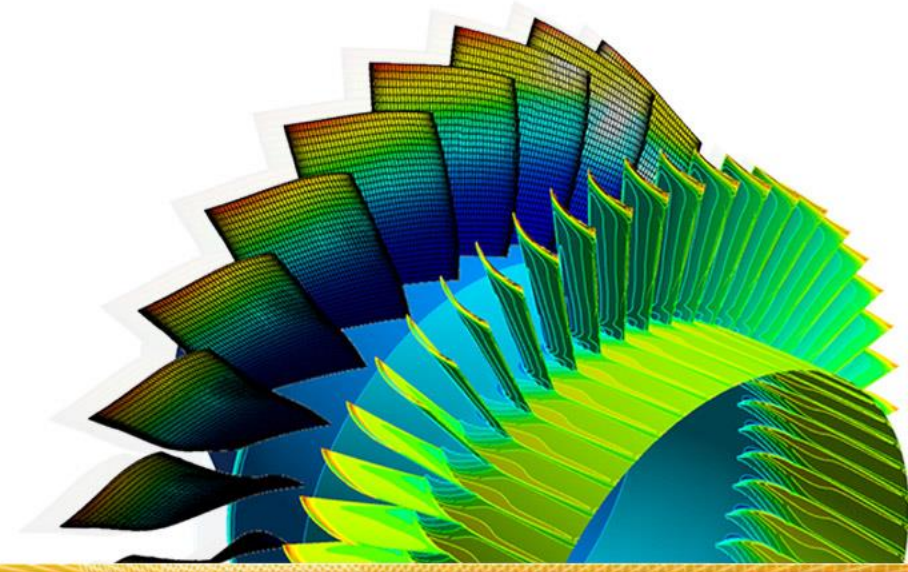
ANSYS Composite PrepPost 17.0

Tutorial Exercise 1

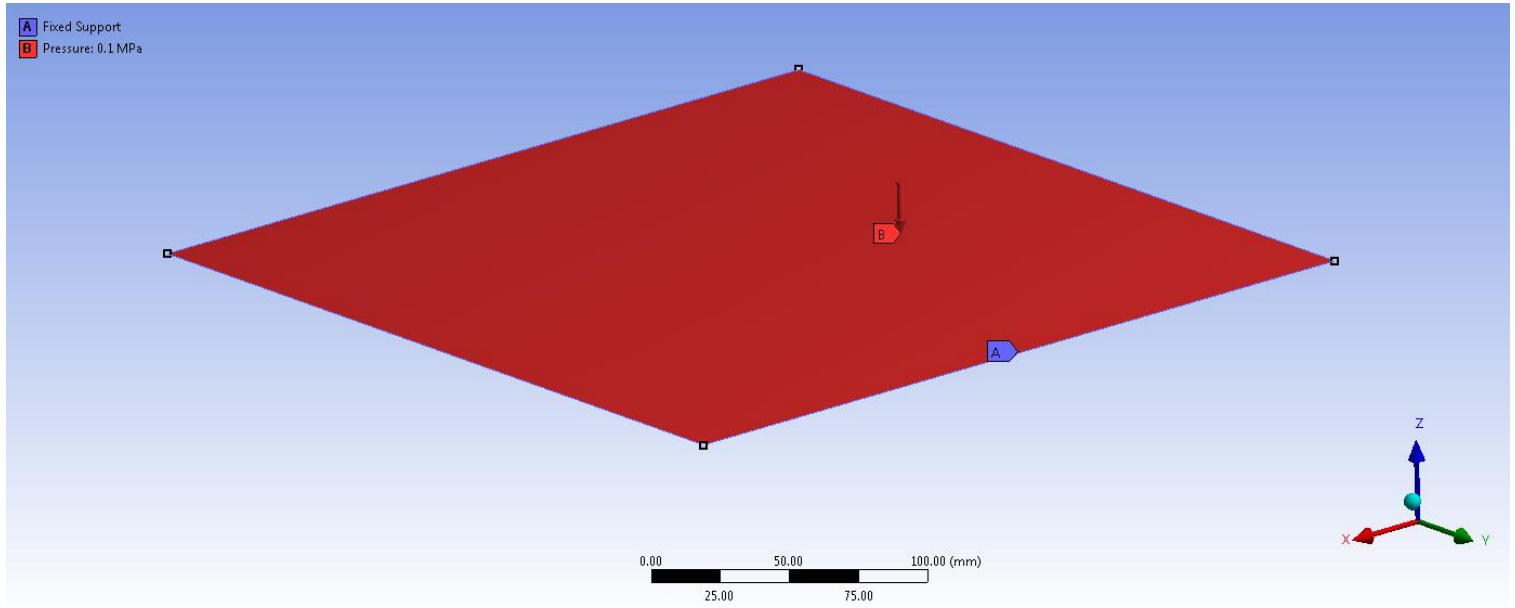


ACP Tutorial

Goals:

- **Basic composite workflow from a geometry to post-processing.**
- **Build a simple sandwich panel.**

Load Case: Clamped panel under uniform pressure.

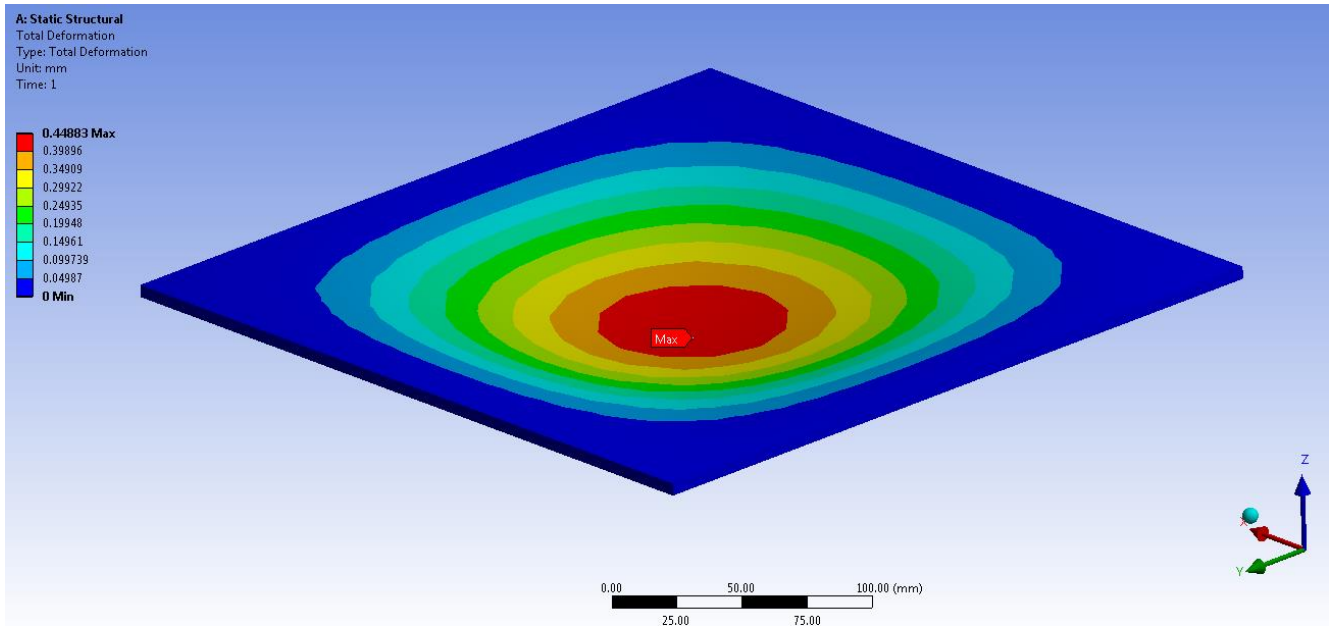


ACP Tutorial

- ~~Open a new Workbench project and restore the archive “tutorial_1.wbpz”.~~

Create a model using ACP (Pre), e.g., a plate.

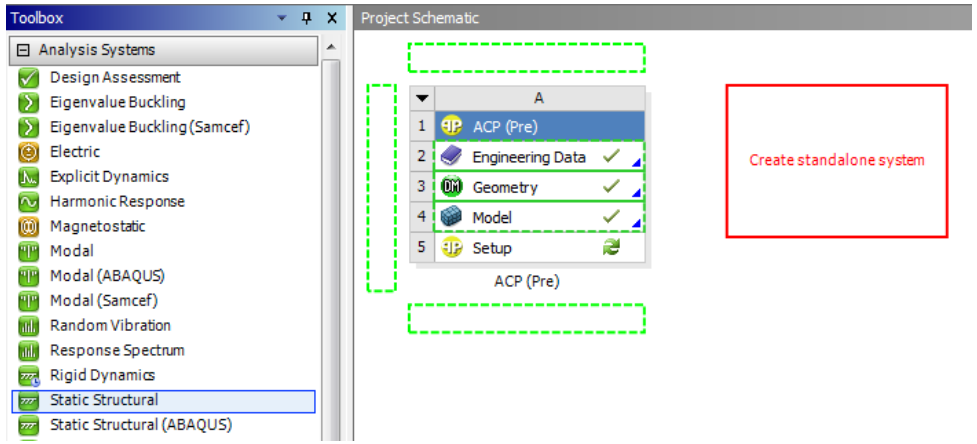
- ~~Review the boundary conditions, and verify that the model is well-defined with the default material. Review the results with an isotropic material.~~



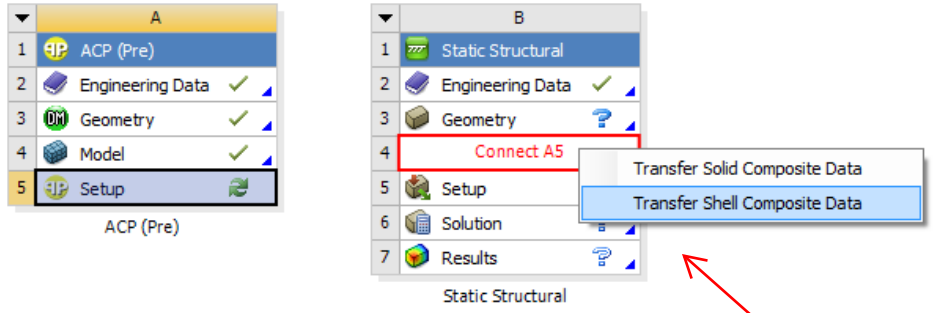
ACP Tutorial

- **Add a Static Structural component to the existing analysis.**

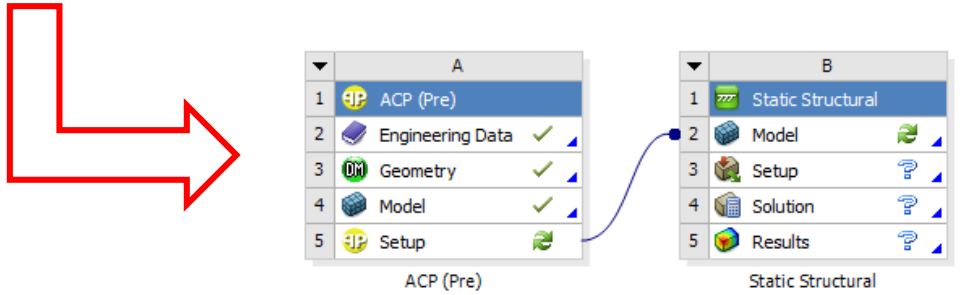
1.



2.



3.



