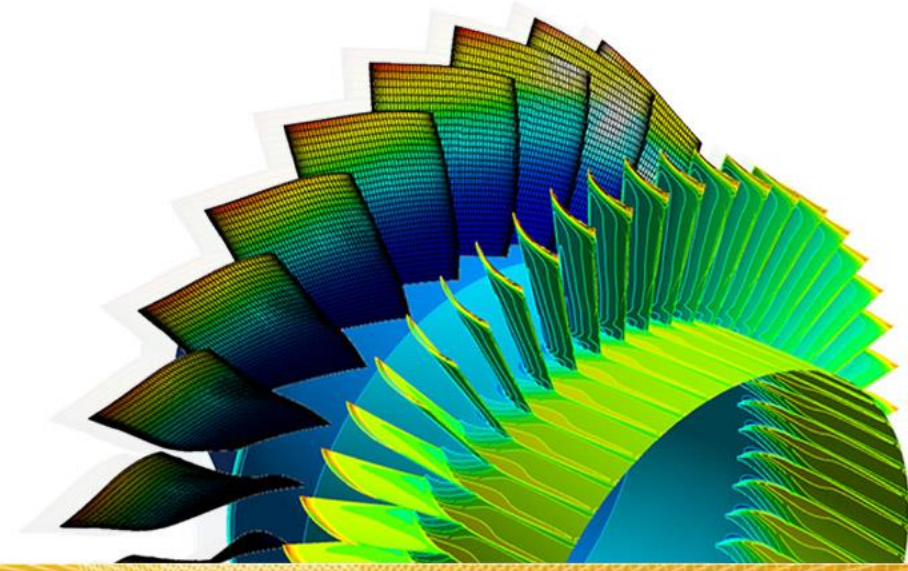
# ANSYS Composite PrepPost 19.0

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

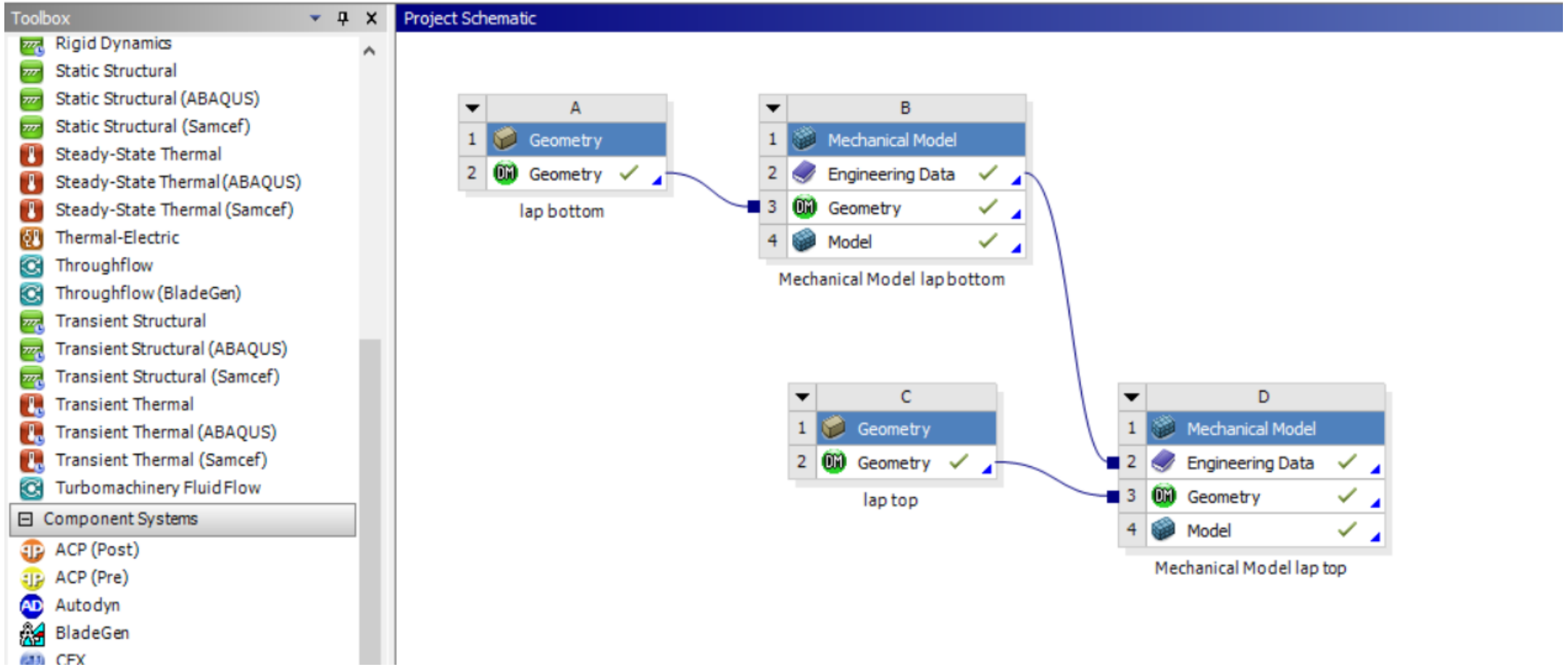
- A cohesive zone simulates the presence of a cured adhesive joining the lap


© 2018 ANSYS, Inc.

# 11. Workshop Single Lap

1. Start ANSYS Workbench and restore from archive

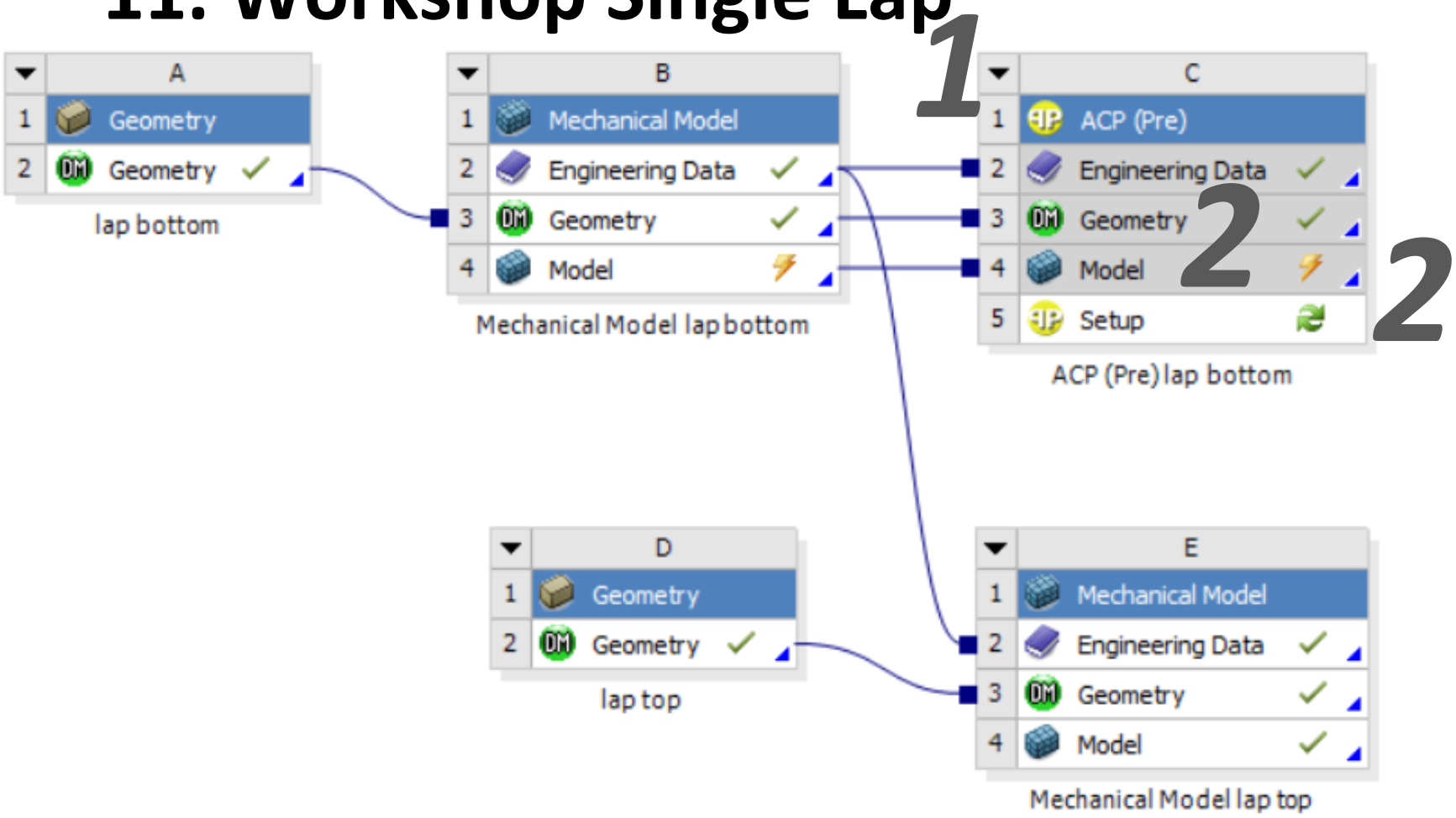
   *lap_joint_test_from_start_19.0.wbpz*





