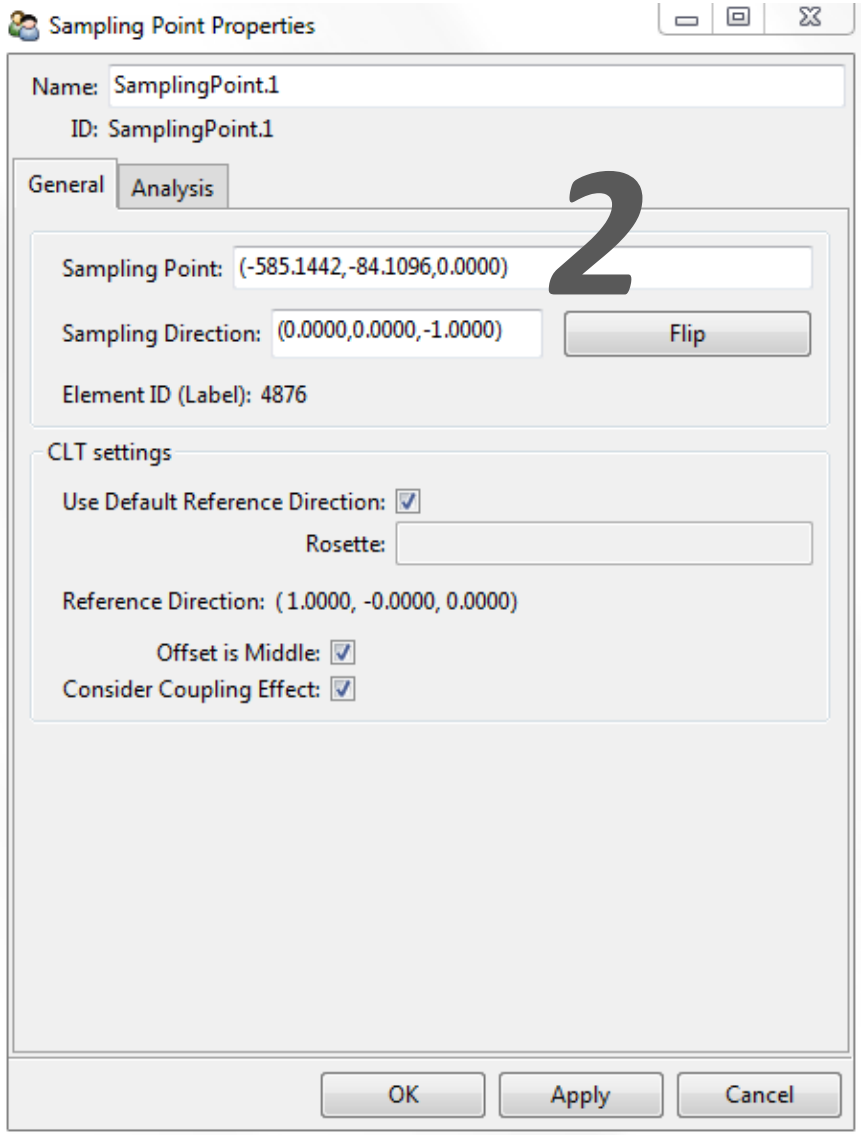
LaRC (2D)	lf (fiber failure) lmt (matrix failure tension) lmc (matrix failure compression)
Cuntze 2D and 3D	cft (fiber tension failure) cfc (fiber compression failure) cmA (matrix tension failure) cmB (matrix compression failure) cmC (matrix wedge shape failure)
Sandwich Failure Wrinkling	wb (wrinkling bottom face) wt (wrinkling top face)
Sandwich Failure Core	cf (core failure)
Hoffman	ho