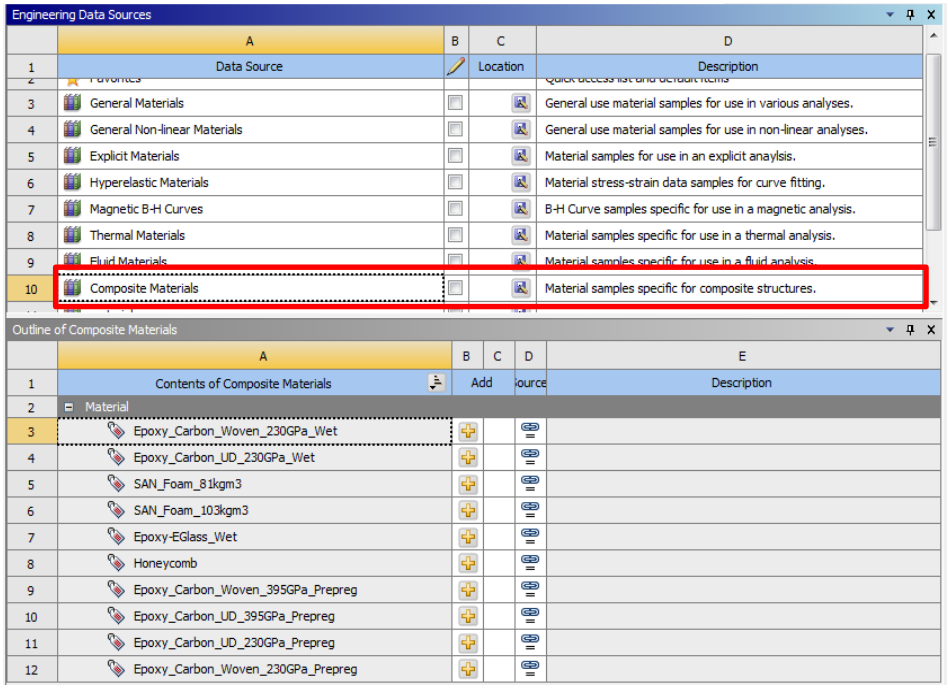
You need to drag "Setup" from ACP (Pre) to "Model" of Static Structural.

ACP Tutorial

First composite materials have to be defined in ANSYS Workbench Engineering Data:

There are two possibilities:

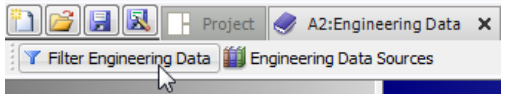
1. **Import preconfigured materials from the *Composite Materials* catalog (see figure below)**
2. **Create new materials**
   In this example, you will create new materials as shown in the following slides.



ACP Tutorial

Define a unidirectional material in ANSYS Workbench Engineering Data with the following properties:

Hint
Uncheck the filter button to display all properties in the toolbox



Outline of Schematic A2, B2, C2: Engineering Data

	A	B	C	D
1	Contents of Engineering Data		Source	Description
2	Material			
3	UD_T700			
*	Click here to add a new material			

Properties of Outline Row 3: UD_T700

	A	B	C	D	E
1	Property	Value	Unit		
2	Orthotropic Elasticity				
3	Young's Modulus X direction	1.15E+05	MPa		
4	Young's Modulus Y direction	6430	MPa		
5	Young's Modulus Z direction	6430	MPa		
6	Poisson's Ratio XY	0.28			
7	Poisson's Ratio YZ	0.34			
8	Poisson's Ratio XZ	0.28			
9	Shear Modulus XY	6000	MPa		
10	Shear Modulus YZ	6000	MPa		
11	Shear Modulus XZ	6000	MPa		
12	Orthotropic Stress Limits				
13	Tensile X direction	1500	MPa		
14	Tensile Y direction	30	MPa		
15	Tensile Z direction	30	MPa		
16	Compressive X direction	-700	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		
22	Tsai-Wu Constants				
23	Coupling Coefficient XY	-1			
24	Coupling Coefficient YZ	-1			
25	Coupling Coefficient XZ	-1			
26	Ply Type				
27	Type	Regular			

ACP Tutorial

Define also a core material:

Outline of Schematic A2, B2, C2: Engineering Data

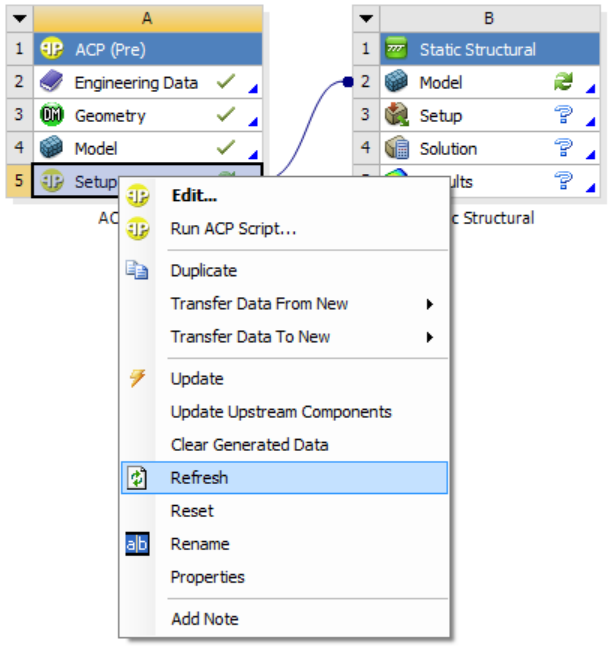
	A	B	C	D
1	Contents of Engineering Data		ource	Description
2	Material			
3	Corecell_A550			
4	UD_T700			
*	Click here to add a new material			

Properties of Outline Row 3: Corecell_A550

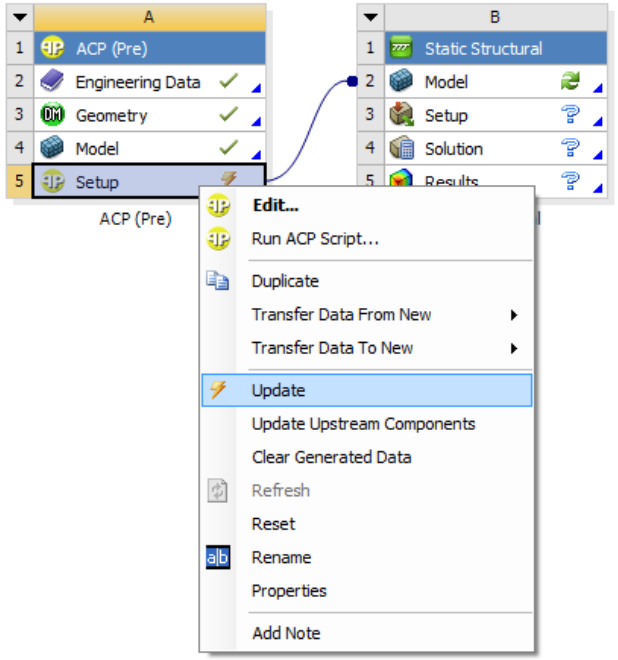
	A	B	C	D	E
1	Property	Value	Unit		
2	Isotropic Elasticity				
3	Derive from	Young's Modulus and Poisson's Ratio			
4	Young's Modulus	85	MPa		
5	Poisson's Ratio	0.3			
6	Bulk Modulus	70.833	MPa		
7	Shear Modulus	32.692	MPa		
8	Orthotropic Stress Limits				
9	Tensile X direction	1.6	MPa		
10	Tensile Y direction	1.6	MPa		
11	Tensile Z direction	1.6	MPa		
12	Compressive X direction	-1.1	MPa		
13	Compressive Y direction	-1.1	MPa		
14	Compressive Z direction	-1.1	MPa		
15	Shear XY	1.1	MPa		
16	Shear YZ	1.1	MPa		
17	Shear XZ	1.1	MPa		
18	Ply Type				
19	Type	Isotropic Homogeneous Core			

ACP Tutorial

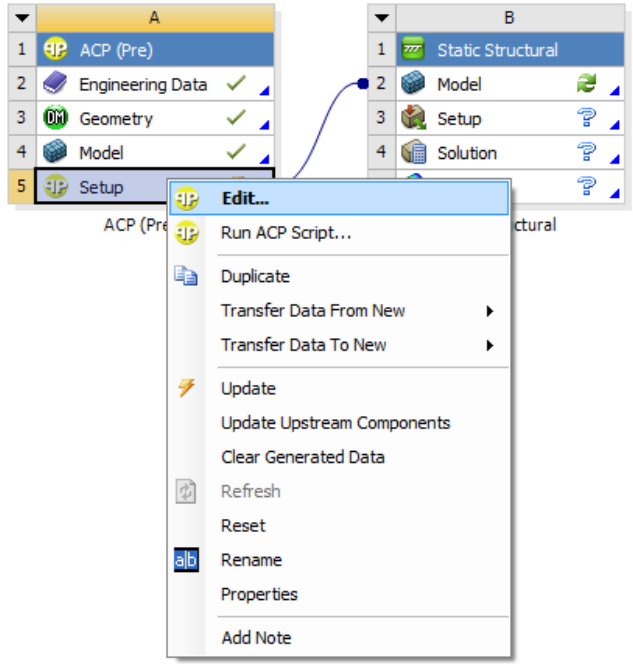
- Update *Model* and then refresh *Setup* in the *ACP (Pre)* component
- Open *Setup* of *ACP (Pre)* with a double-click on *Setup* (or *Edit…* in drop-down menu)



1



2

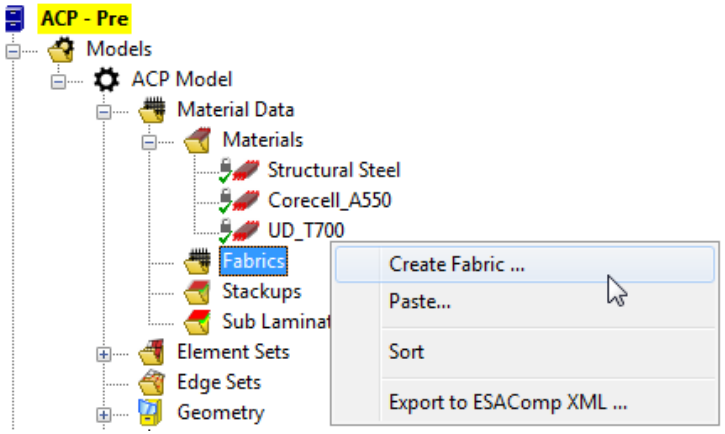


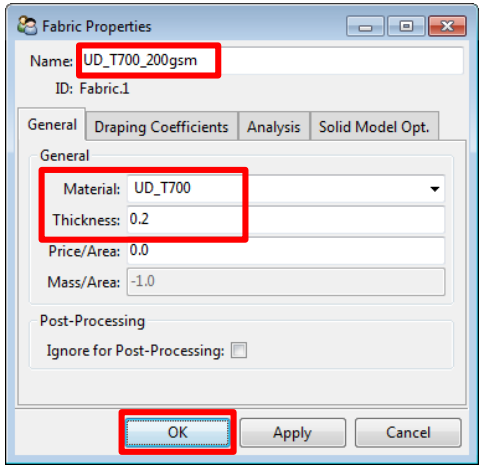
3

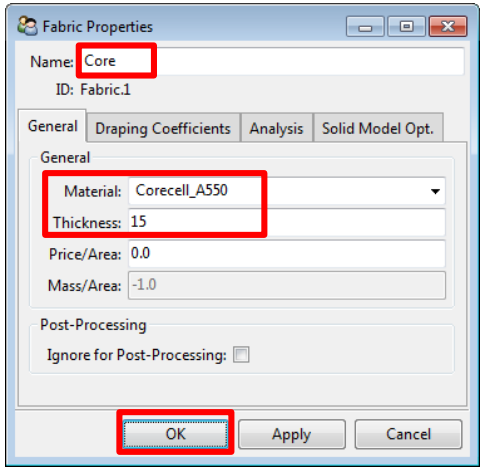
ACP Tutorial

- In ACP further material data (*Fabrics, Stackup and Sub Laminates*) have to be defined.