© 2018 ANSYS, Inc.

# 11. Workshop Single Lap

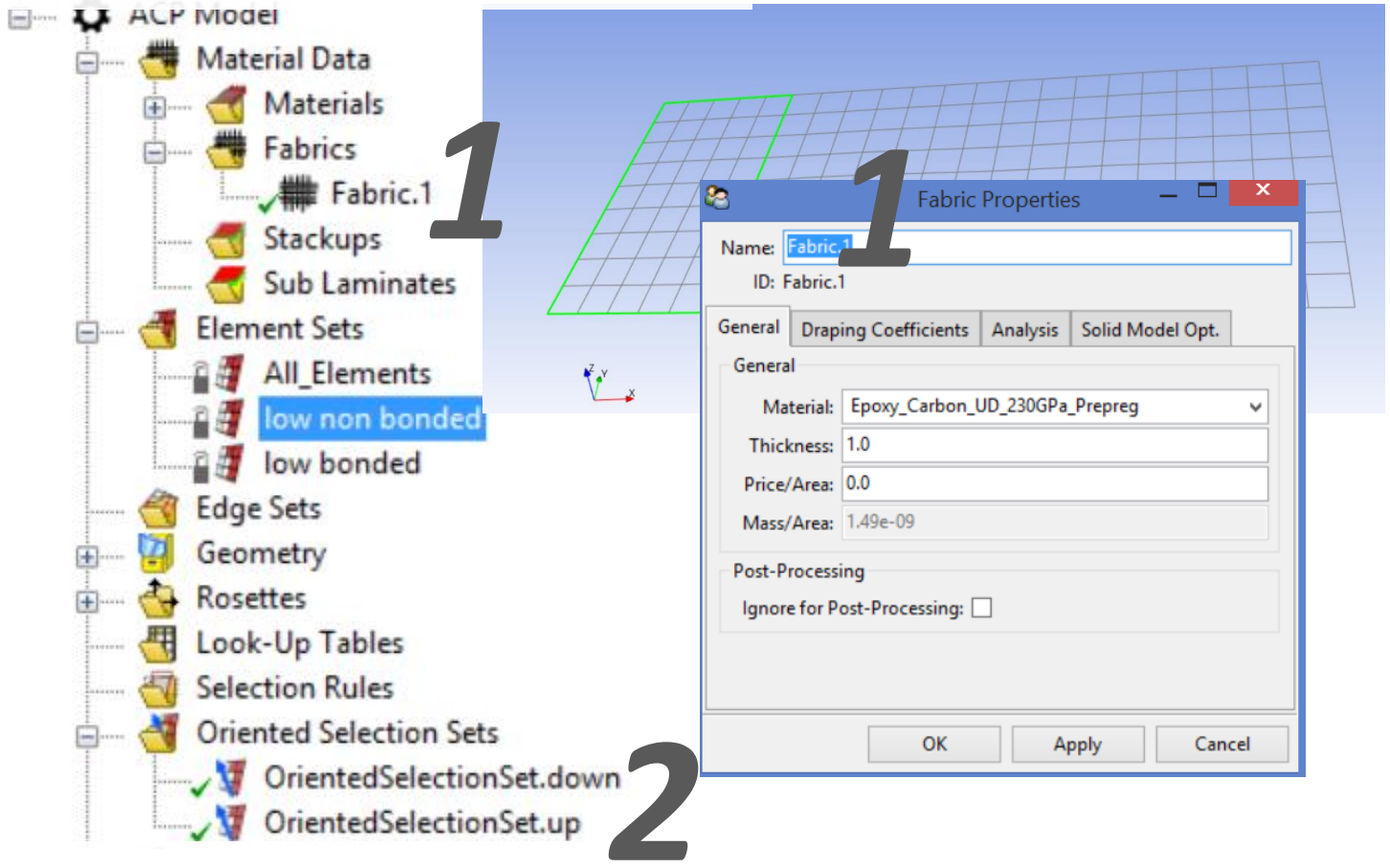



1. Drag and drop ACP (Pre) in the project schematic onto *Mechanical Model lap bottom*, rename it *ACP (Pre) lap bottom*
2. Update the project and double click on ACP (Pre) *Setup*


© 2018 ANSYS, Inc.

