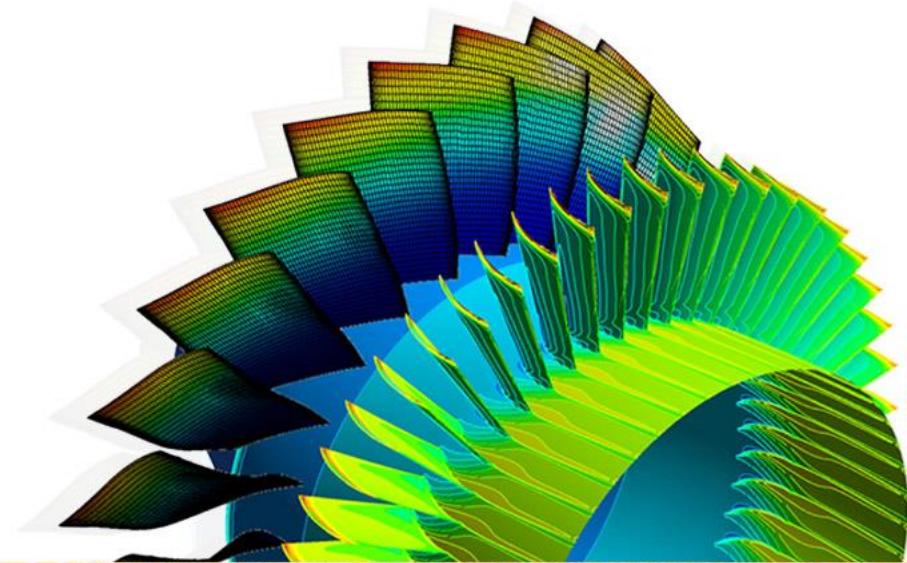




ANSYS Composite PrepPost 19.0

Workshop 10.2 – Lap Joint

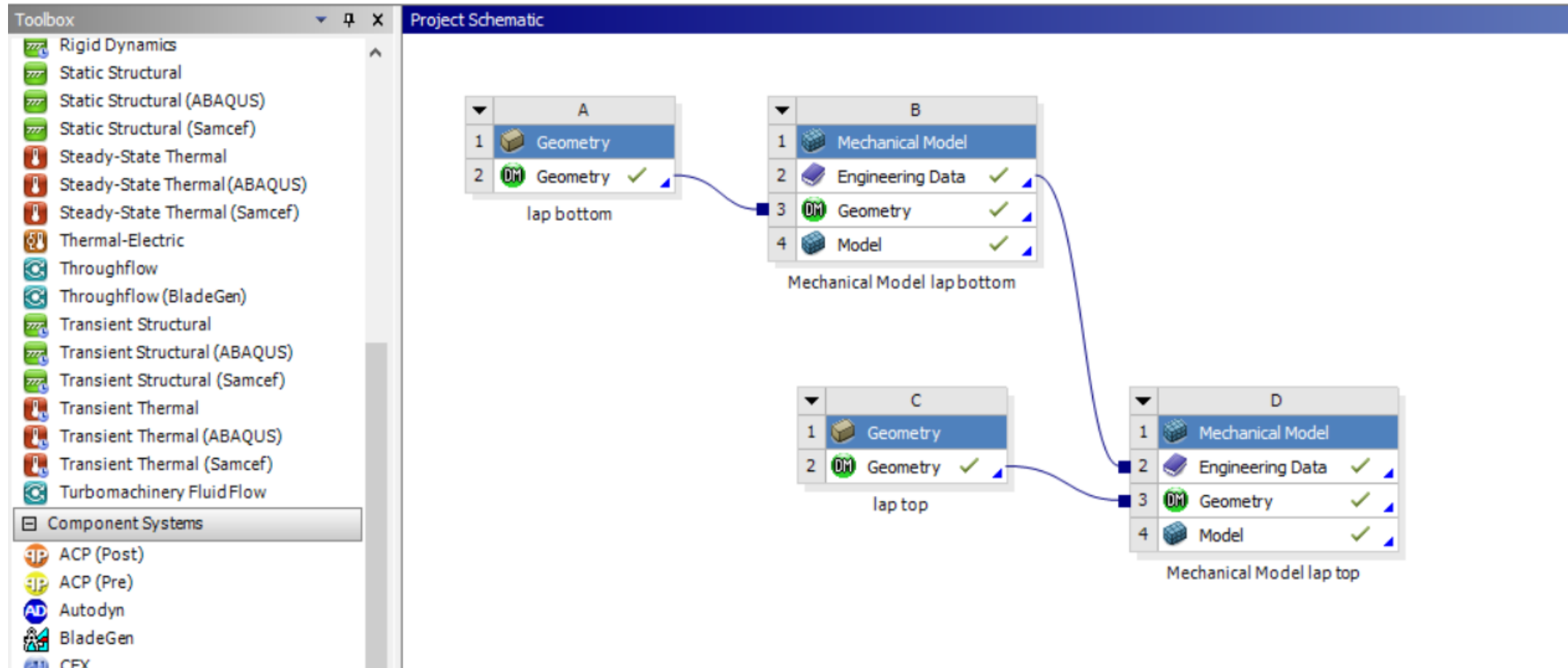


11. Workshop Single Lap

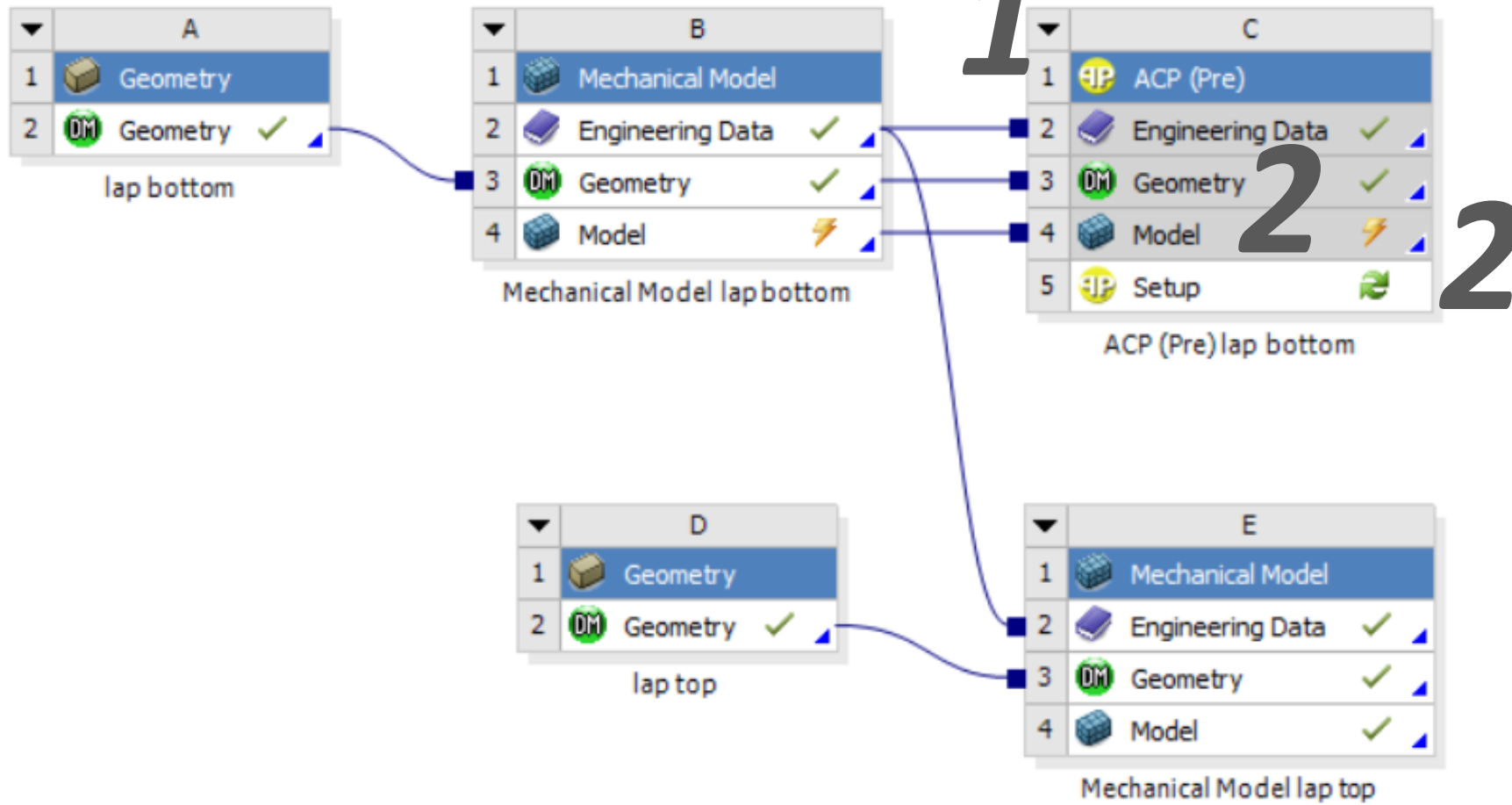
- In this workshop we will model a test on a single lap joint, where the top and bottom parts of the lap are composite materials
- A cohesive zone simulates the presence of a cured adhesive joining the lap