2. Workshop Kiteboard

Sampling Points

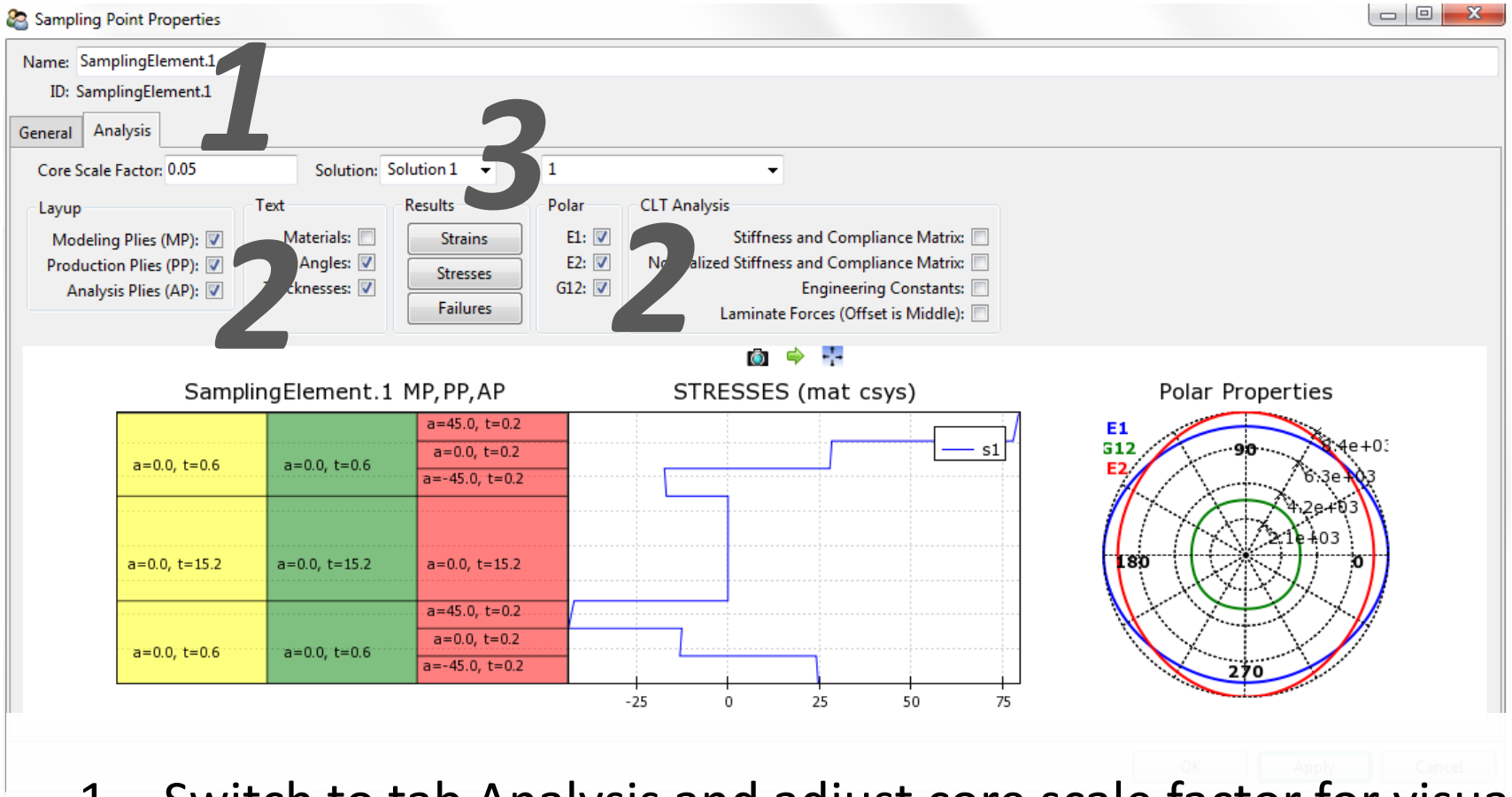


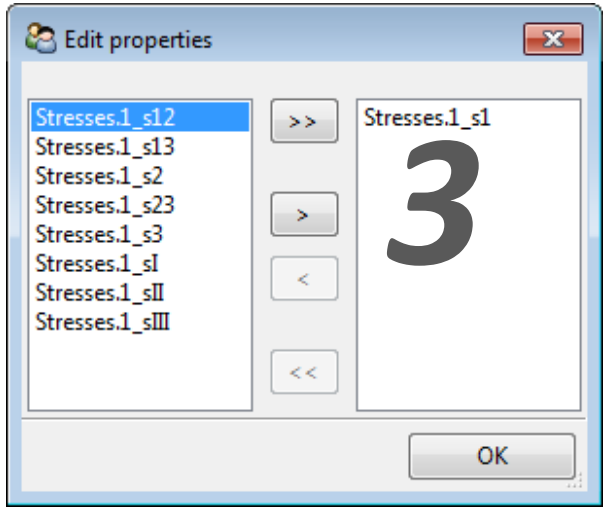




1. Create a sampling point by Right Mouse Button on Sampling Points → Create Sampling Point

2. Workshop Kiteboard





1. Switch to tab Analysis and adjust core scale factor for visualization purposes
2. Select information to show through the element
3. Add Stress plot through the layers of the selected element