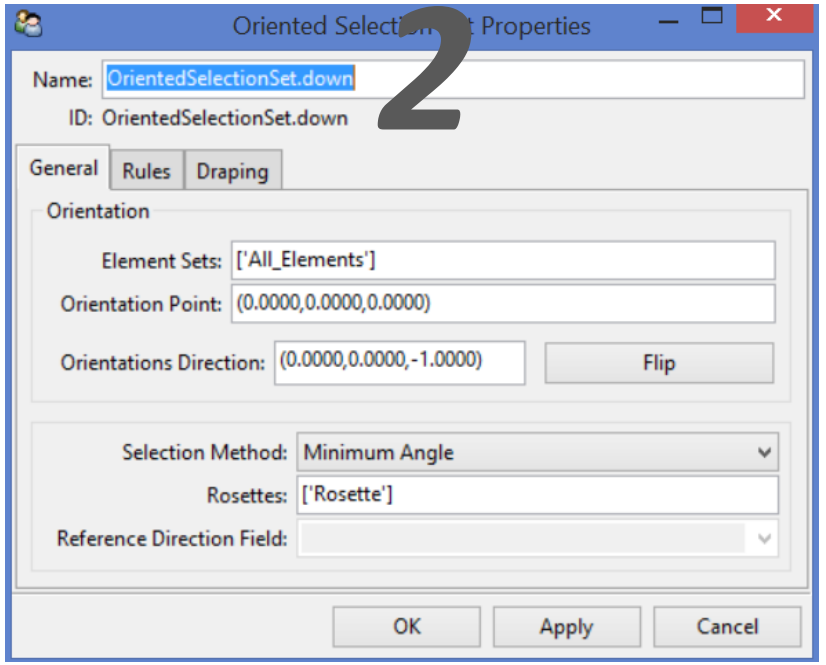
# 11. Workshop Single Lap

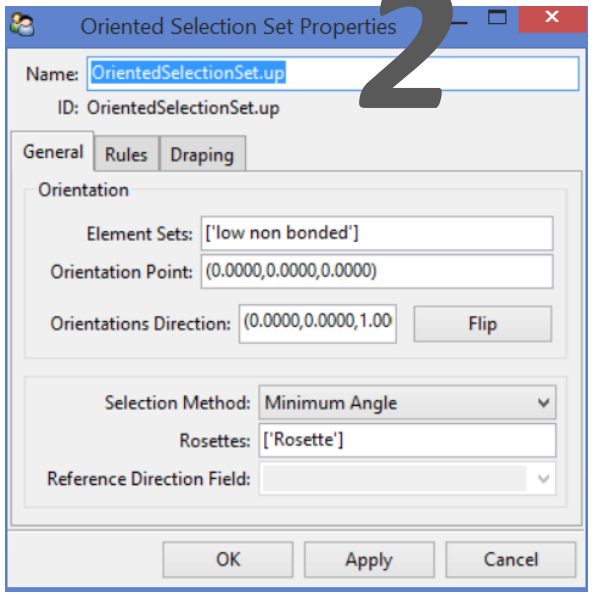
## ACP (Pre) lap bottom









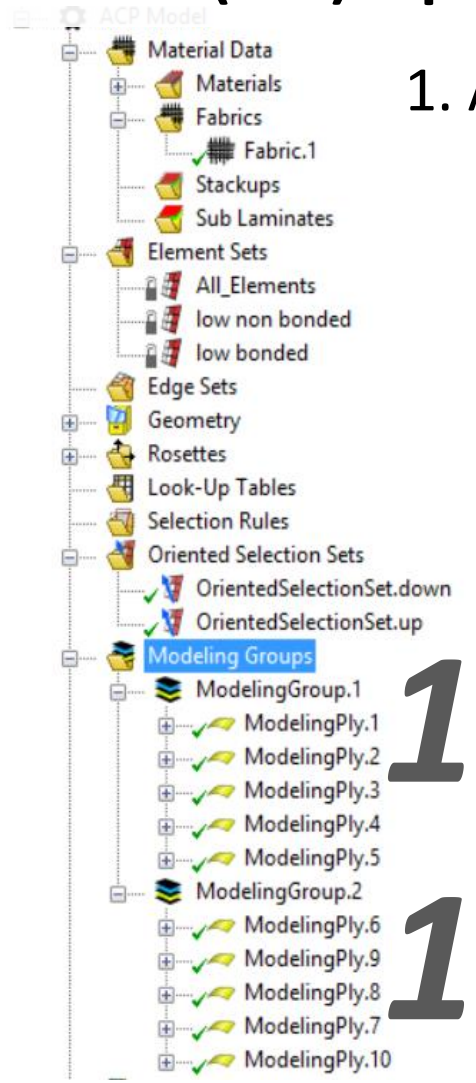




(1) Add a new fabric, (2) add two oriented selection sets for all the elements and for the element set which will not be bonded at a later stage of the workshop

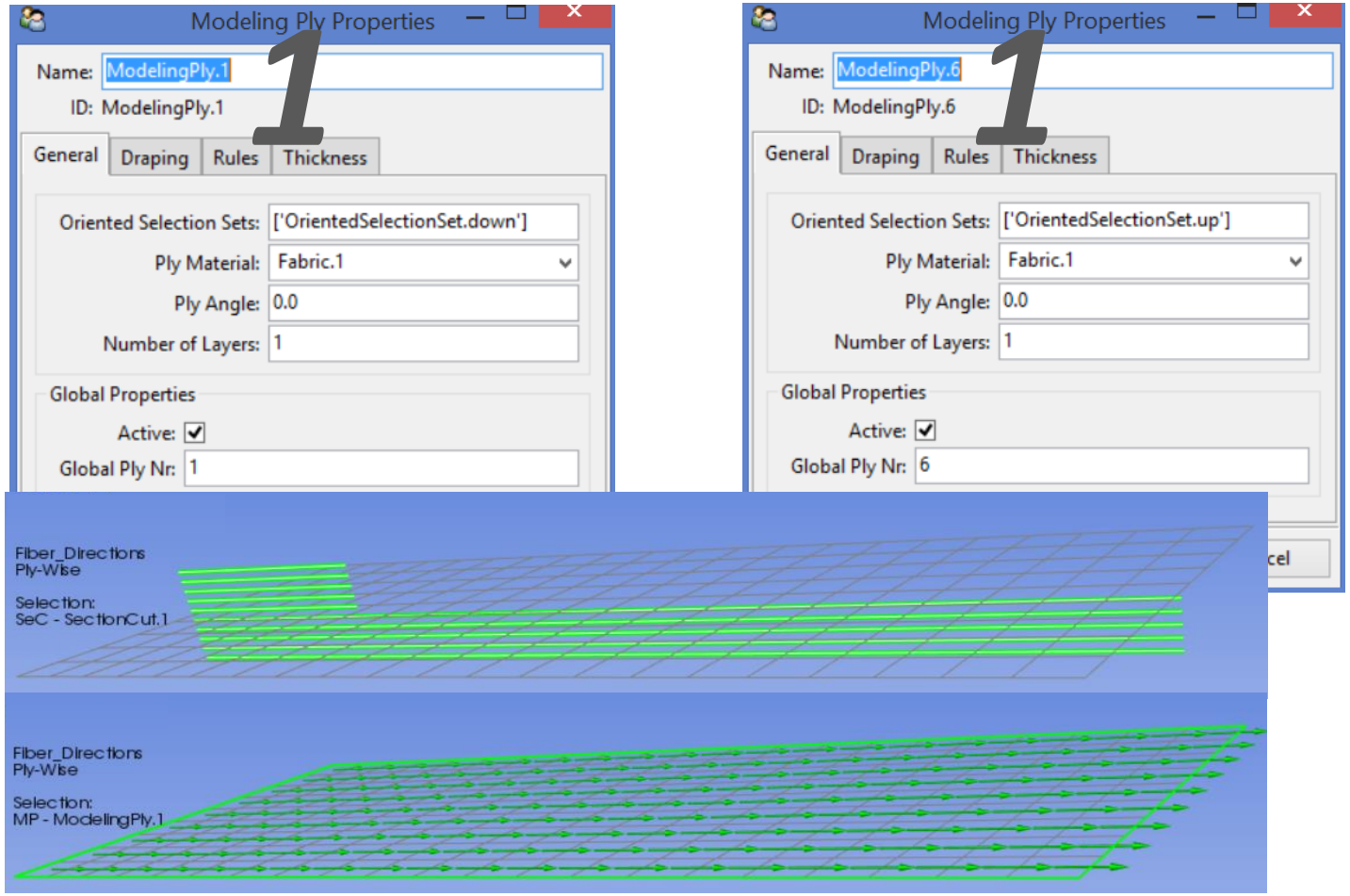

© 2018 ANSYS, Inc.