- Define a new *Fabric* with the defined materials:
  - Carbon UD with 0.2 mm thickness,
  - Foam core with 15 mm thickness.

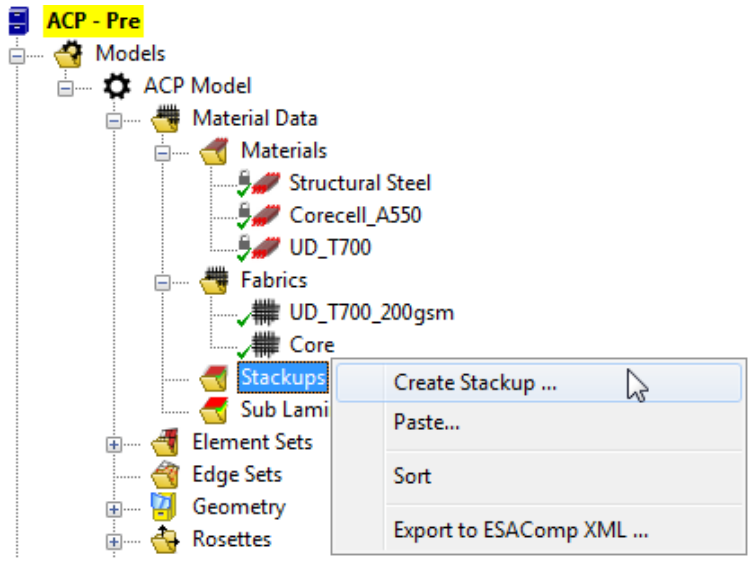


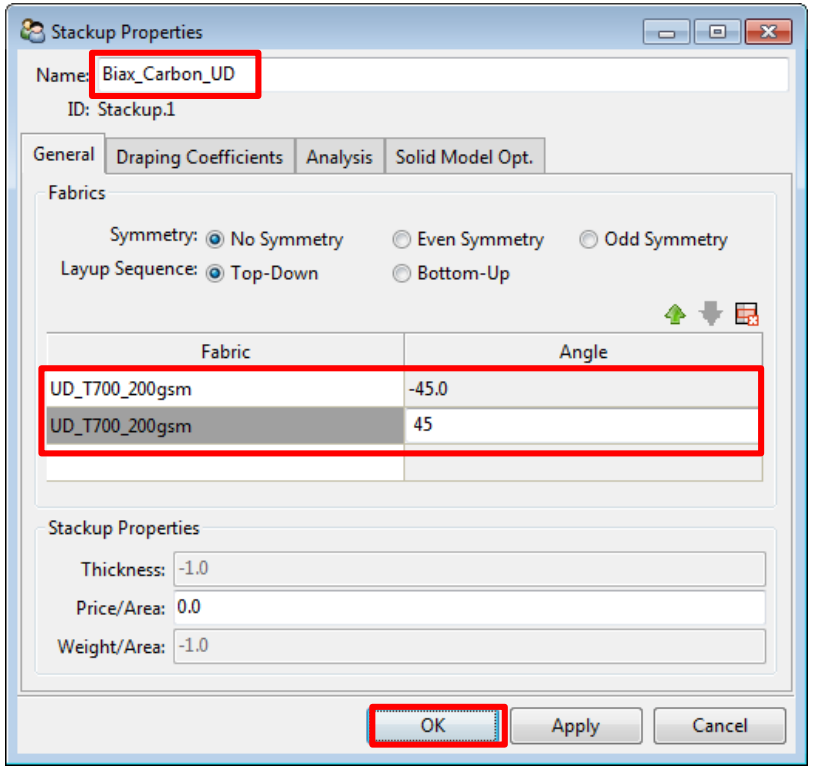




ACP Tutorial

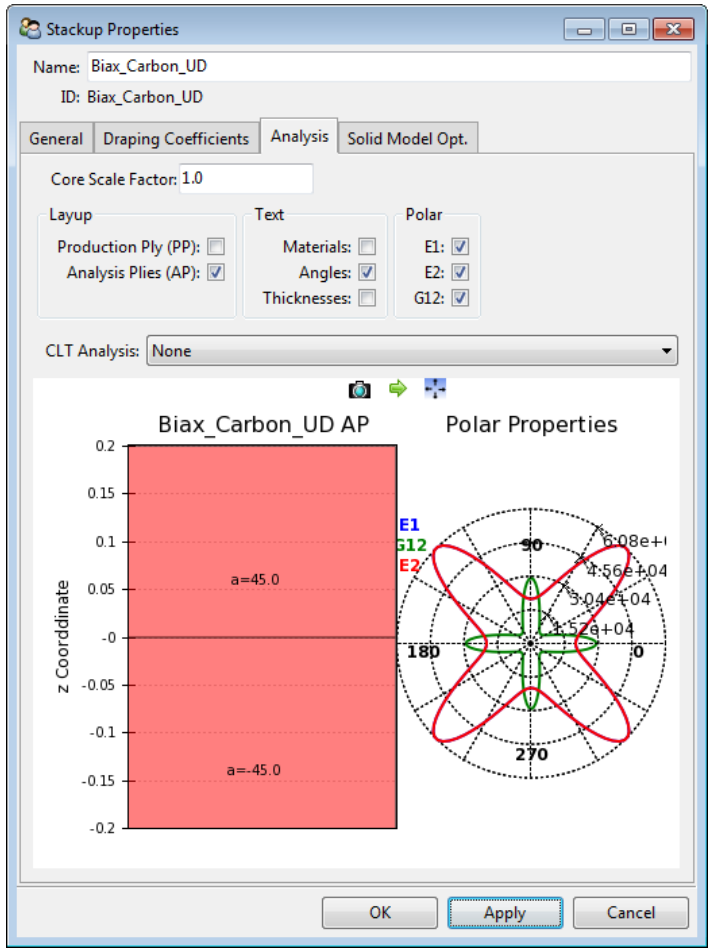
- **Define a new *Stackup* with the UD Carbon. A *Stackup* is a pre-assembled tape also called non-crimp fabric (NCF).**





ACP Tutorial

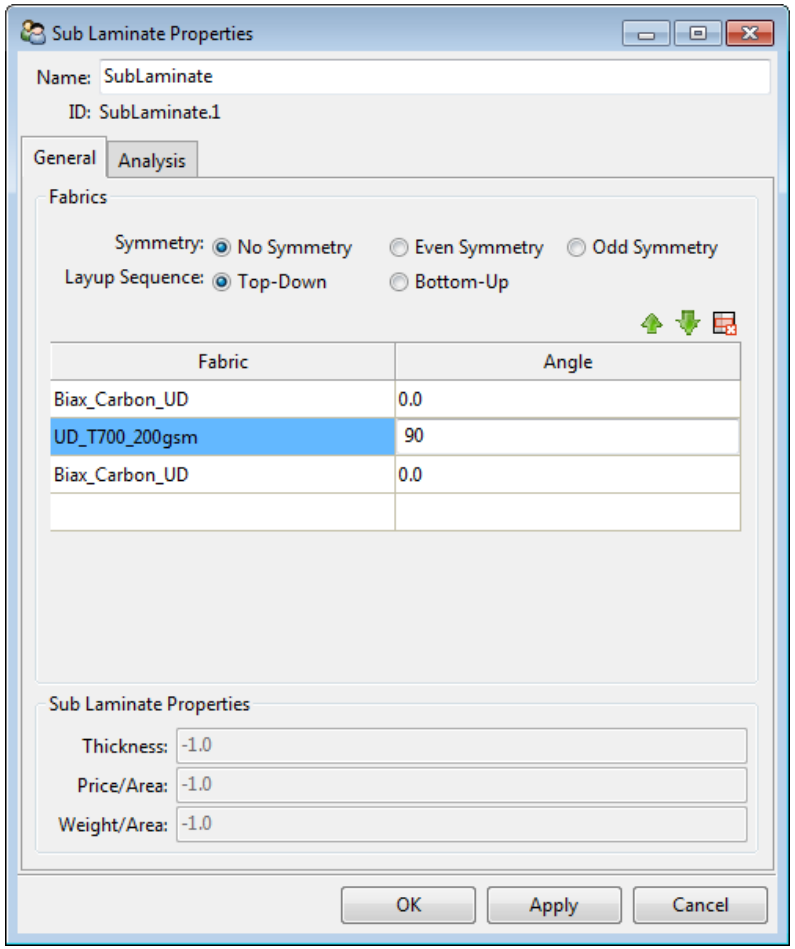
- **Review the Biax properties through the Plot tab.**

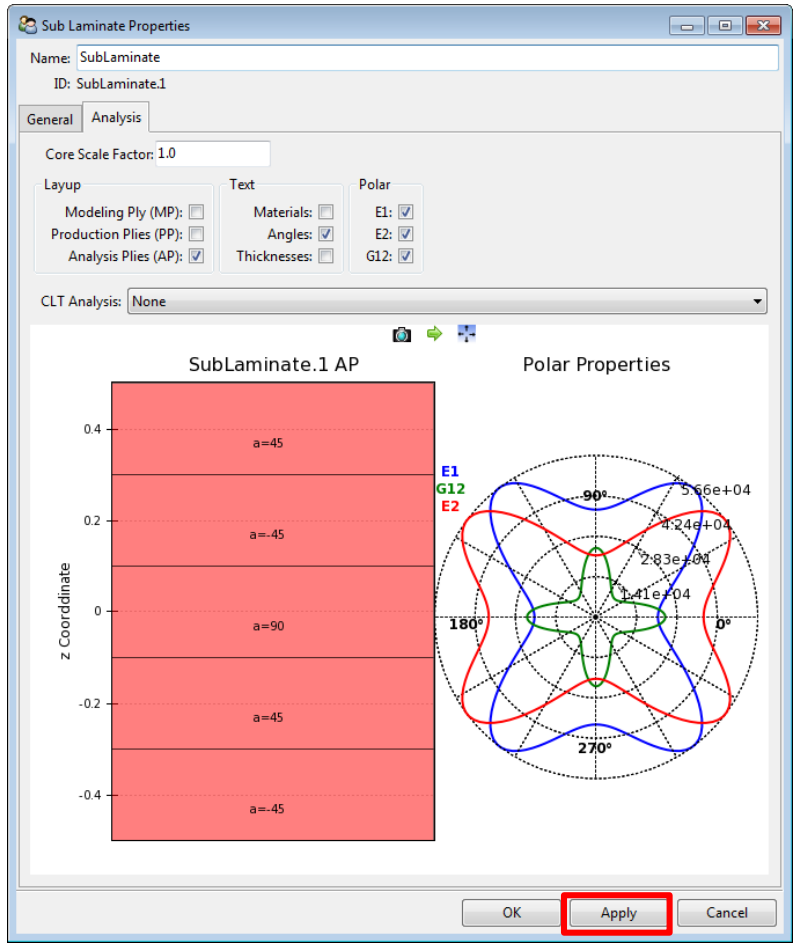


Click on apply to update the model & OK to close the window.