11. Workshop Single Lap

1. Start ANSYS Workbench and restore from archive
lap_joint_test_from_start_19.0.wbpz



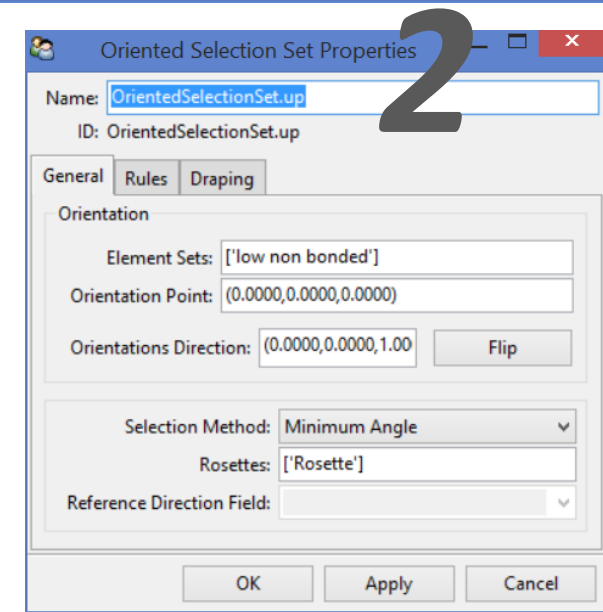
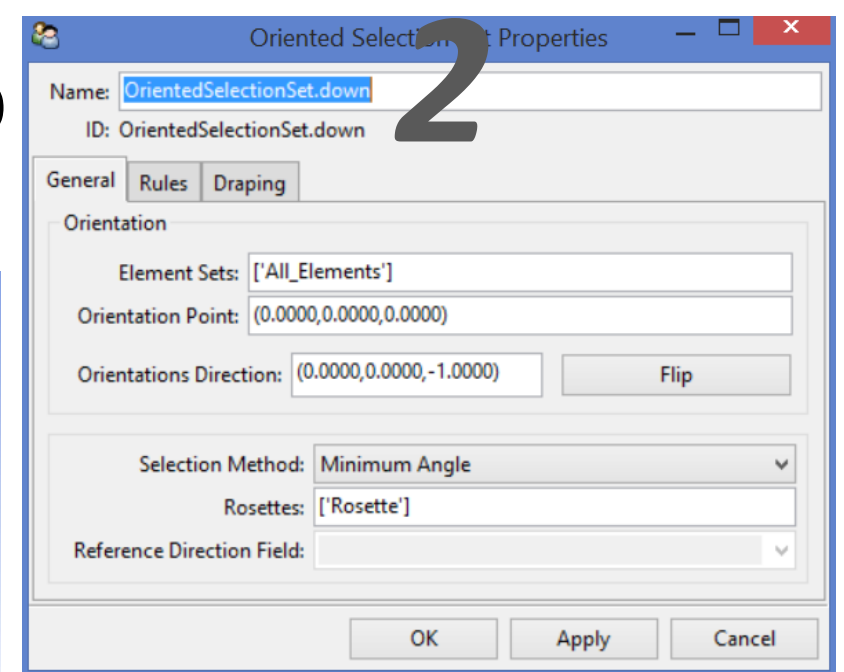
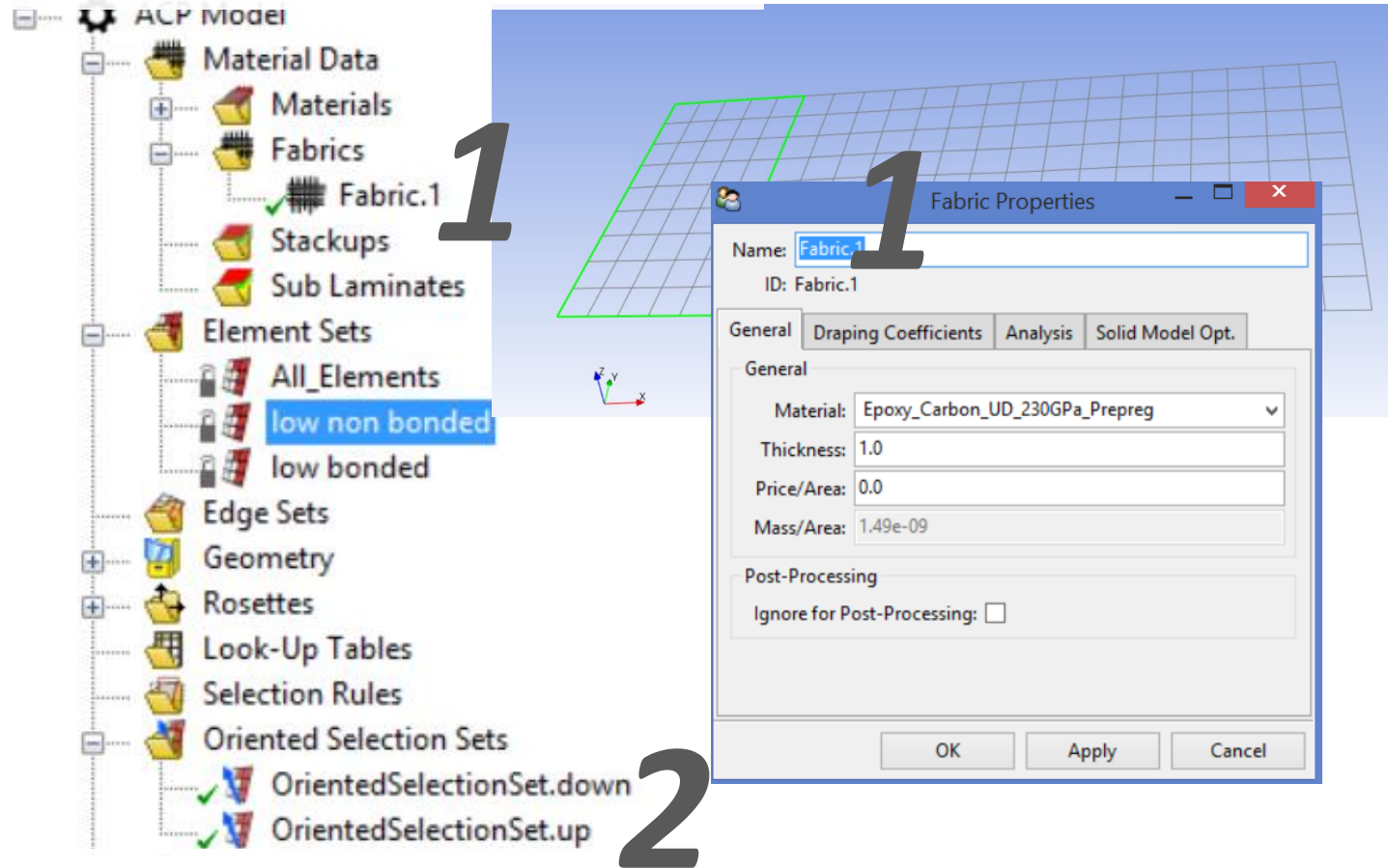
11. Workshop Single Lap