# 11. Workshop Single Lap

## ACP (Pre) lap bottom




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

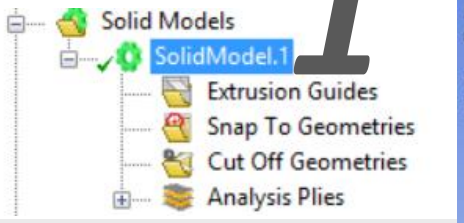




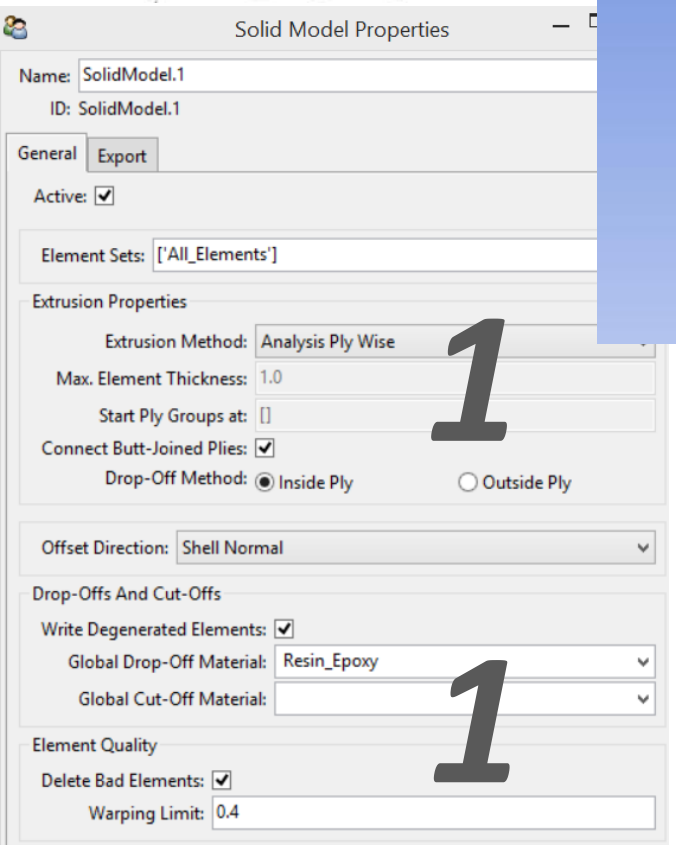
© 2018 ANSYS, Inc.

# 11. Workshop Single Lap

## ACP (Pre) lap bottom

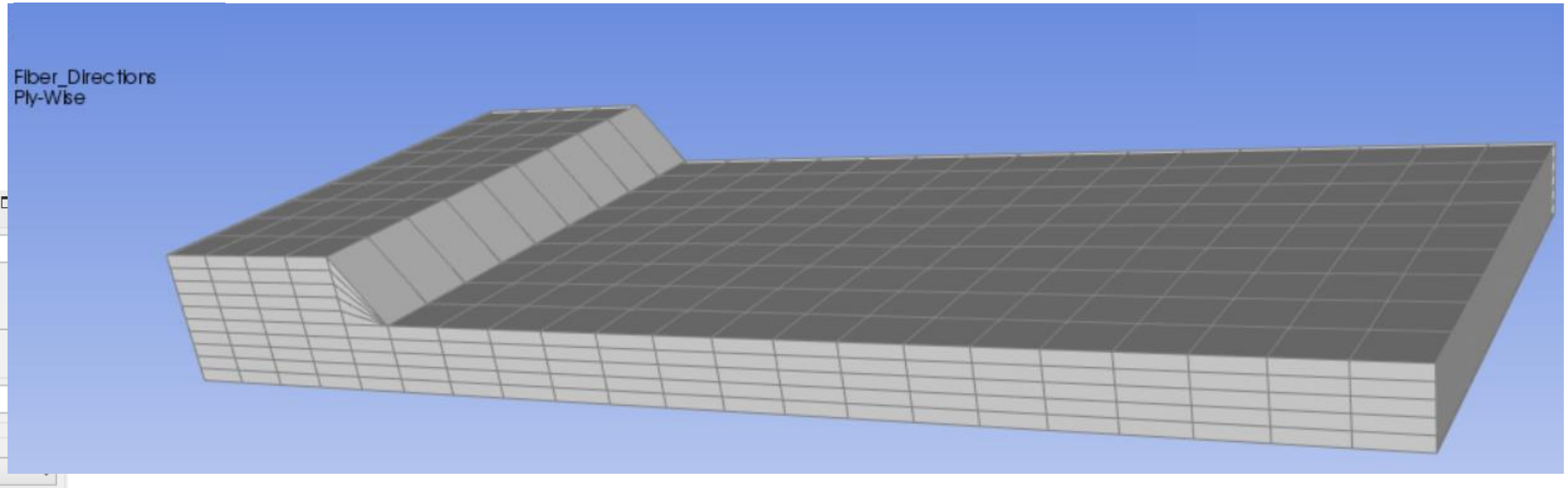



1




1

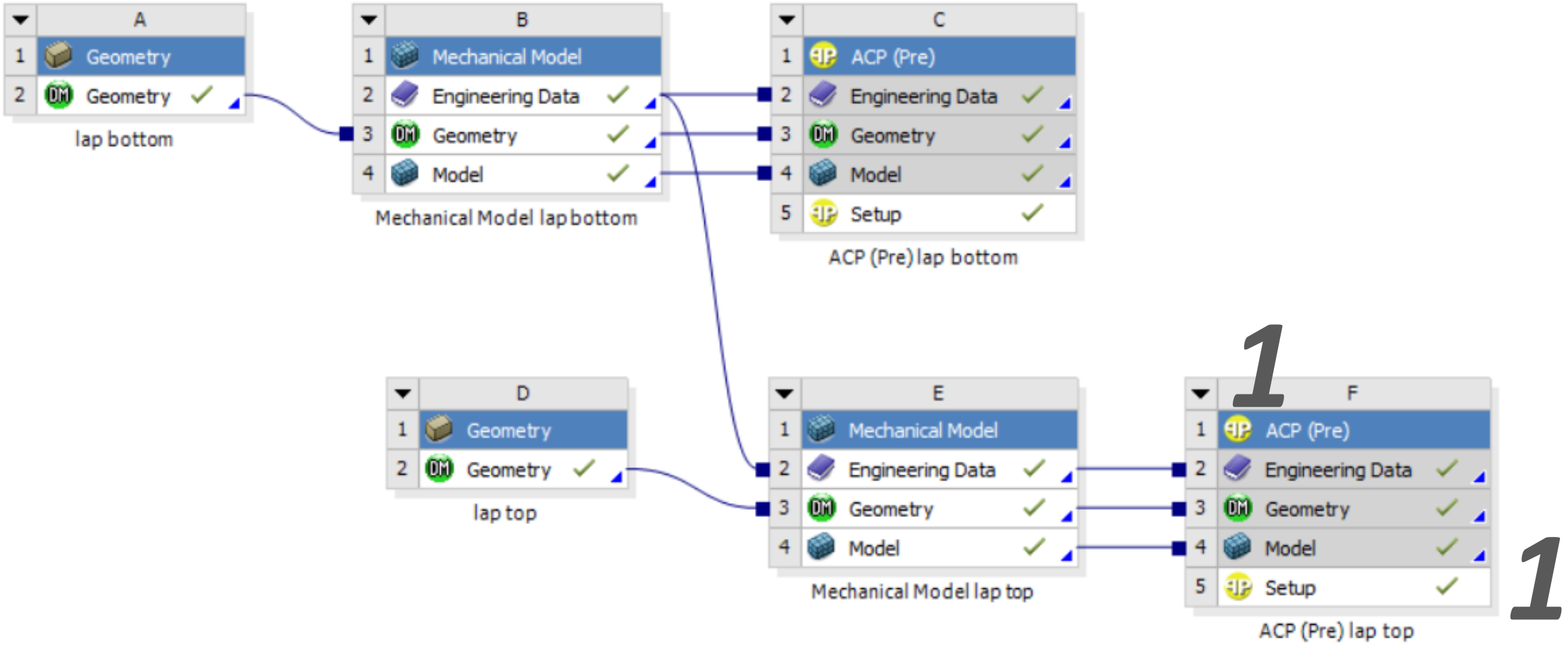
1




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


© 2018 ANSYS, Inc.