ACP Tutorial

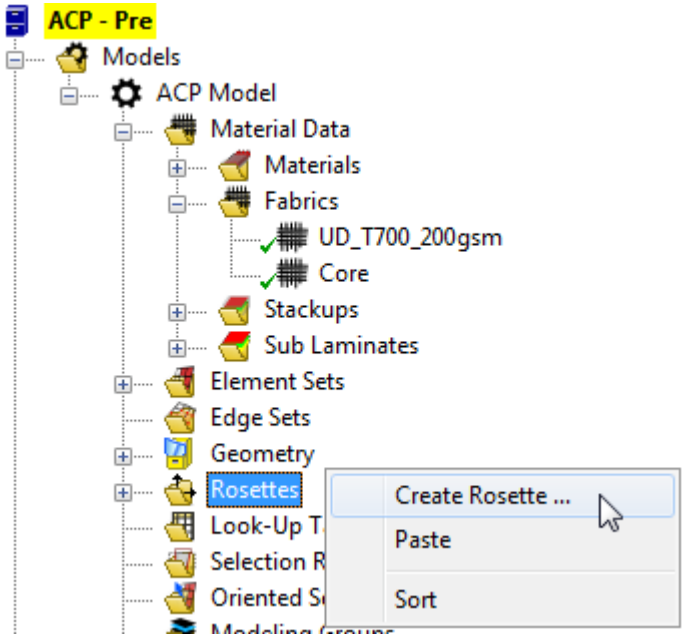
- **Define a *Sub Laminate* as shown below and plot the mechanical properties.**

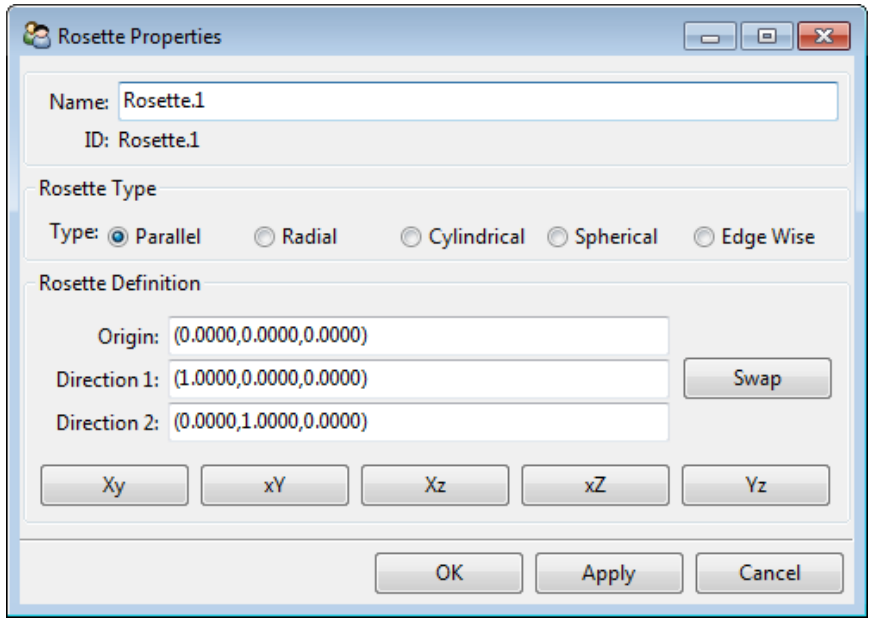




ACP Tutorial

- **Define a new *Rosette* (using the default settings):**

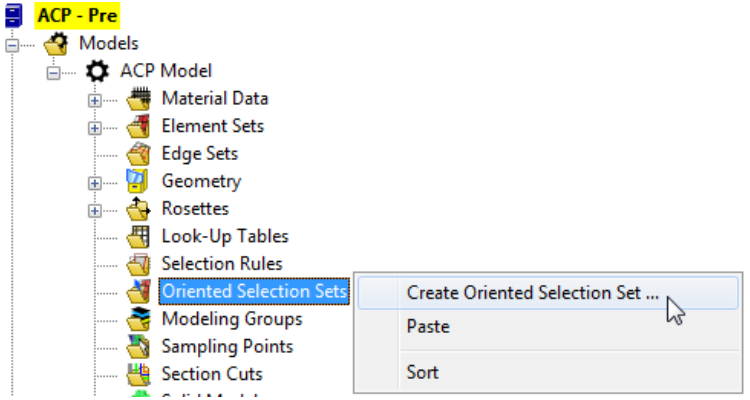


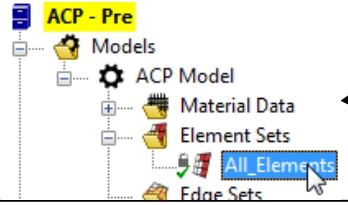


- **After the material definition the orientations and offset directions have to be defined. This is done with *Oriented Element Sets* which are defined now.**

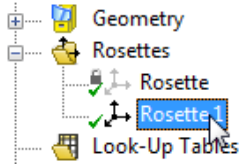
ACP Tutorial

- **Define a new Oriented Selection Set (OSS):**

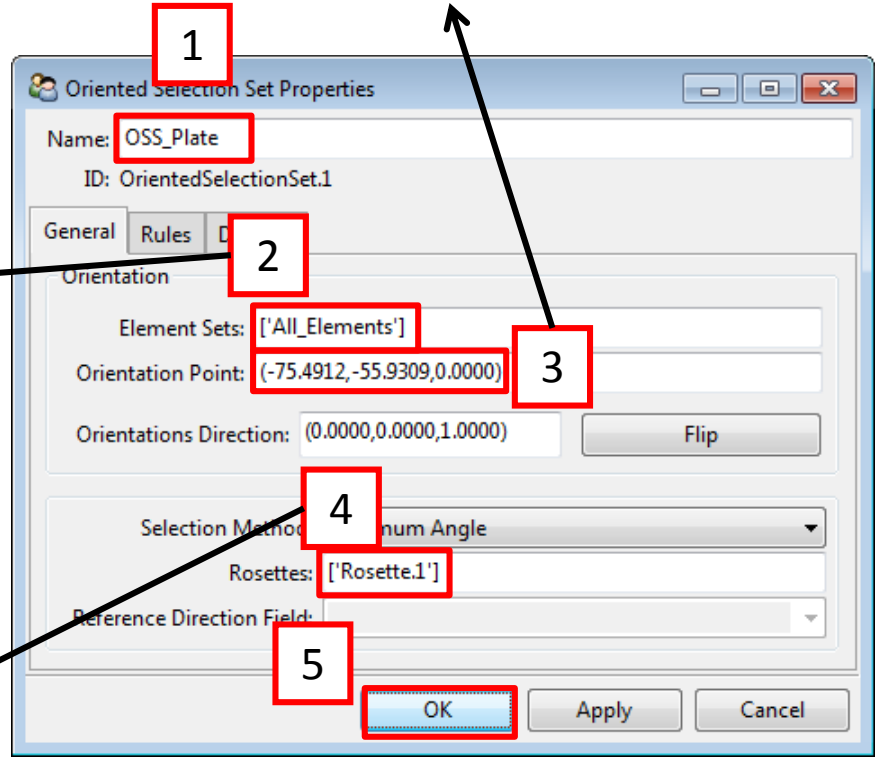




(1) Click in the Element Sets box in the dialog box
(2) Select the desired Element Set in the tree

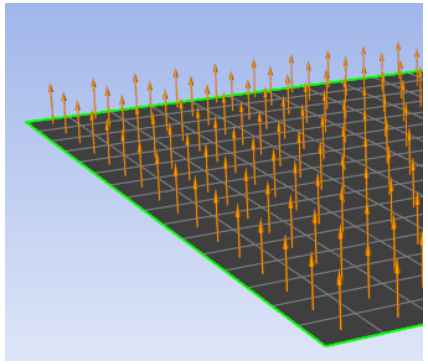


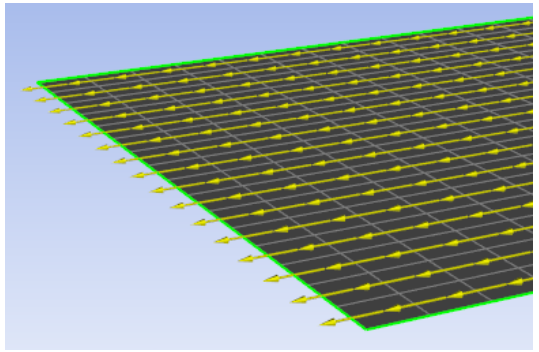
Click on an element of the model.



ACP Tutorial

(1) The Orientation Point and Direction specify the offset direction.
Toggle the button in the toolbar and select the OSS to check the offset direction.
(2) The rosette of an OSS defines the material reference (0°) direction.
Toggle the button to visualize the reference direction.
(Click Update button if no orientation is displayed.)

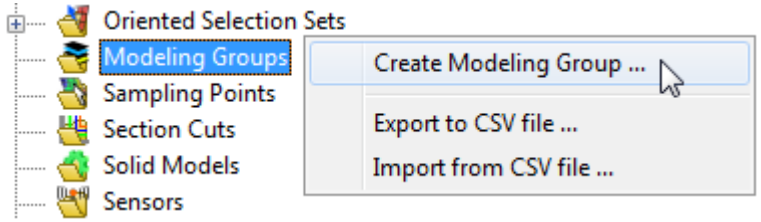


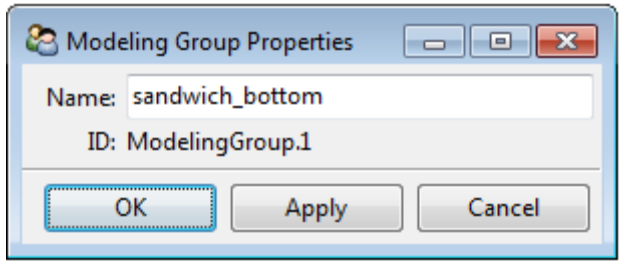


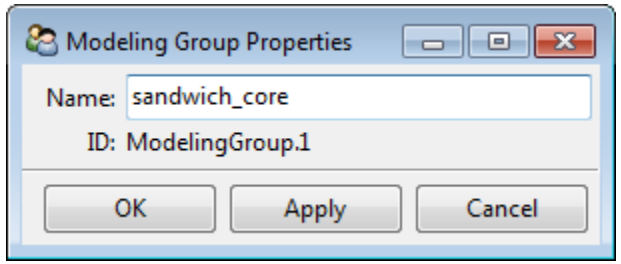
- The OSS is now used to define the layup. The offset direction of the OSS and the order of the *Modeling Plies* define the stacking sequence, the reference direction and the relative angle of the modeling plies specify the fiber alignment.