1. Drag and drop ACP (Pre) in the project schematic onto *Mechanical Model lap bottom*, rename it *ACP (Pre) lap bottom*
2. Update the project and double click on *ACP (Pre) Setup*

11. Workshop Single Lap

ACP (Pre) lap bottom

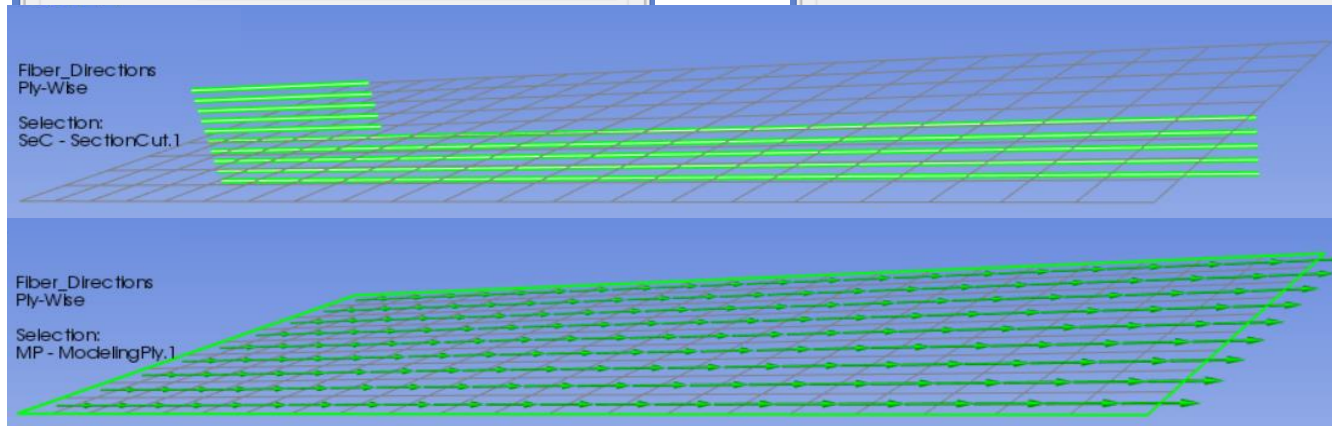
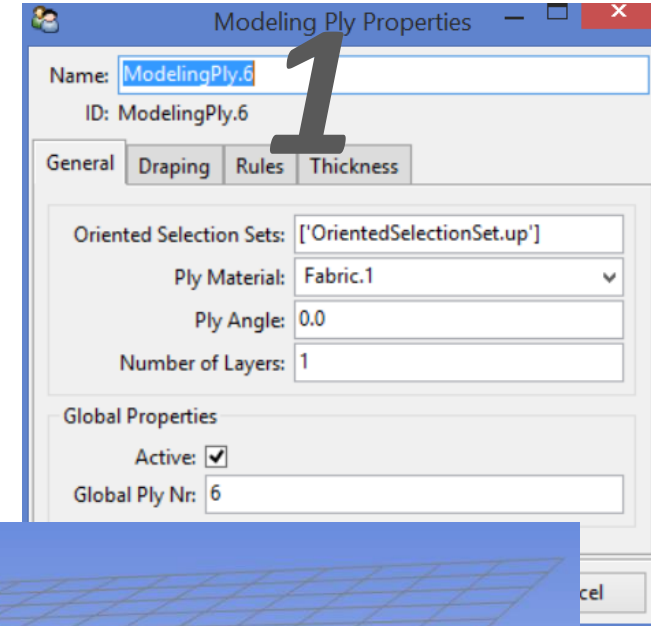
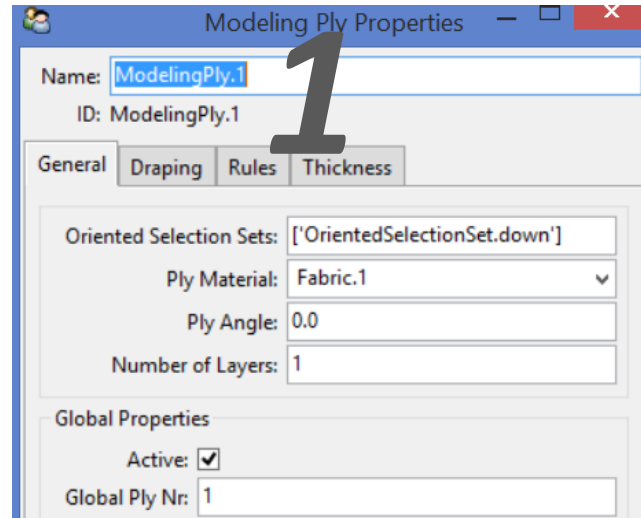
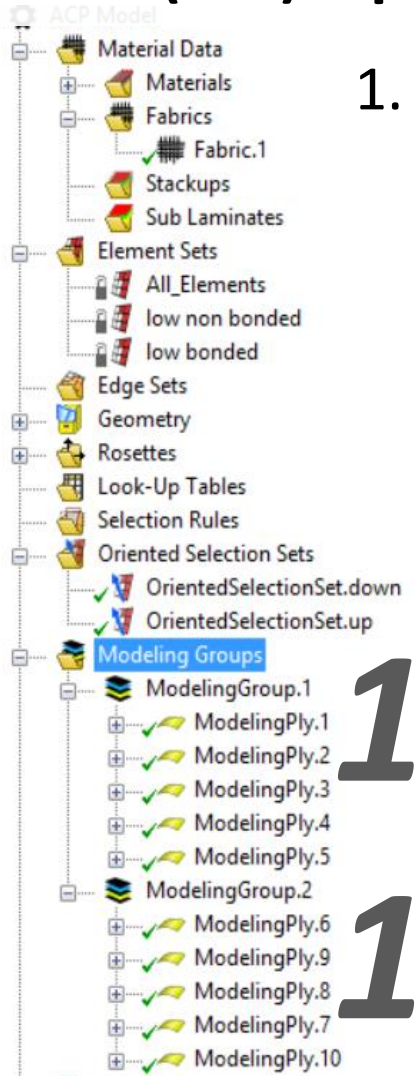


(1) Add a new fabric, (2) add two oriented selection sets for all the elements and for the element set which will not be bonded at a later stage of the workshop