# 11. Workshop Single Lap

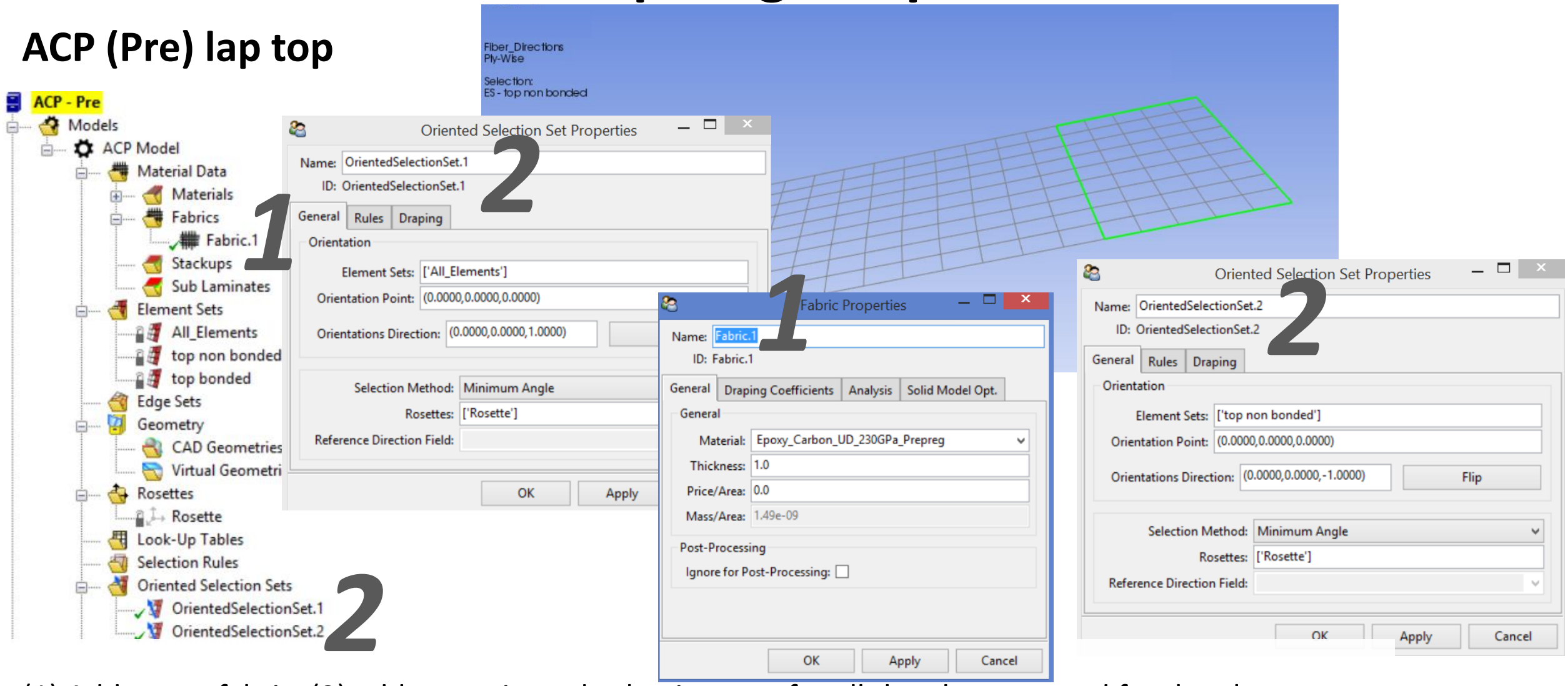



1. Drag and Drop another ACP (Pre) model onto the mechanical model for the top part creating the lap, rename it ACP (Pre) lap top and double click on *Setup*


© 2018 ANSYS, Inc.