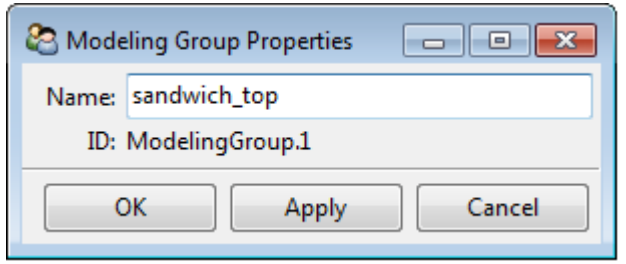
ACP Tutorial

- **Define 3 *Ply Groups*:**

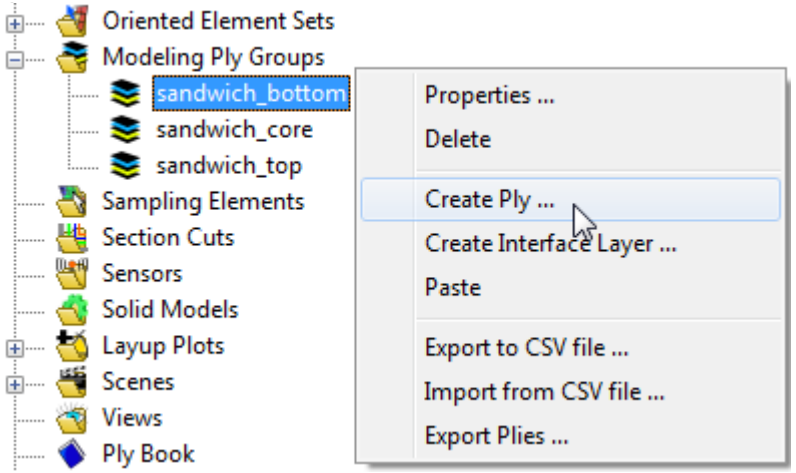






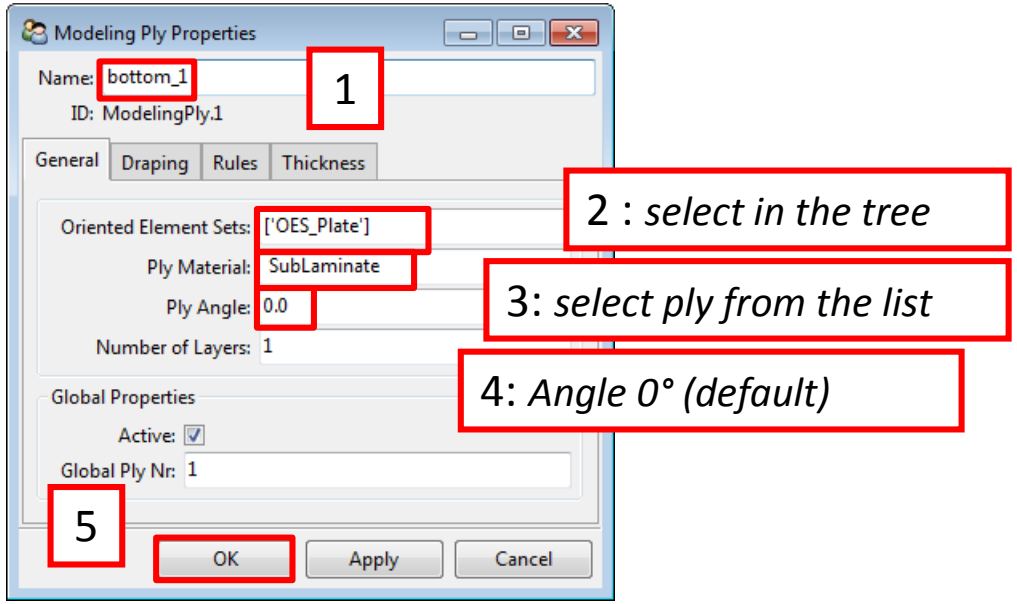


- **Create the first *Modeling Ply*:**



ACP Tutorial

1. **Configure the first ply:**

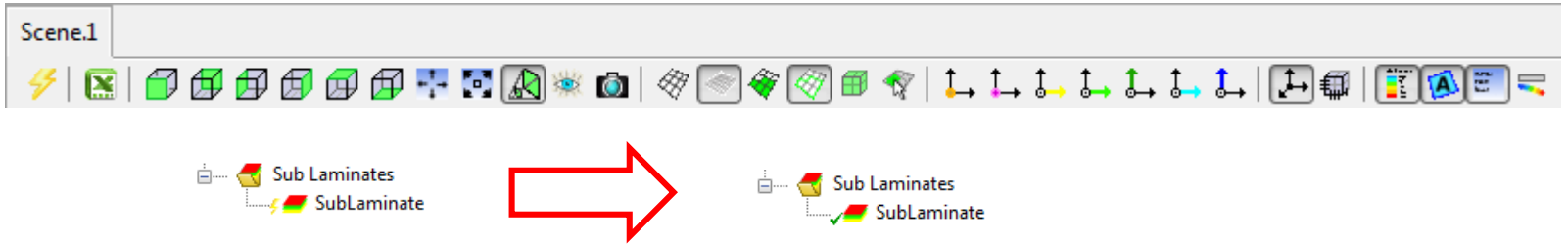


2. **Define a second ply in the "sandwich_core" *Modeling Group* with the *Fabric* Core and a Ply angle of 0°.**

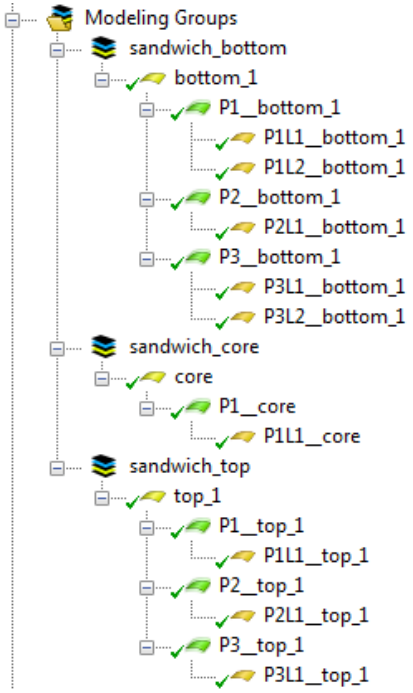
3. **Define a third ply in the "sandwich_top" *Modeling Group* with the *Fabric* UD_T700_200gsm and a Ply angle of 90°. Set *Number of Layers* to 3.**

ACP Tutorial

- **Update the model:**

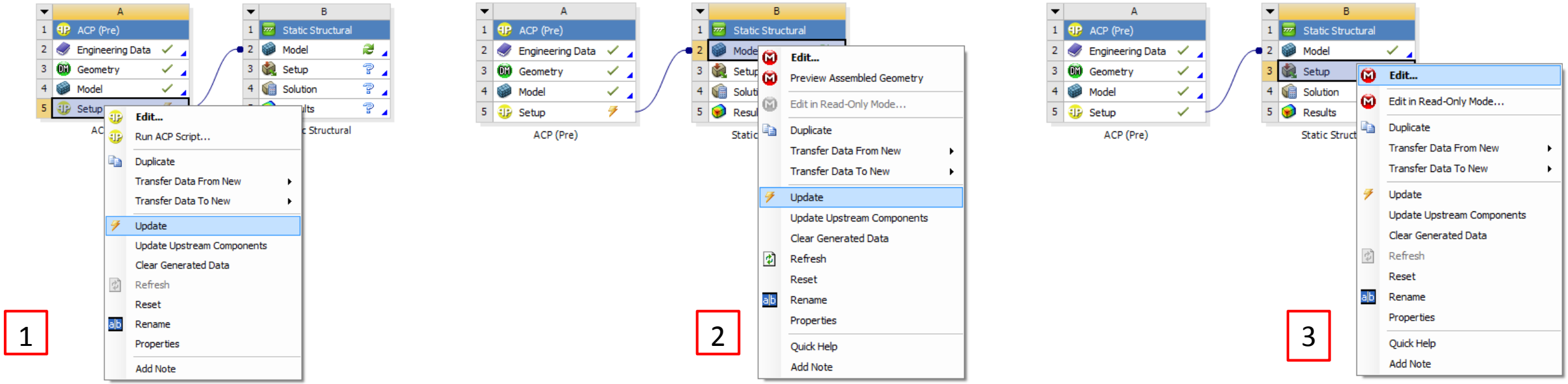


The ply definition should look like this:



ACP Tutorial

- **Return to ANSYS Project Schematic.**
  - **Update the *ACP (Pre) Setup*.**
  - **Update the *Model* of *Static Structural*.**
  - **Edit the *Setup* of *Static Structural*.**

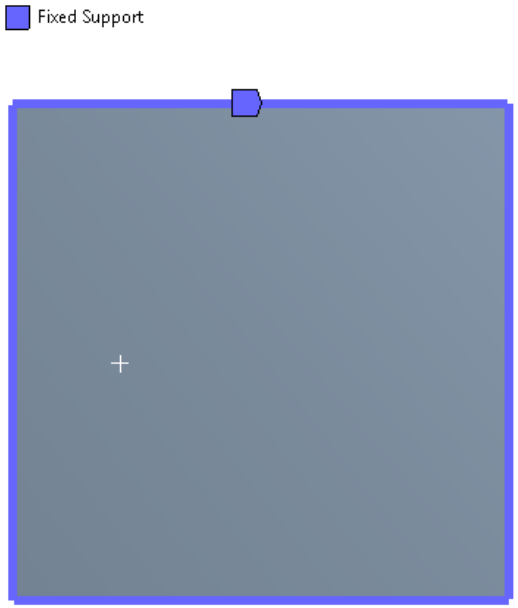


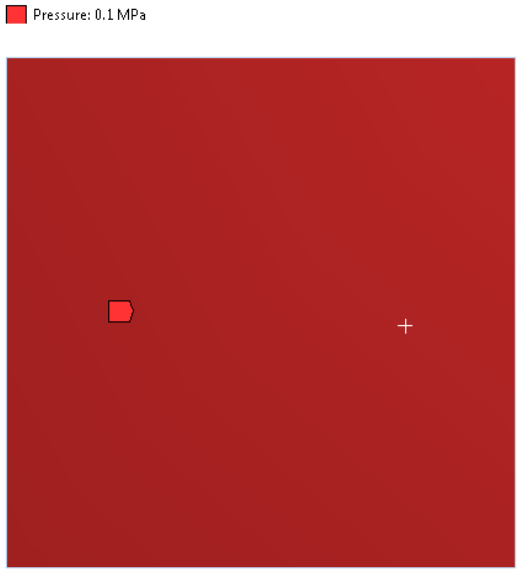
ACP Tutorial

- **Add boundary conditions and applied load to the Mechanical model.**

Details of "Fixed Support"	
Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
Definition	
Type	Fixed Support
Suppressed	No

Details of "Pressure"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Pressure
Define By	Normal To
Applied By	Surface Effect
Magnitude	0.1 MPa (ramped)
Suppressed	No