11. Workshop Single Lap

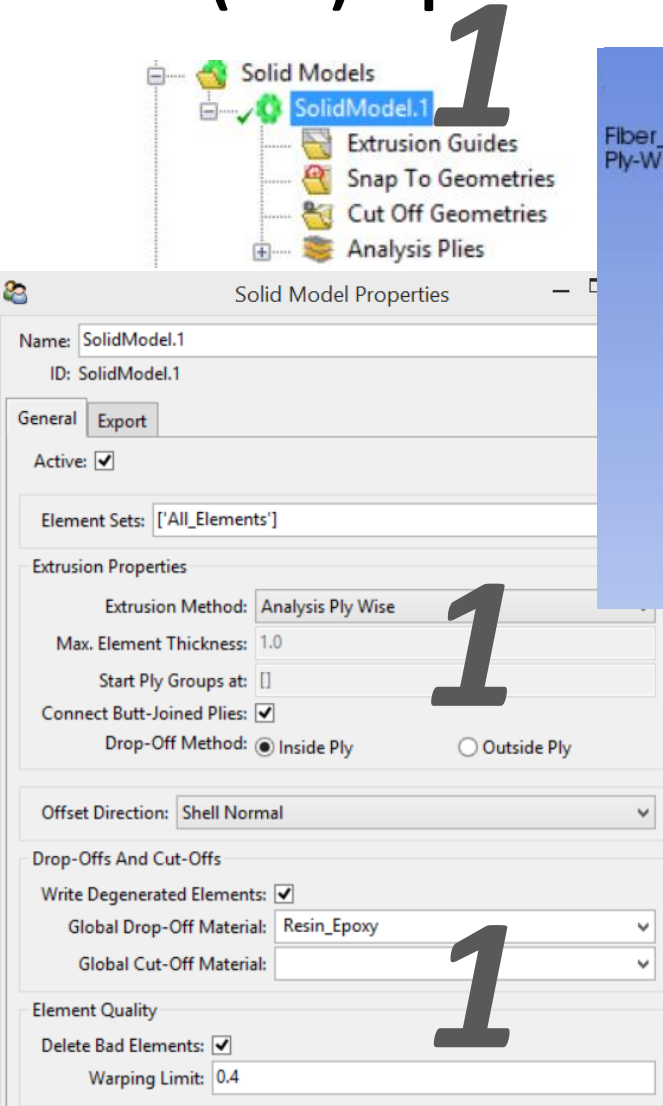
ACP (Pre) lap bottom

1. Add two *Modeling Groups* with 5 plies each, all oriented in the global x direction



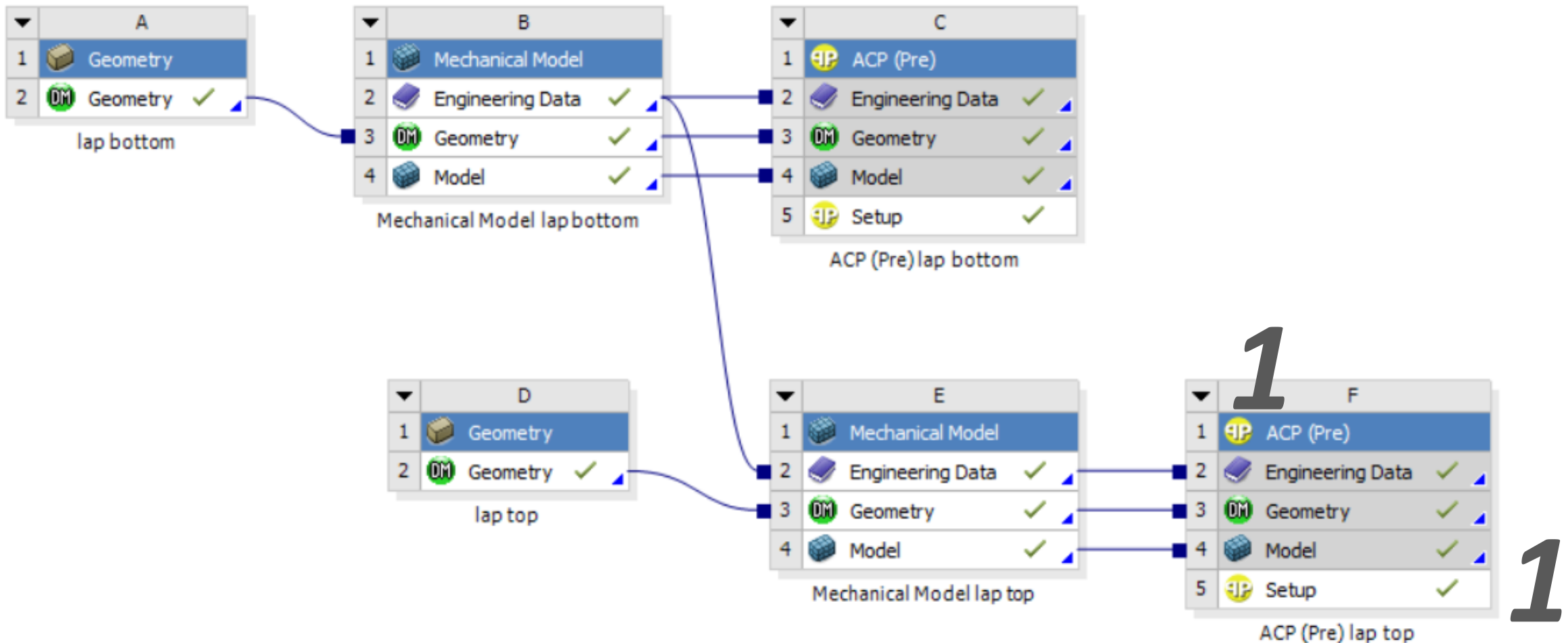
11. Workshop Single Lap

ACP (Pre) lap bottom



1. Extrude a solid model, Analysis Ply Wise and Resin_Epoxy as global drop-off material
2. Update the project and leave ACP (Pre)

11. Workshop Single Lap



1. Drag and Drop another ACP (Pre) model onto the mechanical model for the top part creating the lap, rename it ACP (Pre) lap top and double click on *Setup*