# 11. Workshop Single Lap

## ACP (Pre) lap top

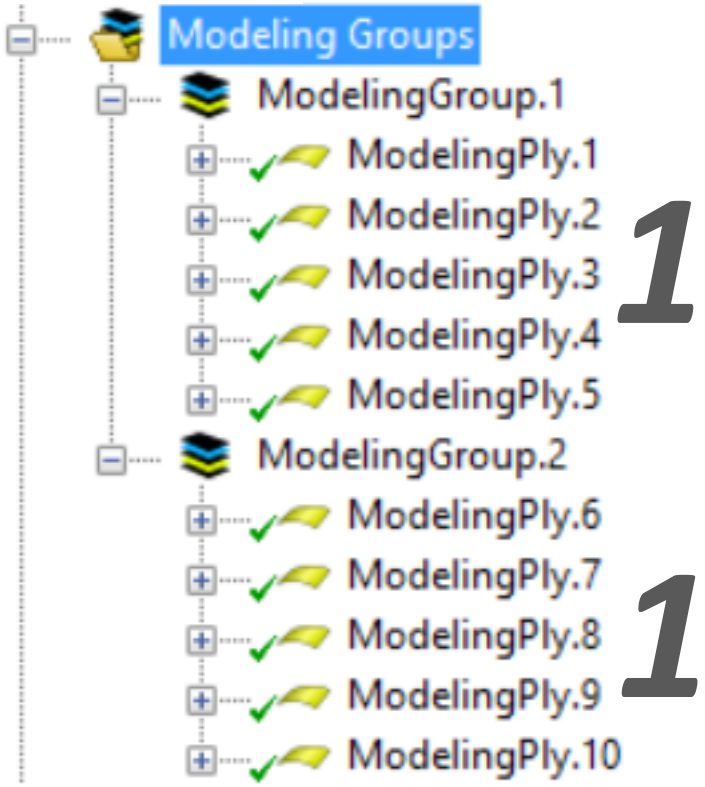


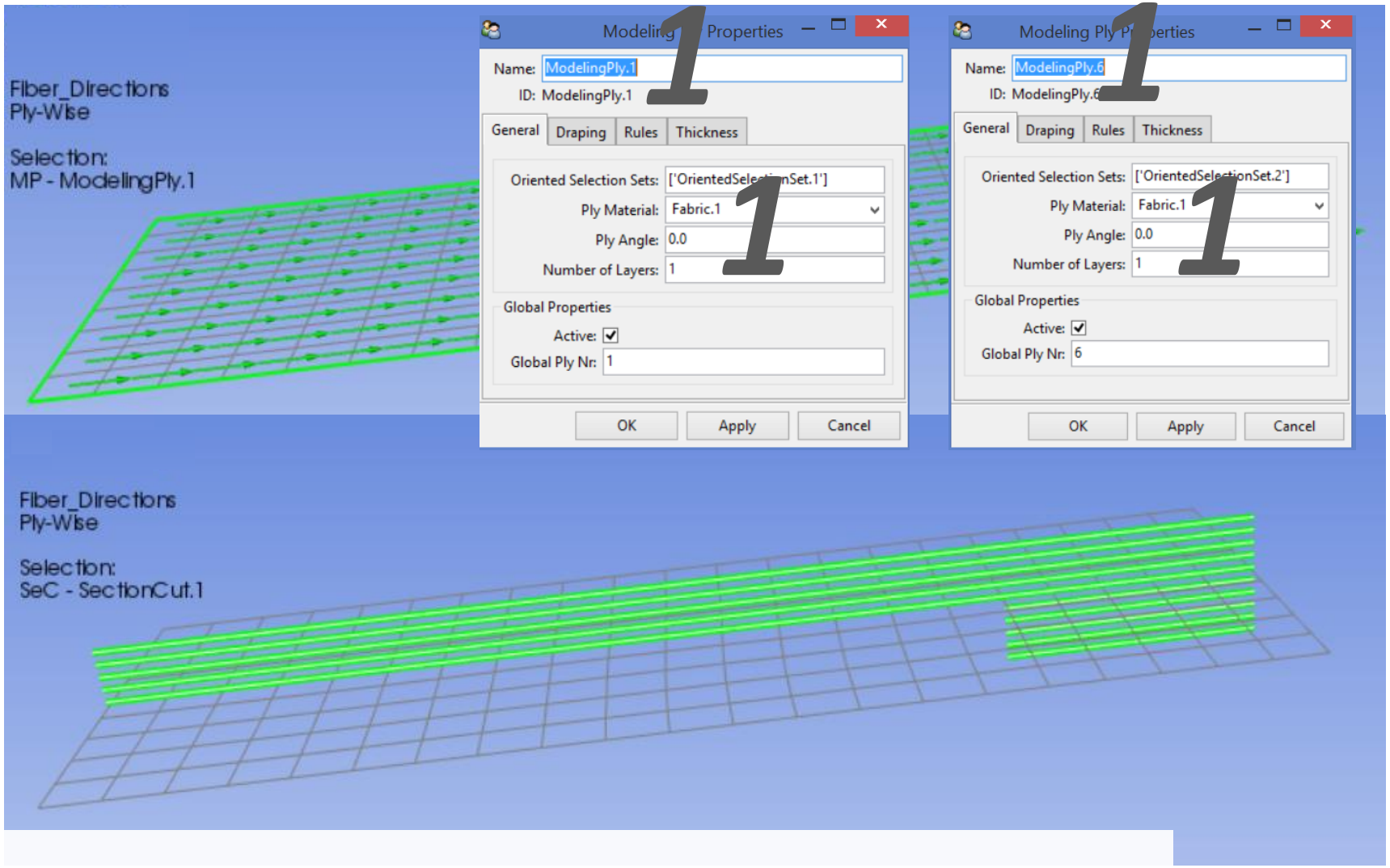

(1) Add a new fabric, (2) add two oriented selection sets for all the elements and for the element set which will not be bonded at a later stage of the workshop


© 2018 ANSYS, Inc.

# 11. Workshop Single Lap

## ACP (Pre) lap top



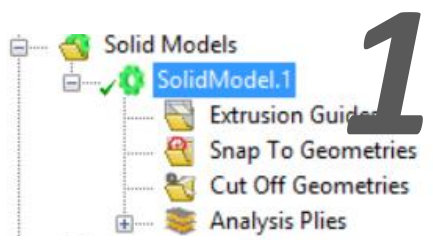



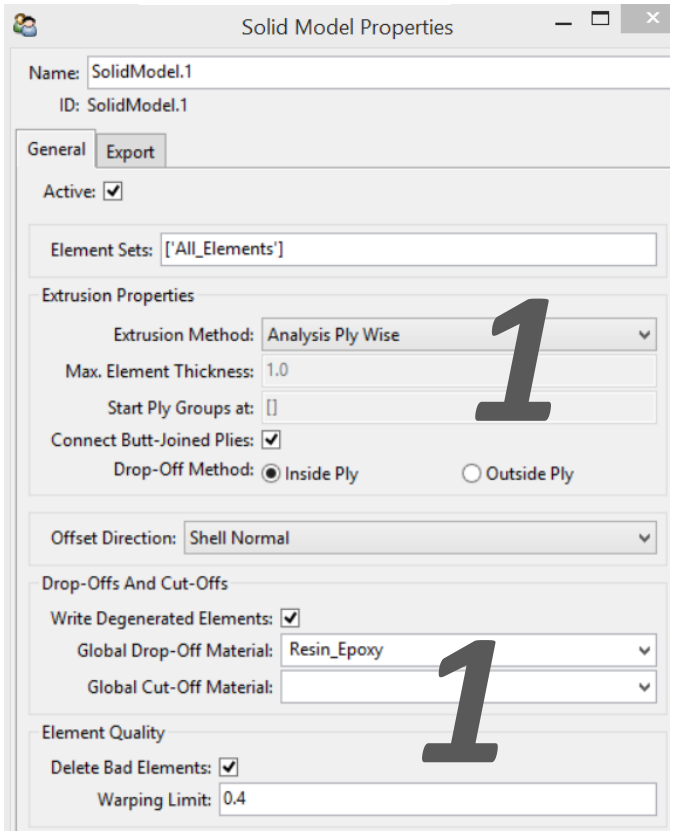

1. Add two *Modeling Groups* with 5 plies each, all oriented in the global x direction

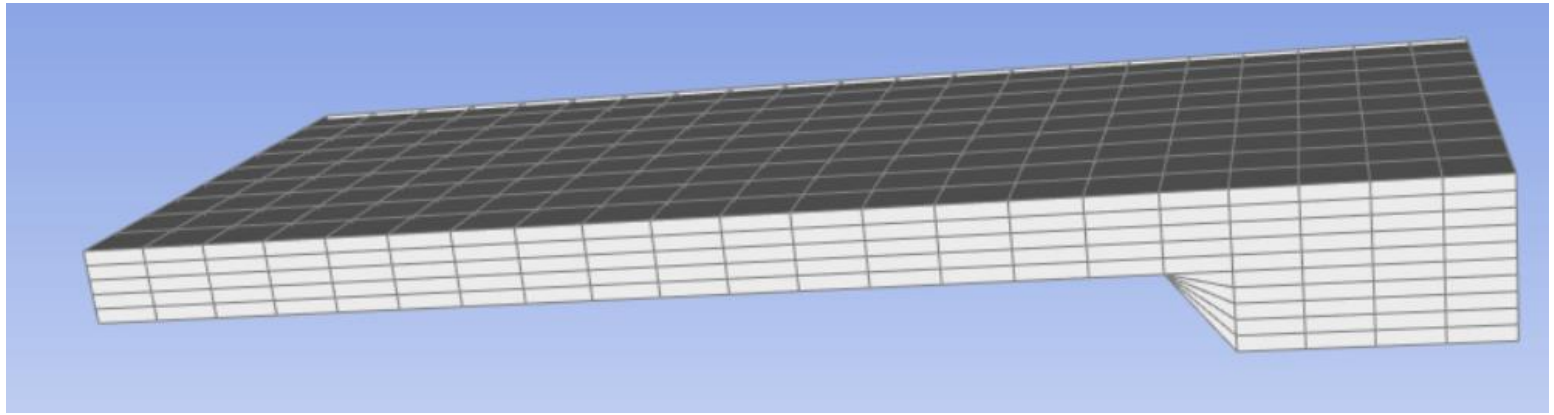

© 2018 ANSYS, Inc.