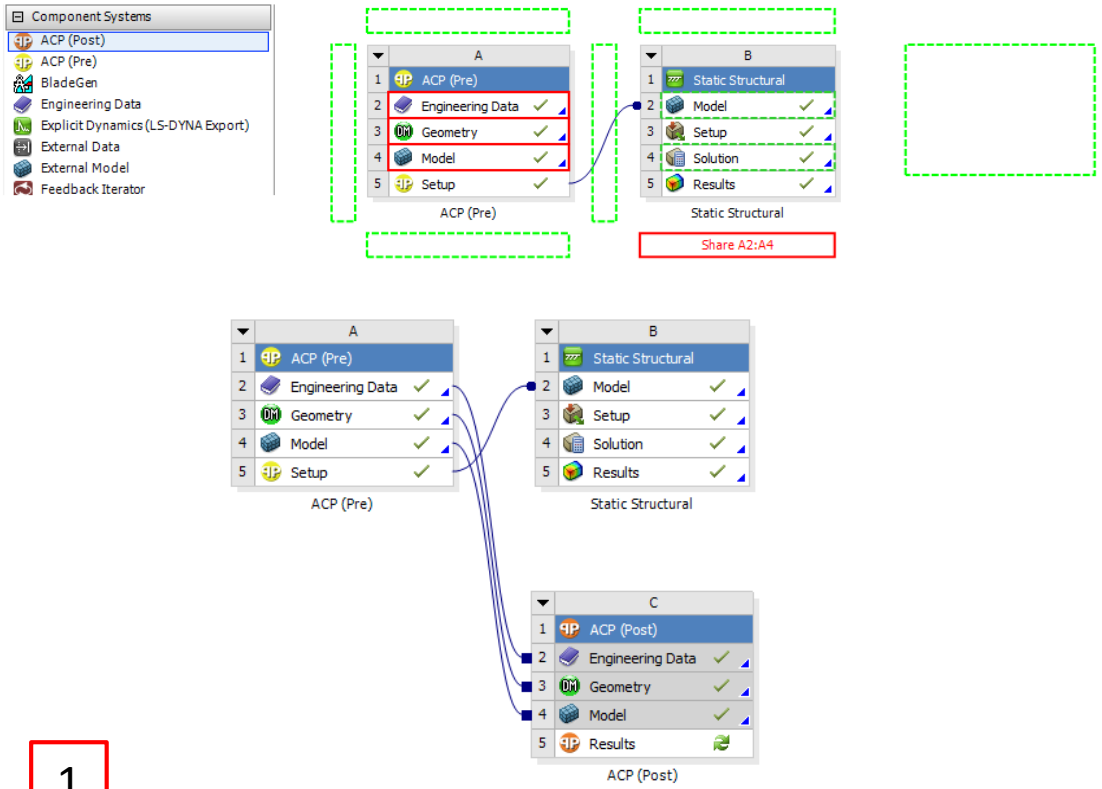




- **Return to ANSYS Project Schematic and update the whole project.**

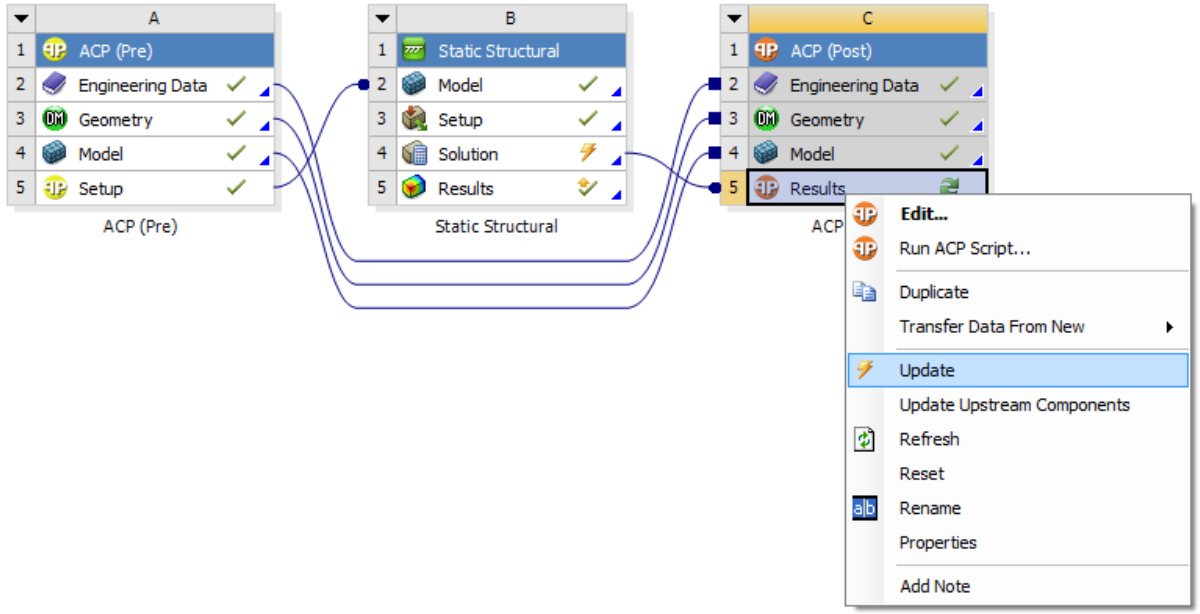
ACP Tutorial

- **Add a *ACP (Post)* System to the project**
  - **(1) Drag and Drop an ACP (Post) system over the ACP (Pre) system**
  - **(2) Link Solution of Static Structural to Results of ACP (Post)**



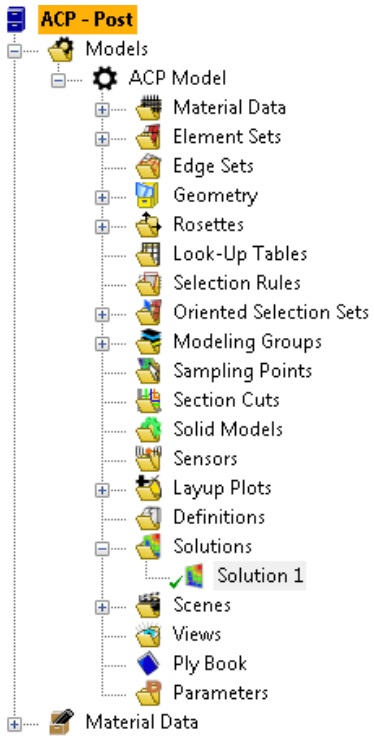
ACP Tutorial

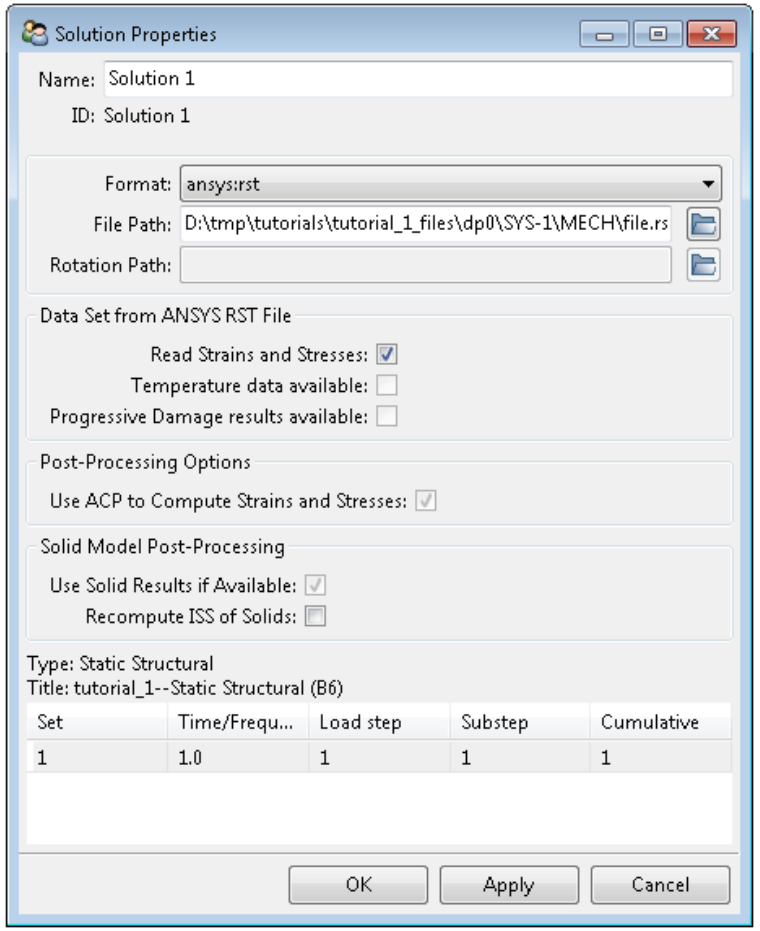
- Update *Results* of *ACP (Post)*
- Enter *Results* of *ACP (Post)* by double clicking on the cell



ACP Tutorial

The *Solution* is already imported from the *.rst file.

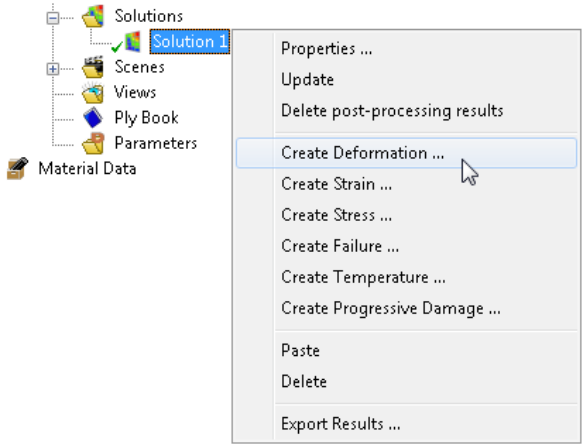


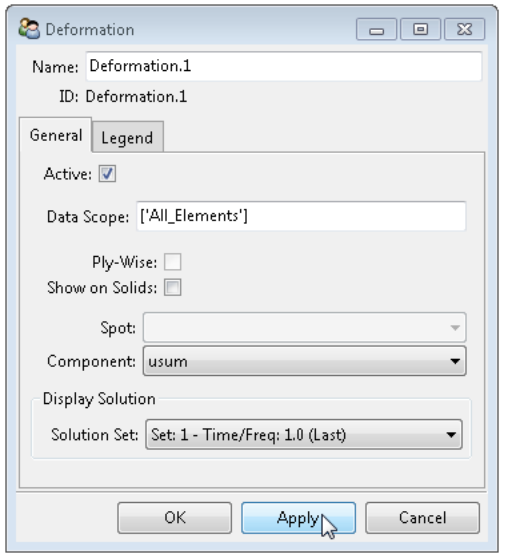


ACP Tutorial

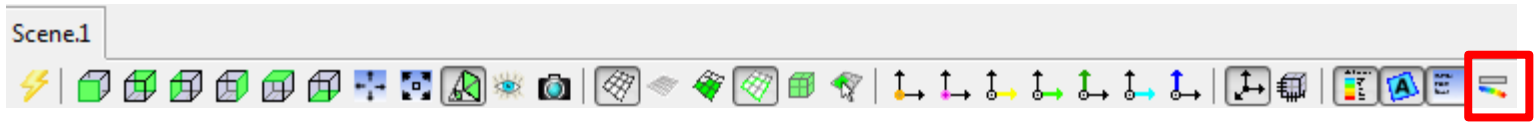
- **Create a plot to visualize the deformations:**
  - **Insert a deformation plot under the solution**
  - **Set the deformation scale in the in the toolbar**