11. Workshop Single Lap

ACP (Pre) lap top

1

2

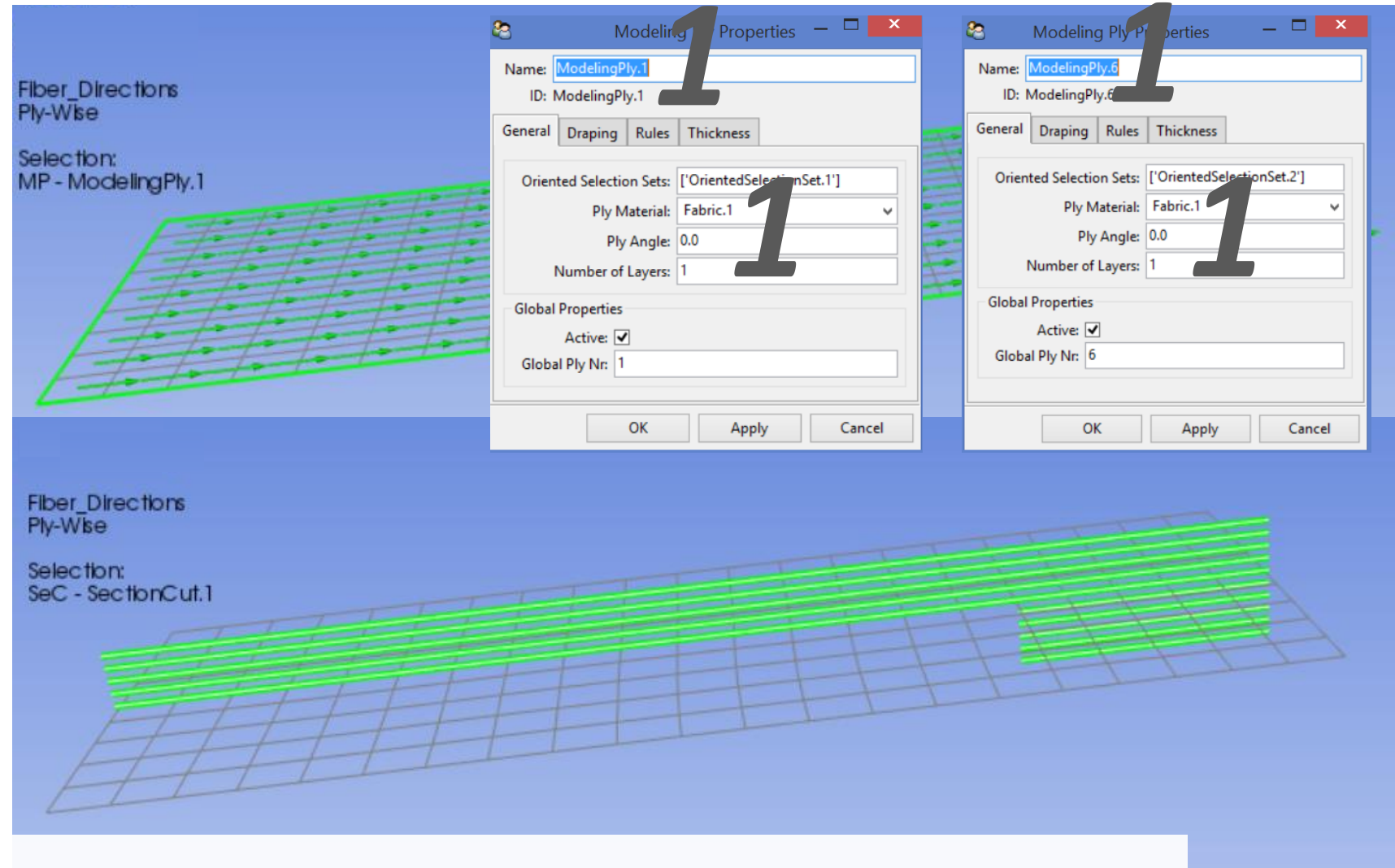
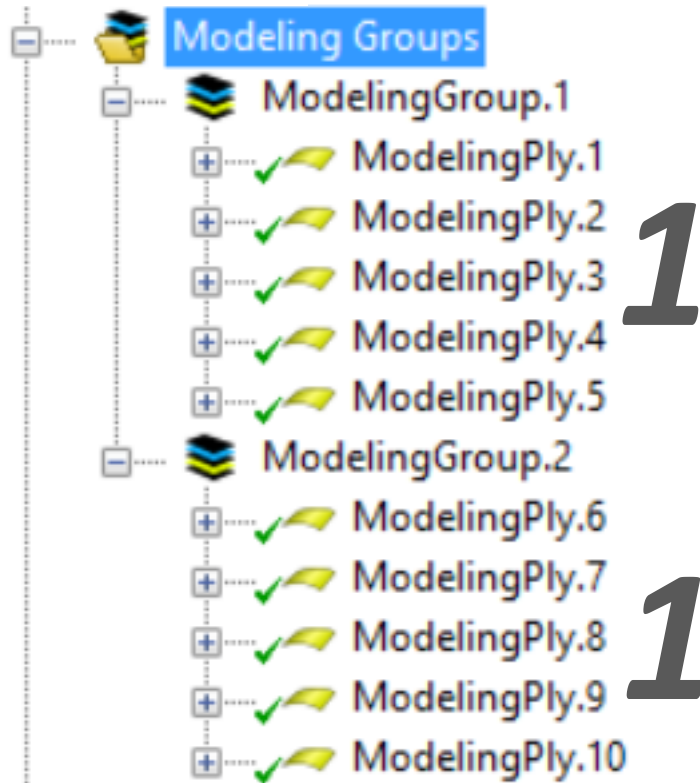
1

2

(1) Add a new fabric, (2) add two oriented selection sets for all the elements and for the element set which will not be bonded at a later stage of the workshop

11. Workshop Single Lap

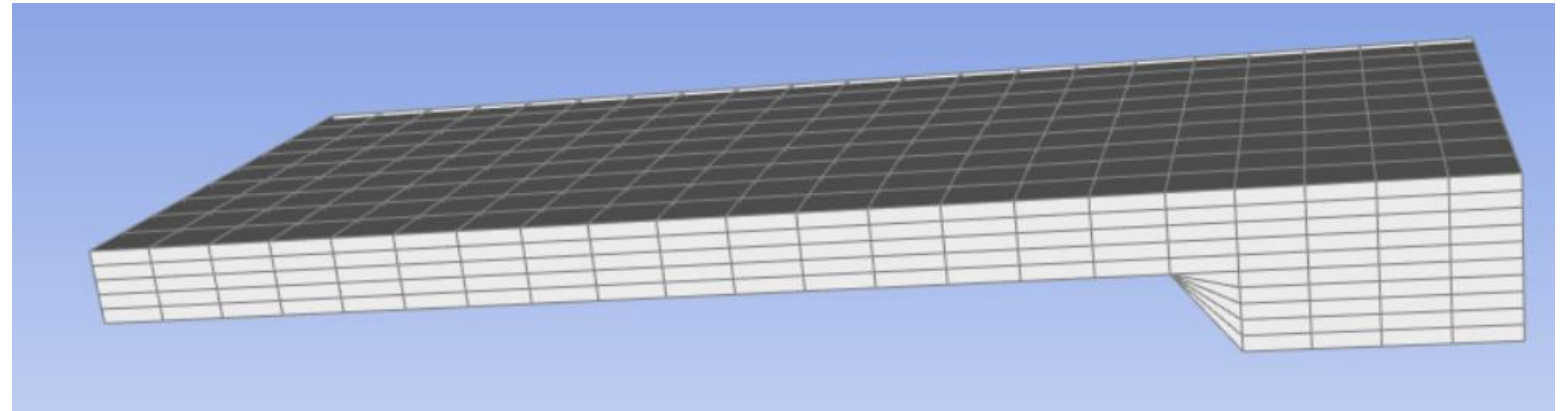
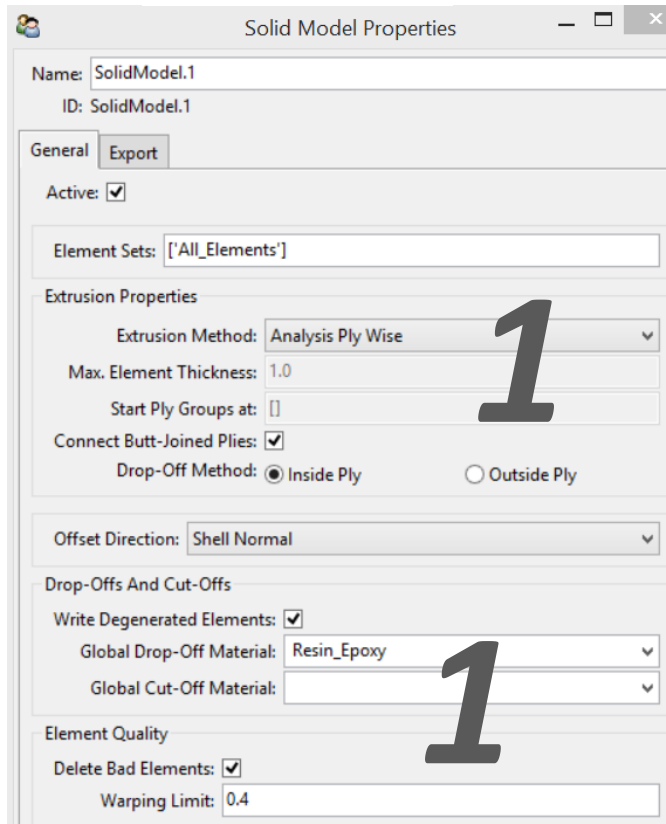
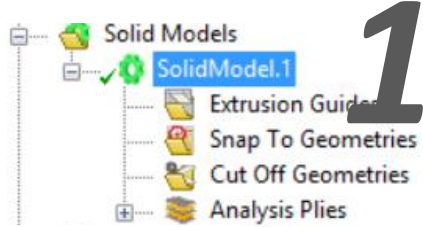
ACP (Pre) lap top



1. Add two *Modeling Groups* with 5 plies each, all oriented in the global x direction

11. Workshop Single Lap

ACP (Pre) lap top



1. Extrude a solid model, Analysis Ply Wise and Resin_Epoxy as global drop-off material
2. Update the project and leave ACP (Pre)