# 11. Workshop Single Lap

## ACP (Pre) lap top



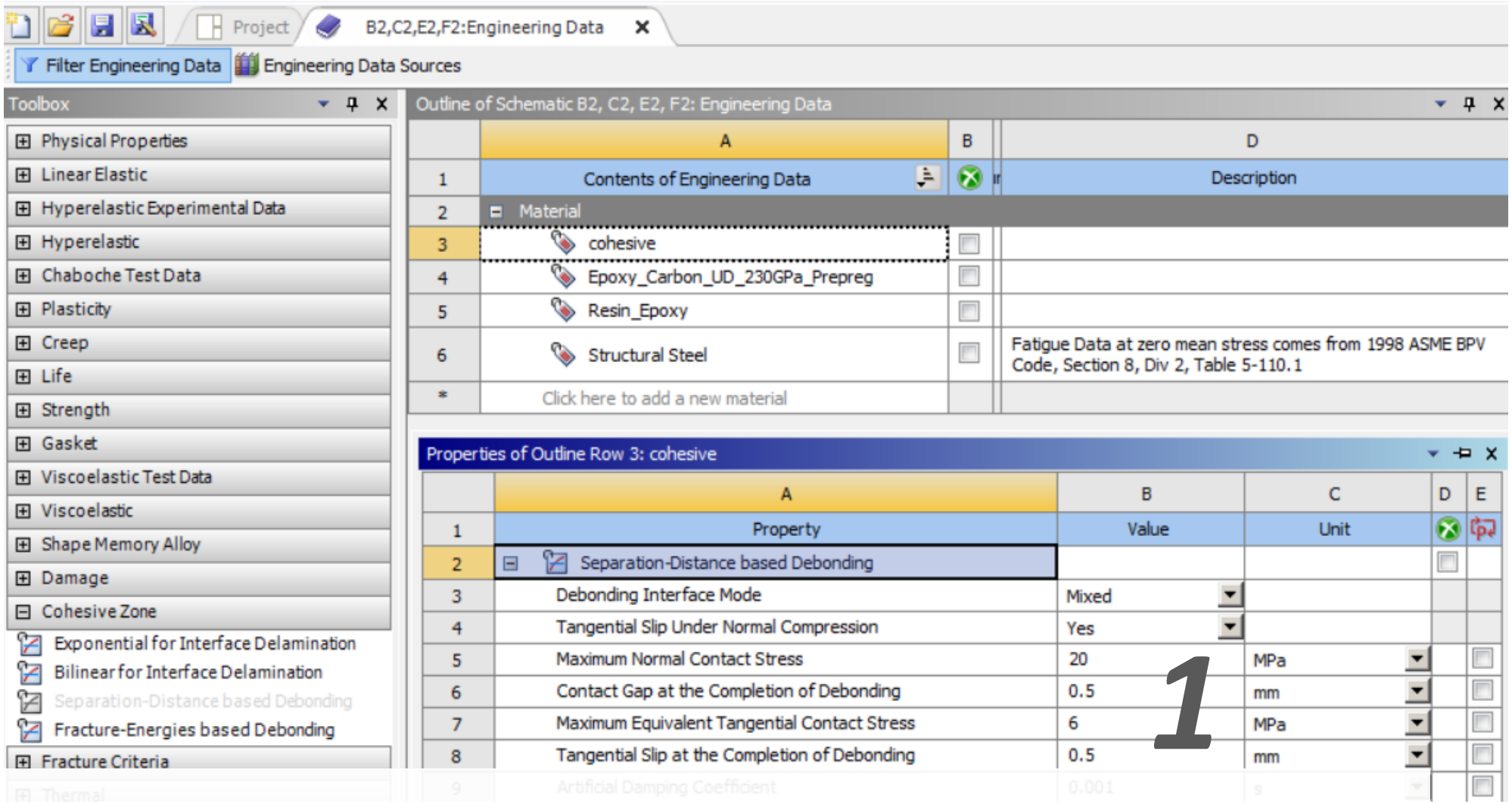



1. Extrude a solid model, Analysis Ply Wise and Resin_Epoxy as global drop-off material
2. Update the project and leave ACP (Pre)


© 2018 ANSYS, Inc.

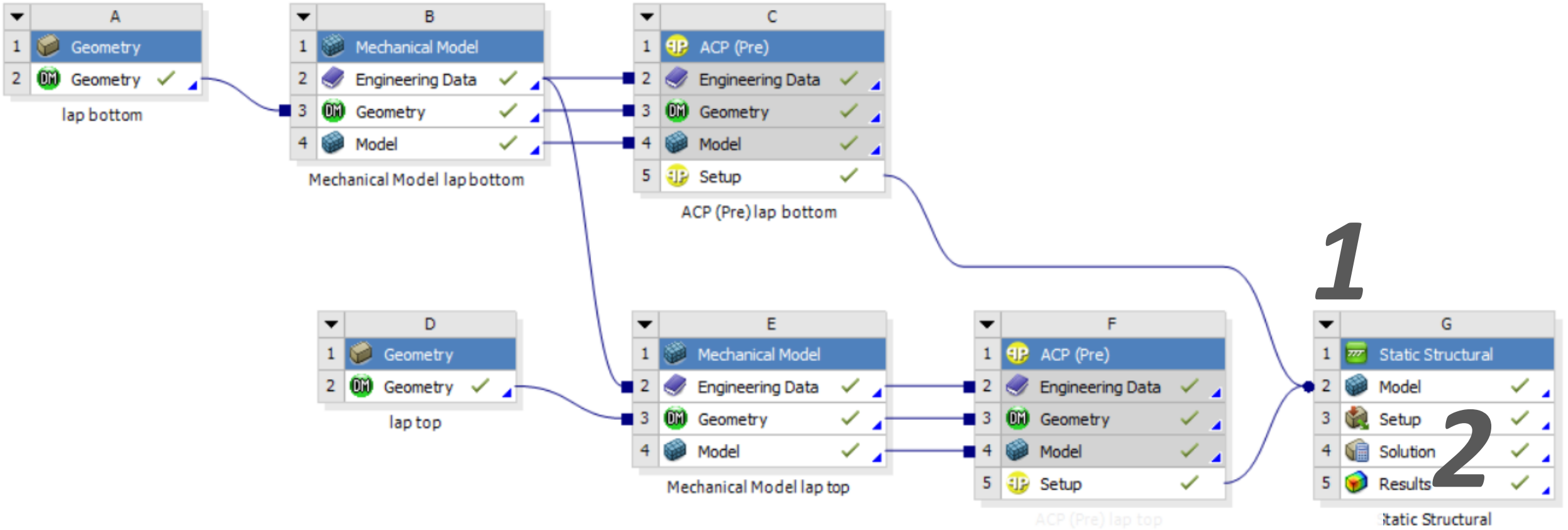
# 11. Workshop Single Lap




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


© 2018 ANSYS, Inc.