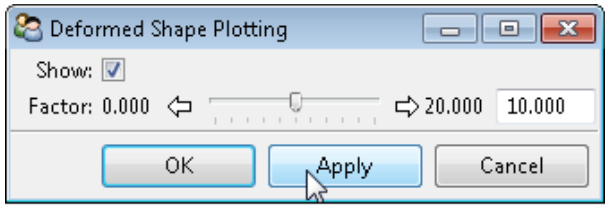
1





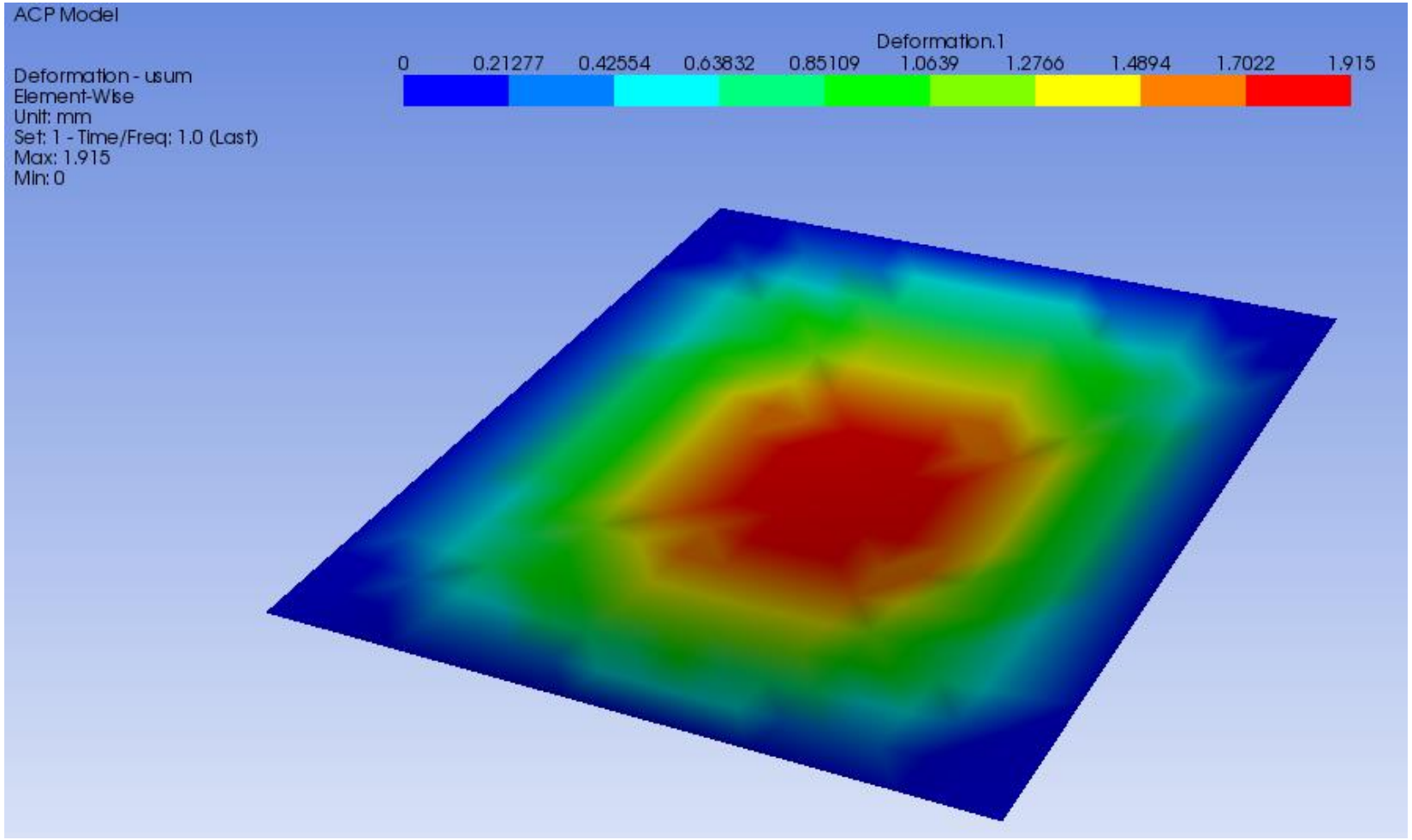
2





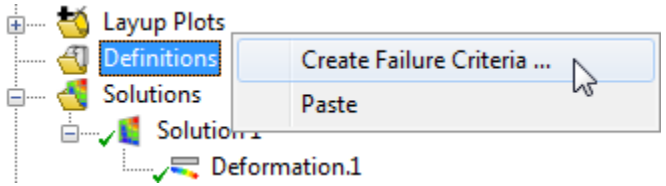
ACP Tutorial

- **Toggle element edges and surface in the toolbar.**
- **View deformation results**



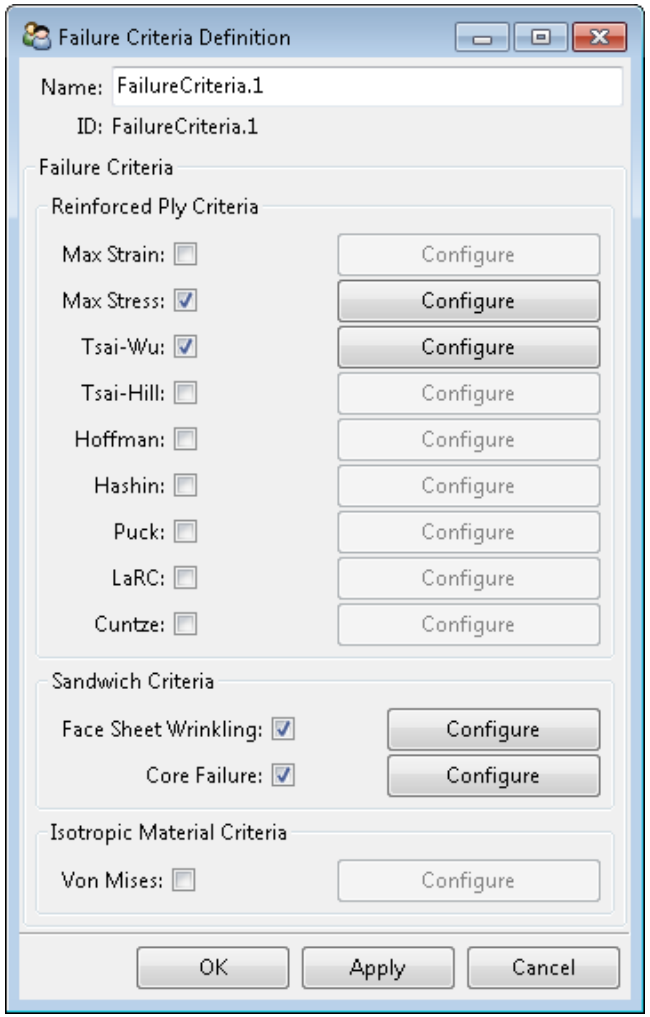
ACP Tutorial

- In the next step a combined *Failure Criteria* is configured to create an overall failure plot of the composite structure.
- For the 2 materials, the stress limits were defined in the Engineering Data at the beginning of this tutorial.
- Definitions: Create a *Failure Criteria*.



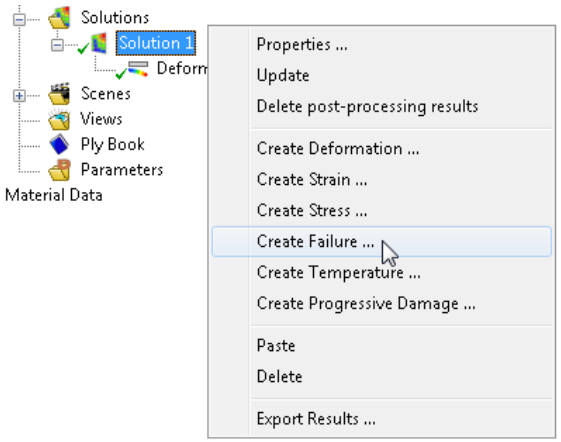
ACP Tutorial

- **Chose the following failure criteria to define a combined failure criteria definition**

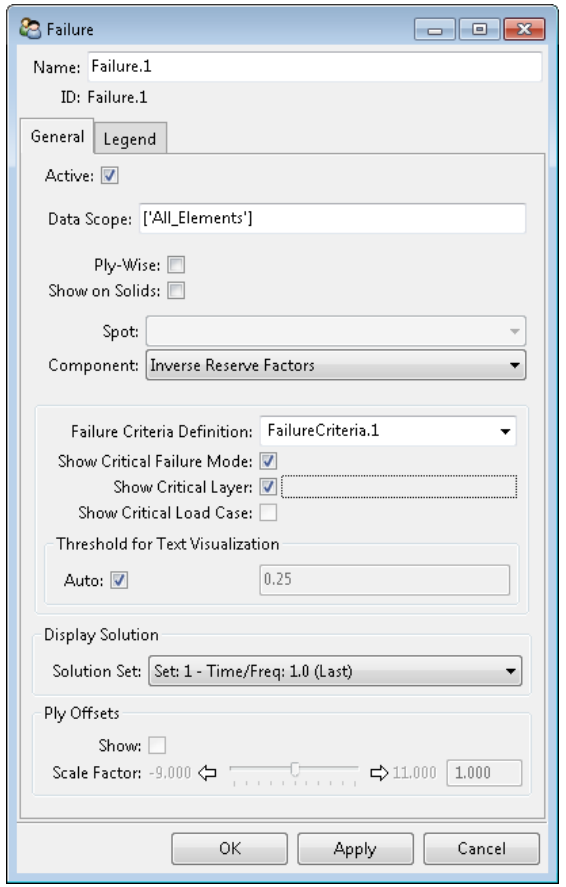


ACP Tutorial

- **Insert a failure plot under the solution**
- **Activate checkboxes for critical layer and critical layer**



1



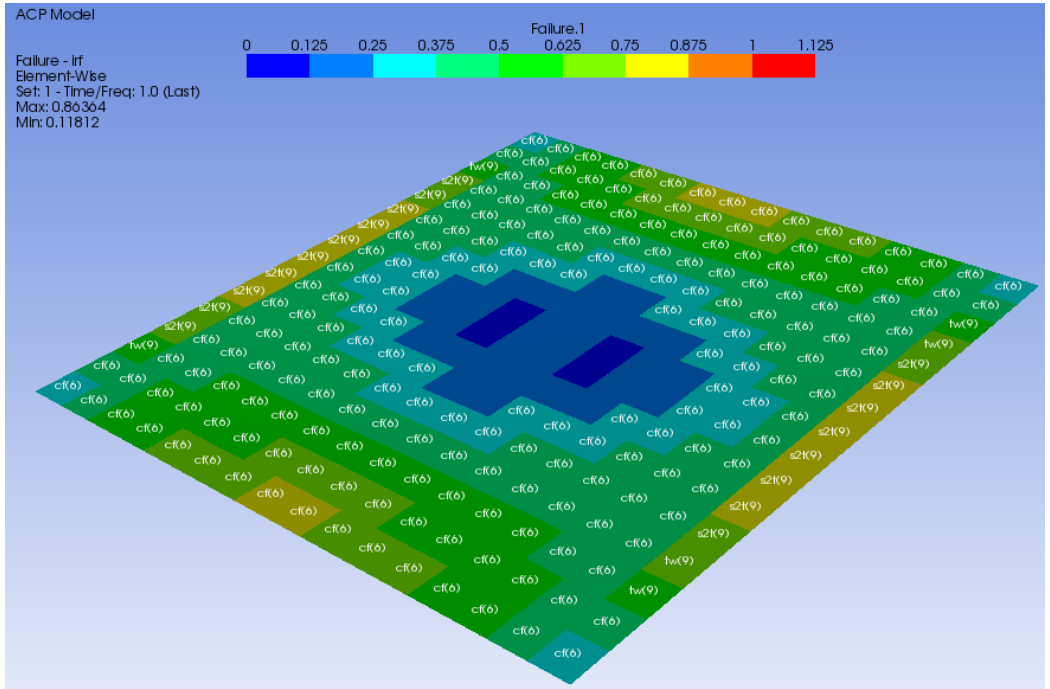
2

ACP Tutorial

Remove the deformation scale from the solution.

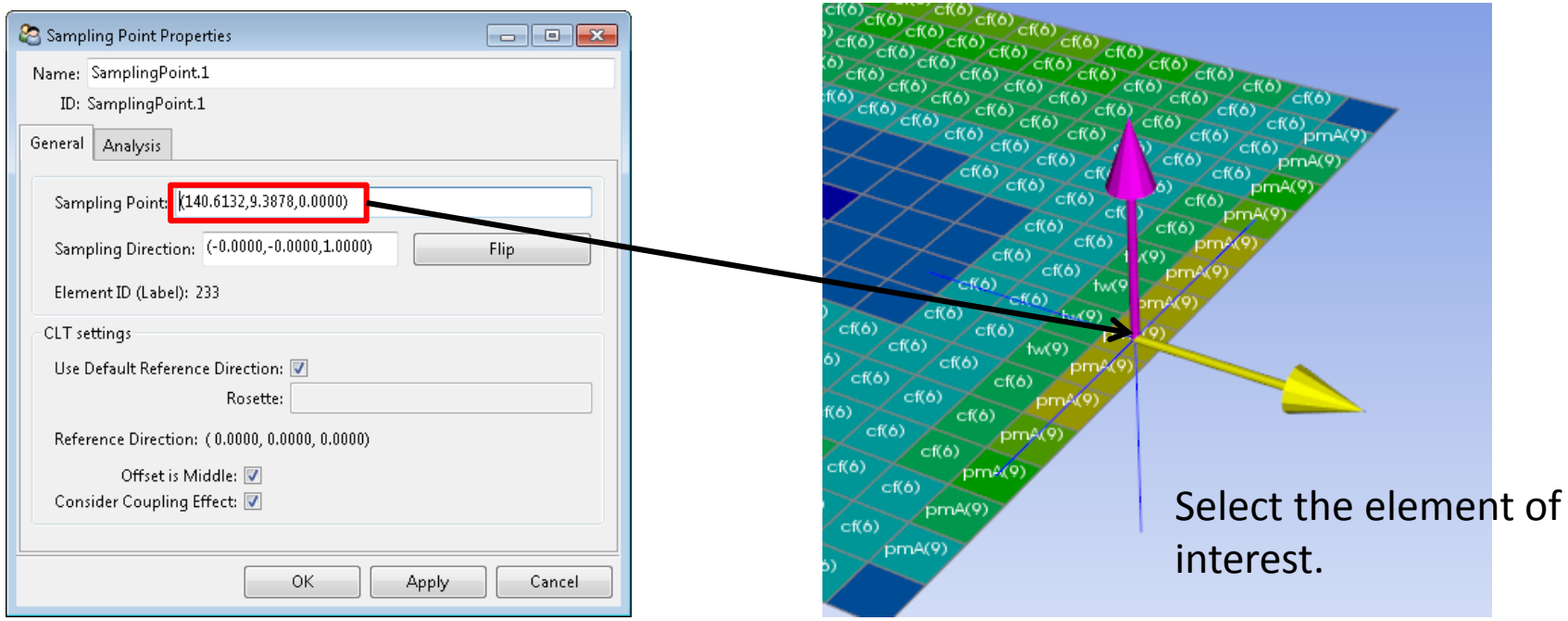
Update the model to see the overall failure plot:

- **The contour plot shows the maximum inverse reserve factor of each element (through all layers, all selected failure criteria and integration points)**
- **The text plot indicates the critical layer and the critical failure mode.**



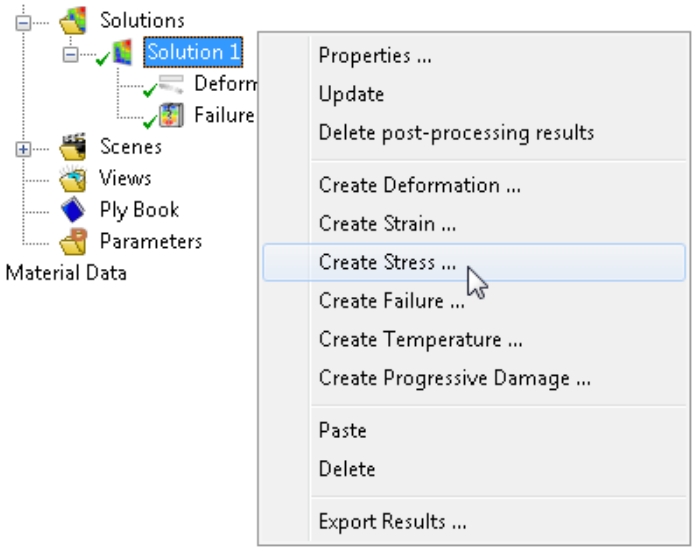
ACP Tutorial

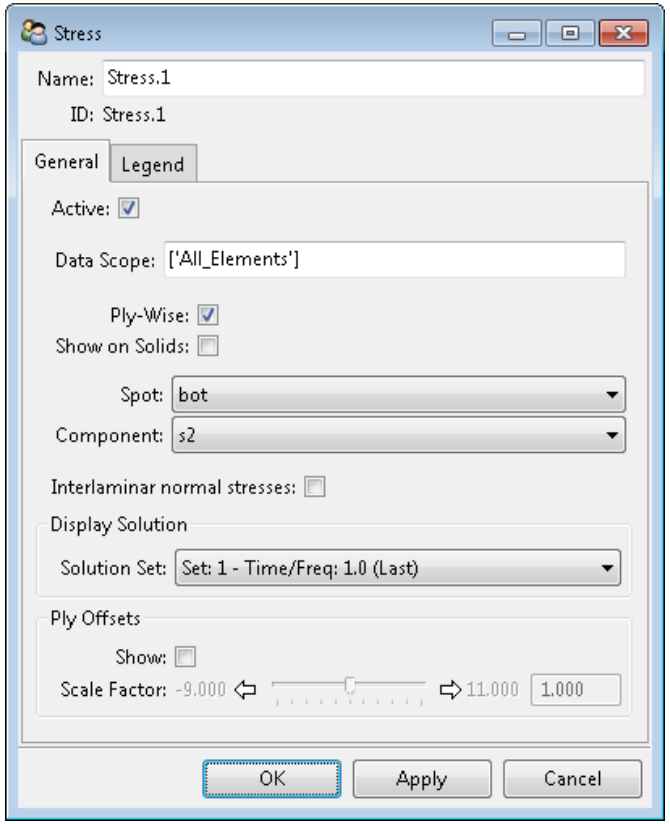
- **Sampling points and ply-wise plots can be used to investigate the results in more detail.**
- **Create a Sampling Point and select the element of interest.**



ACP Tutorial

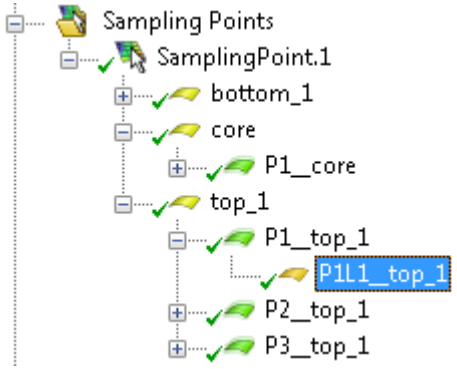
- **Insert a ply-wise stress plot for the transverse stresses *s2***
- **Ply-wise plots only show results when a ply is selected.**

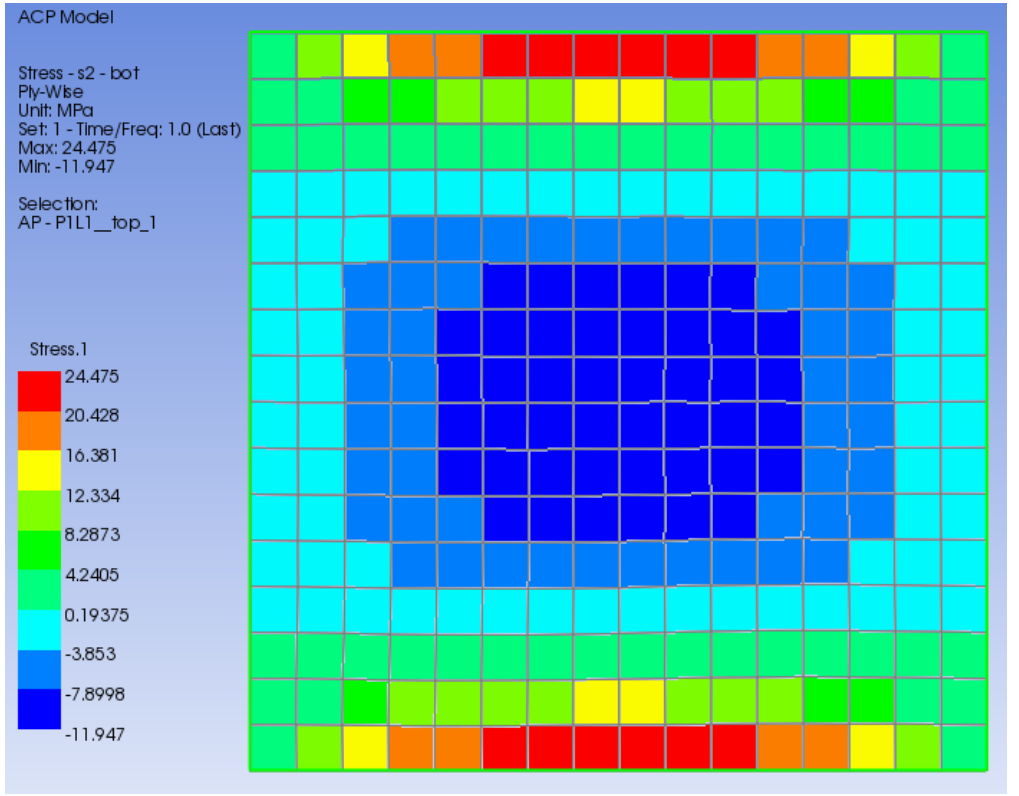




ACP Tutorial

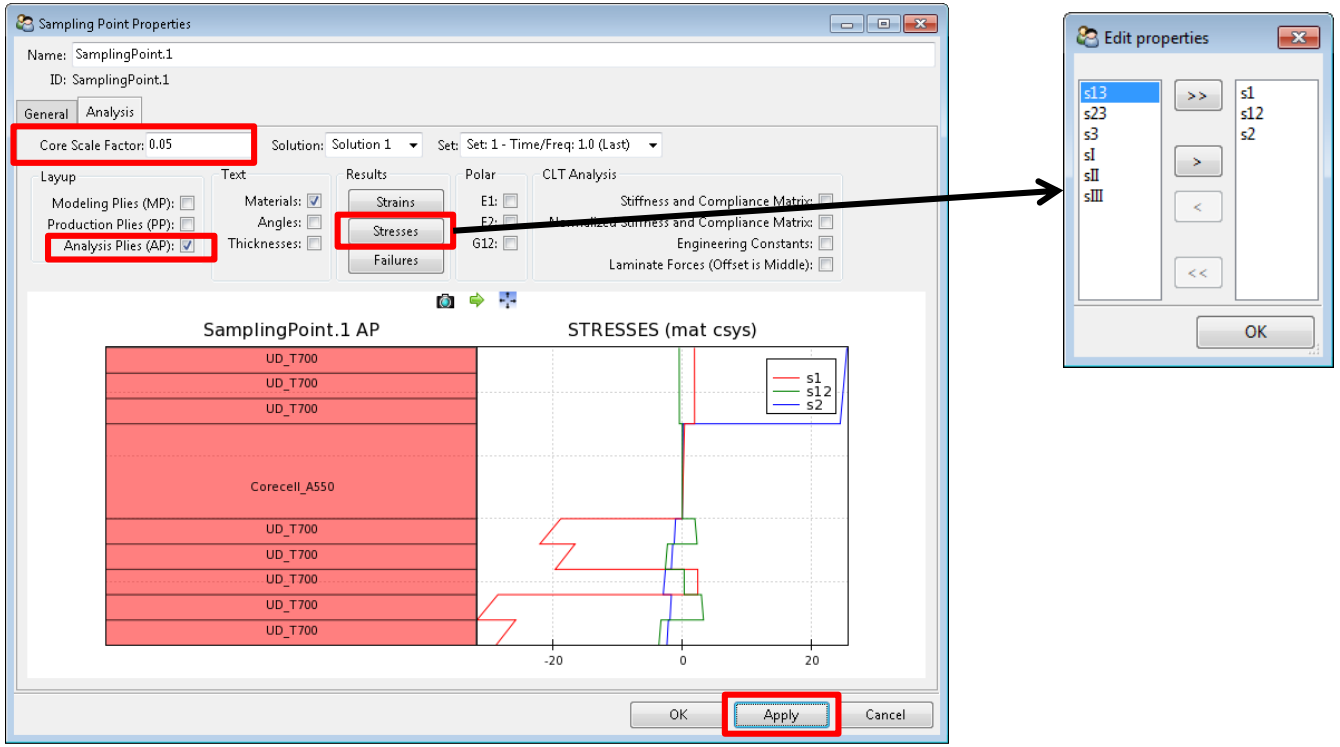
- **Select the *Analysis Plies* of the *Sampling Point* to visualize the stress distribution of a single ply.**





ACP Tutorial

- The *through-the-thickness distribution* of strains, stresses or failure results can be visualized in the *Analysis* tab of the *Sampling Point*.
- Change to the Analysis Tab of the *Sampling Point* to configure the *through-the-thickness post-processing plot*:



End of Tutorial 1

Thank you