11. Workshop Single Lap

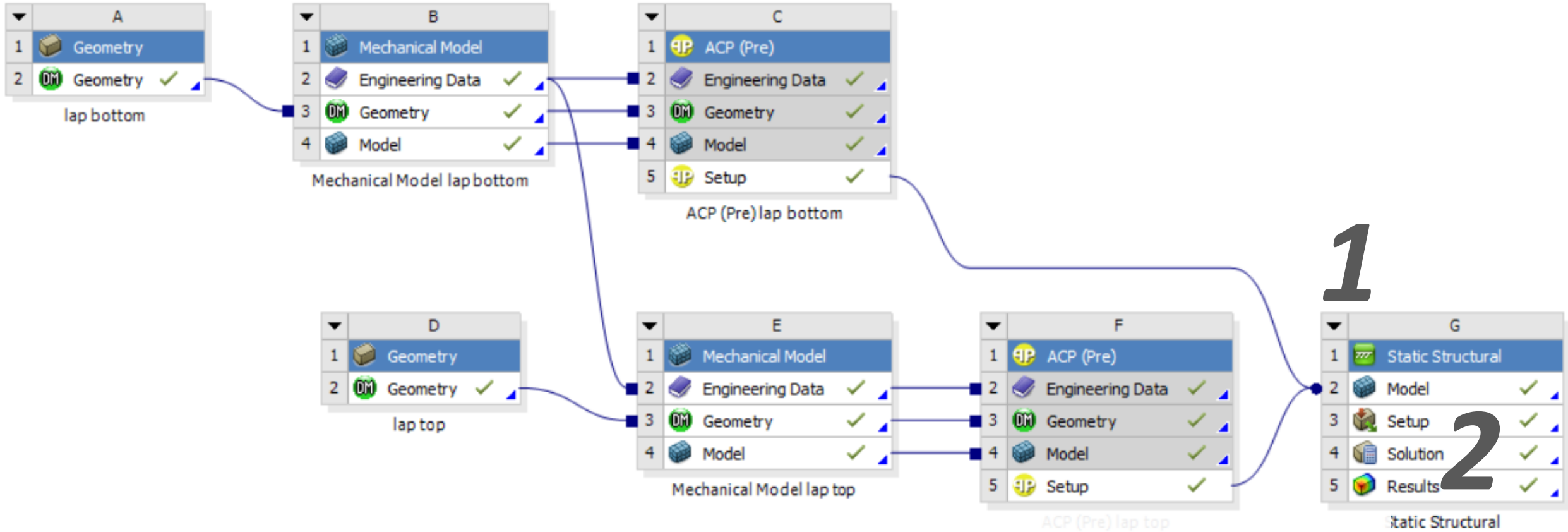
The screenshot shows the ANSYS Engineering Data interface. On the left is the 'Toolbox' with various material models. The main area displays the 'Outline of Schematic B2, C2, E2, F2: Engineering Data' with a table of materials. Row 3, 'cohesive', is selected. A dialog box titled 'Properties of Outline Row 3: cohesive' is open, showing the properties for the cohesive law. A large number '1' is overlaid on the dialog, indicating the first step in the process.

	A	B	D
1	Contents of Engineering Data		Description
2	Material		
3	cohesive		
4	Epoxy_Carbon_UD_230GPa_Pregreg		
5	Resin_Epoxy		
6	Structural Steel		Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material		

	A	B	C	D	E
1	Property	Value	Unit		
2	Separation-Distance based Debonding				
3	Debonding Interface Mode	Mixed			
4	Tangential Slip Under Normal Compression	Yes			
5	Maximum Normal Contact Stress	20	MPa		
6	Contact Gap at the Completion of Debonding	0.5	mm		
7	Maximum Equivalent Tangential Contact Stress	6	MPa		
8	Tangential Slip at the Completion of Debonding	0.5	mm		
9	Artificial Damping Coefficient	0.001	s		

1. Check the properties of the cohesive law in the Engineering Data

11. Workshop Single Lap



1. Add a *Static Structural* block in the project schematic and connect its cell *Model* to the *Setup* cell of the *ACP (Pre)* blocks modeling the two parts of the lap joint (transfer composite solid data)
2. Update the project and open *Mechanical* by double clicking the cell *Solution*

11. Workshop Single Lap

1

Outline

- Geometry
- Coordinate Systems
- Connections
 - Contacts
 - Contact Region
- Mesh
- Fracture
- Contact Debonding
- Imported Plies
- Named Selections
- Static Structural (G3)
 - Analysis Settings
 - Displacement lap top
 - Fixed Support
- Solution (G4)
 - Solution Information
 - Directional Deformation
 - Contact Tool
 - Force Reaction

Details of "Contact Region"

Scoping Method	Geometry Selection
Contact	2 Faces
Target	2 Faces
Contact Bodies	SolidModel.1(ACP (Pre) lap top)
Target Bodies	SolidModel.1(ACP (Pre) lap bottom)
Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	0.31721 mm
Suppressed	No

1

Contact Region

Contact Body View

0.00 20.00

Target Body View

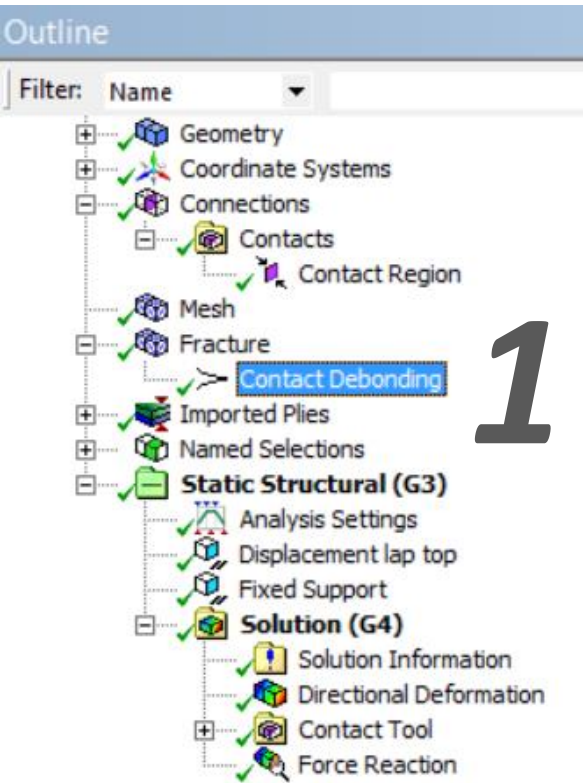
Details of "Contact Region"

Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	0.31721 mm
Suppressed	No
Formulation	Pure Penalty
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Pinball Region	Program Controlled
Contact Geometry Correction	None
Target Geometry Correction	None

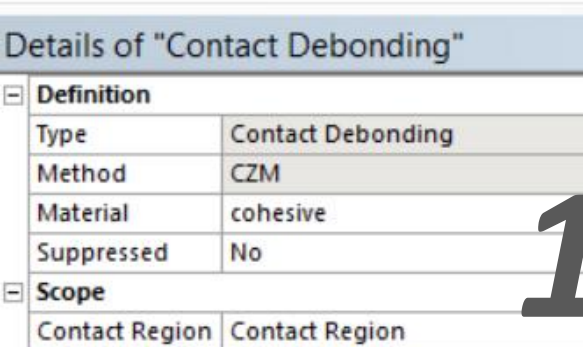
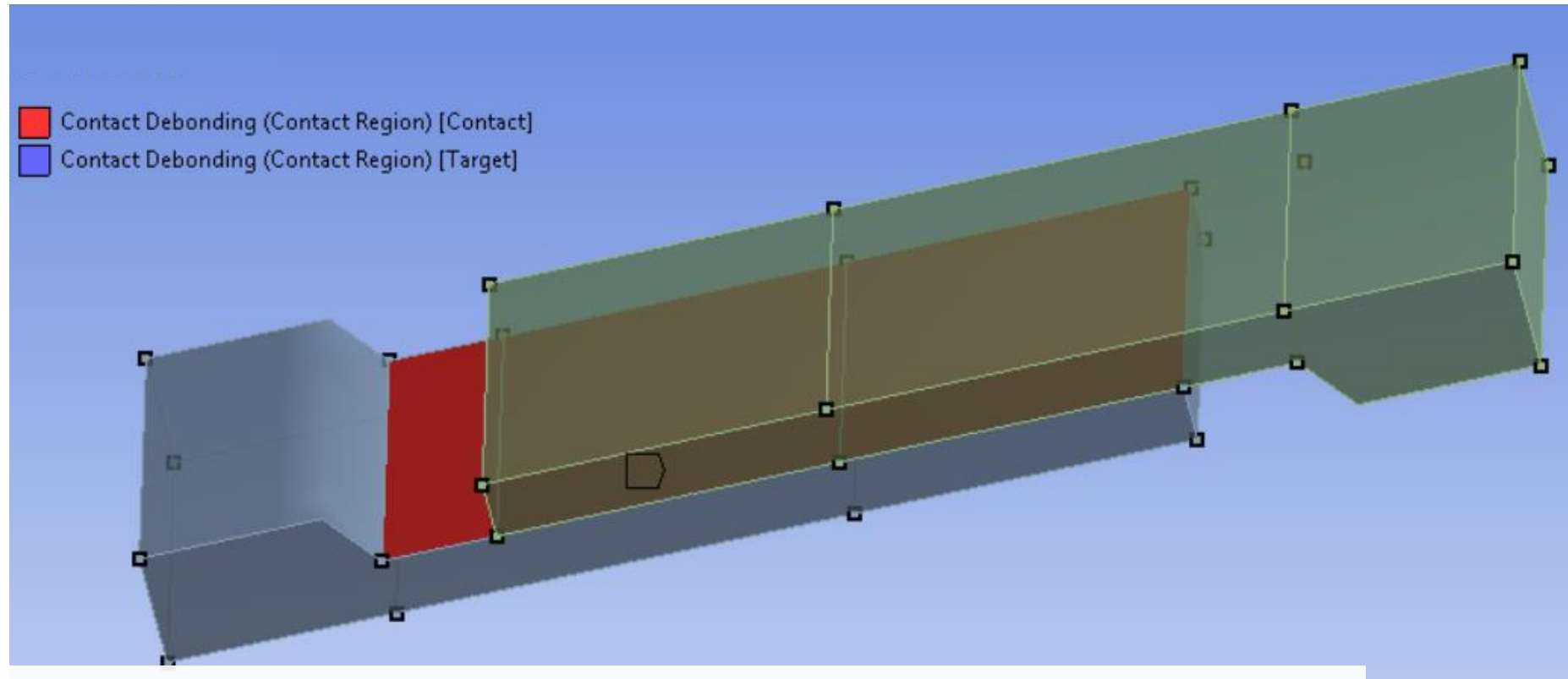
1

1. Check the bonded contact automatically created by Workbench, **modify the formulation to Pure Penalty**

11. Workshop Single Lap



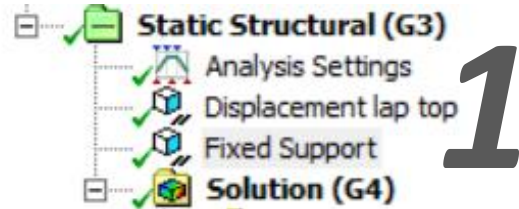
1



1

1. Insert *Contact-Debonding* in Fracture, select CZM method, the cohesive material defined in *Engineering Data*, and the *Contact Region* defined before by Workbench

11. Workshop Single Lap

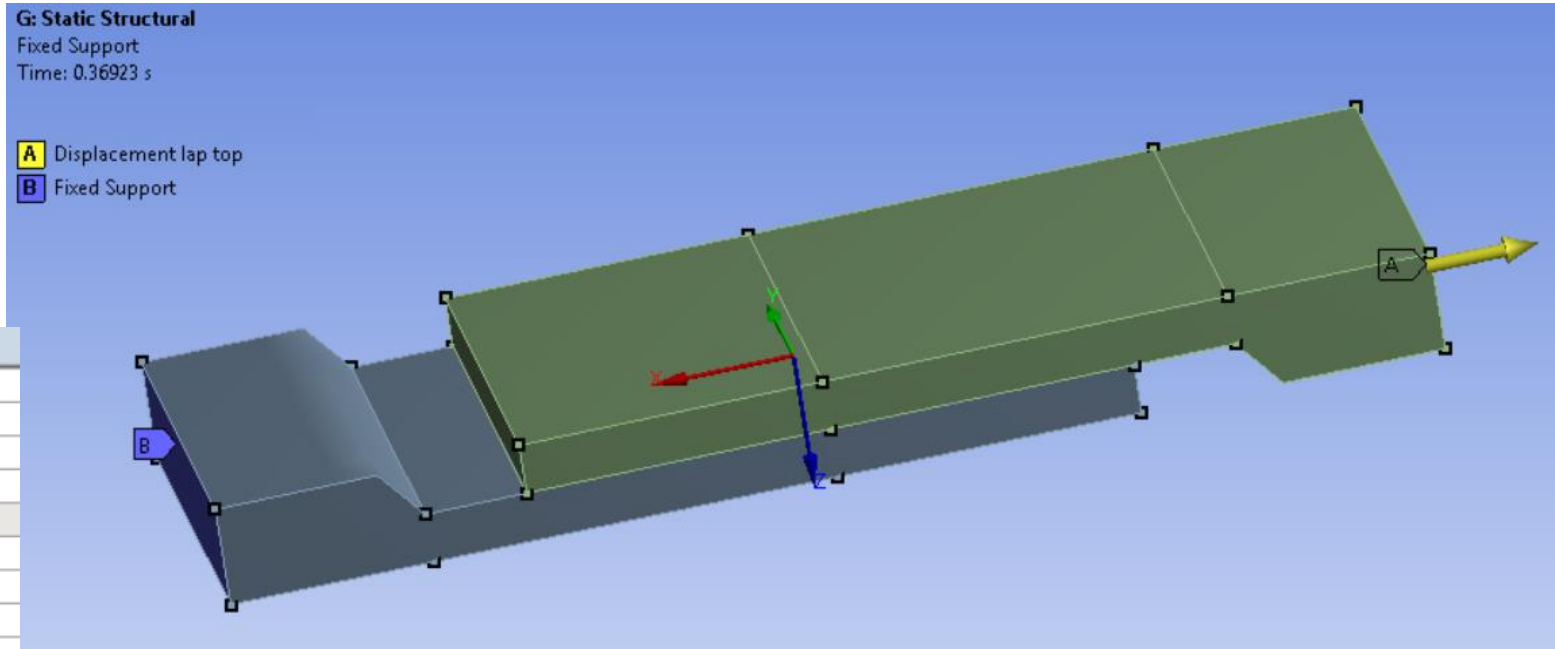


Details of "Fixed Support"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Fixed Support
Suppressed	No

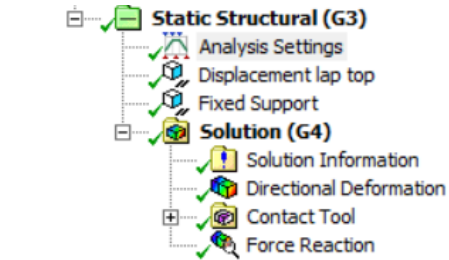
Details of "Displacement lap top"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
<input type="checkbox"/> X Component	-0.8 mm (ramped)
<input type="checkbox"/> Y Component	0. mm (ramped)
<input type="checkbox"/> Z Component	0. mm (ramped)
Suppressed	No



1. Impose boundary conditions (fixed support and applied displacement)

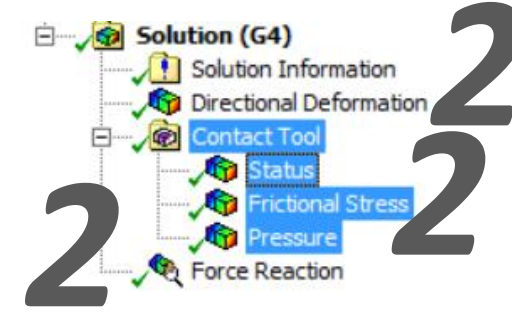
11. Workshop Single Lap



Details of "Analysis Settings"

Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	On
Define By	Substeps
Initial Substeps	30.
Minimum Substeps	30.
Maximum Substeps	100.
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Solver Pivot Checking	Program Controlled
Large Deflection	Off
Inertia Relief	Off
Fracture	On

1



Details of "Directional Deformation"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Directional Deformation
Orientation	X Axis
By	Time
<input type="checkbox"/> Display Time	0.25 s
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
Results	
<input type="checkbox"/> Minimum	-0.20257 mm
<input type="checkbox"/> Maximum	2.6013e-003 mm
Minimum Occurs On	SolidModel.1(ACP (Pre) lap bottom)
Maximum Occurs On	SolidModel.1(ACP (Pre) lap top)
Minimum Value Over Time	
<input type="checkbox"/> Minimum	-0.8 mm

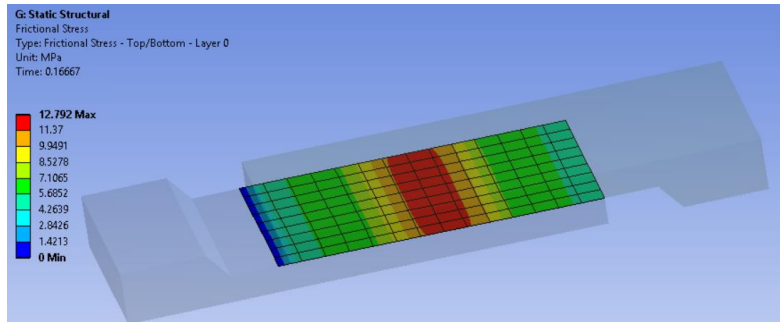
Details of "Force Reaction"

Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Displacement lap top
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	All
<input type="checkbox"/> Display Time	0.35 s
Results	
Maximum Value Over Time	
<input type="checkbox"/> X Axis	1.7306e-010 N
<input type="checkbox"/> Y Axis	0.61232 N
<input type="checkbox"/> Z Axis	1.5603e-011 N
<input type="checkbox"/> Total	18995 N
Minimum Value Over Time	
<input type="checkbox"/> X Axis	-18963 N
<input type="checkbox"/> Y Axis	-0.63948 N
<input type="checkbox"/> Z Axis	-1114.7 N

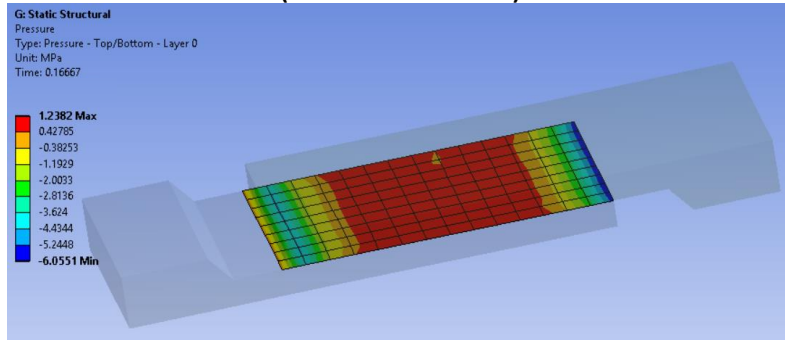
1. Define the substeps for the non-linear analysis
2. Add in the solution plots for directional deformation and force reaction in the loading direction, and contact tool to plot the status of the contact elements, the pressure and frictional (shear) stress levels
3. Solve the model

11. Workshop Single Lap

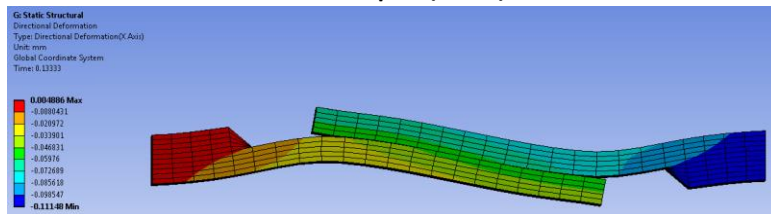
Frictional (Shear) Stress at Peak



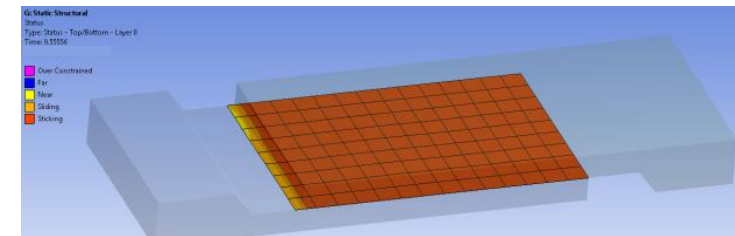
Pressure (Normal Stress) at Peak



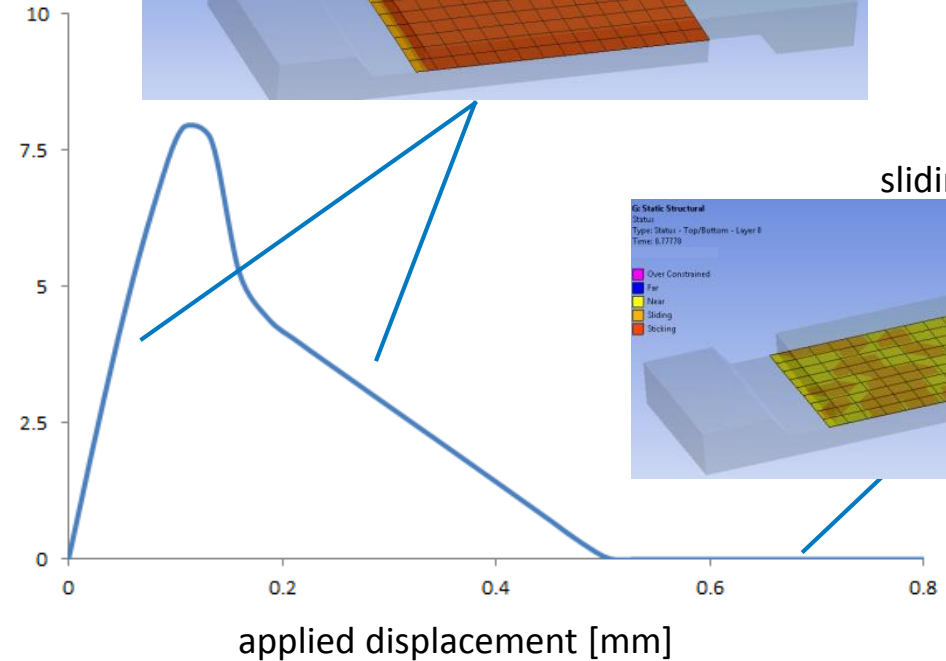
Deformed Shape (57X) at Peak



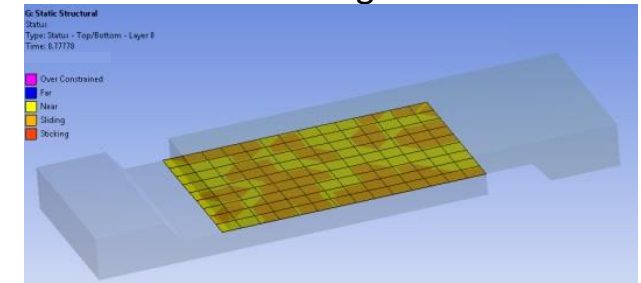
Nonlinear Bonding (sticking)



Reaction Force / Initial Bonded Area [MPa]



sliding



1. Investigate solution, contour plots using the contact tool, reaction force vs applied displacement diagram (compare to engineering data of CZM for mode II), deformed shape