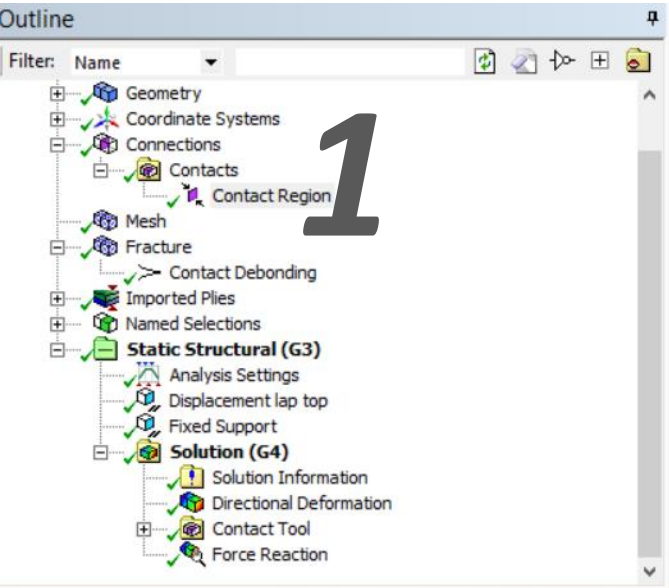
# 11. Workshop Single Lap

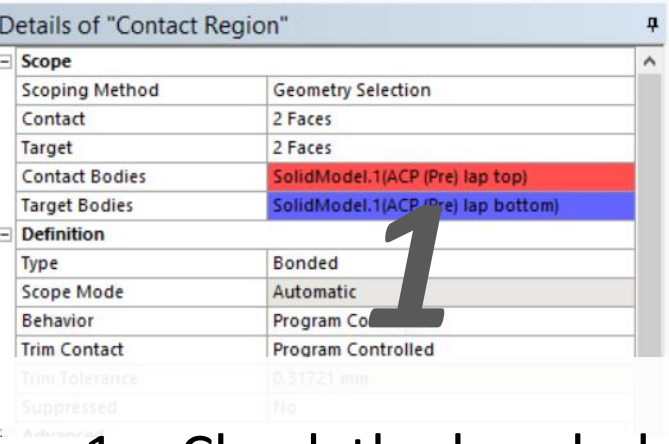


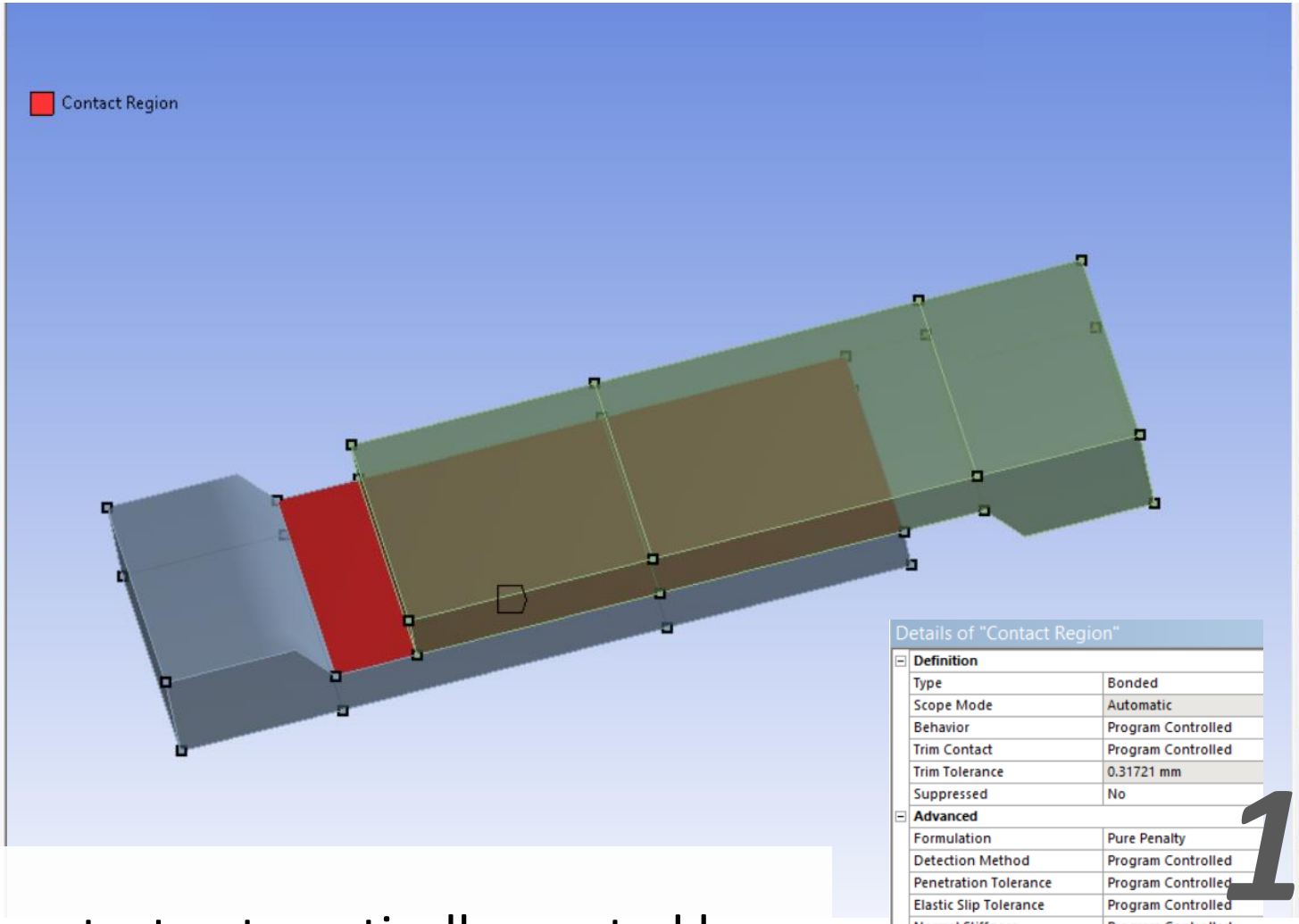

1. Add a *Static Structural* block in the project schematic and connect its cell *Model* to the *Setup* cell of the *ACP (Pre)* blocks modeling the two parts of the lap joint (transfer composite solid data)
2. Update the project and open *Mechanical* by double clicking the cell *Solution*

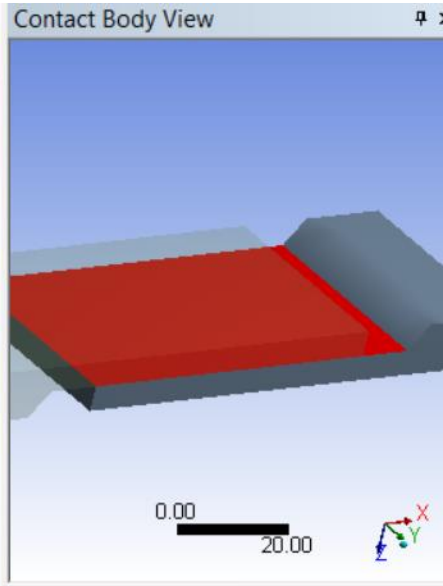

© 2018 ANSYS, Inc.

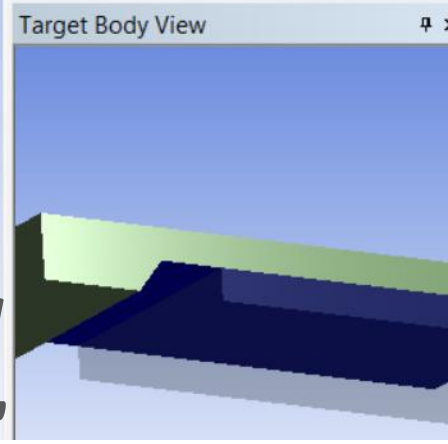
# 11. Workshop Single Lap



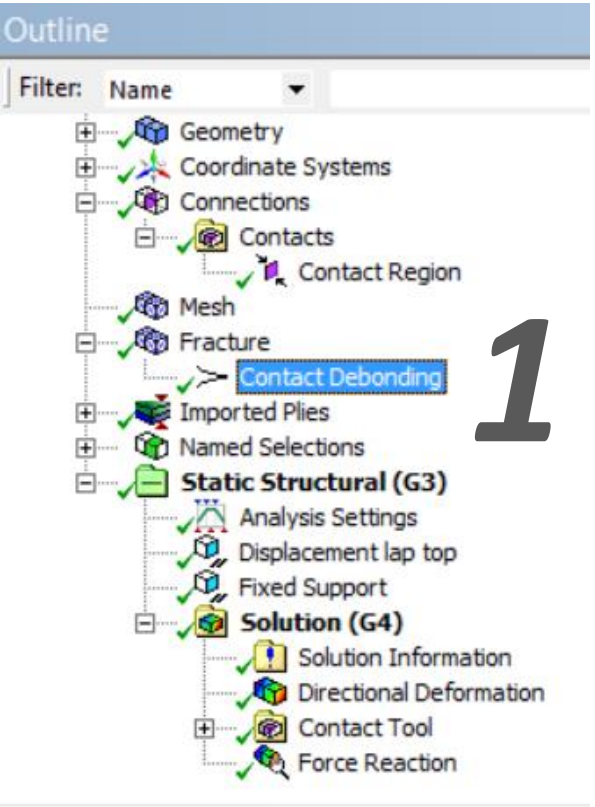


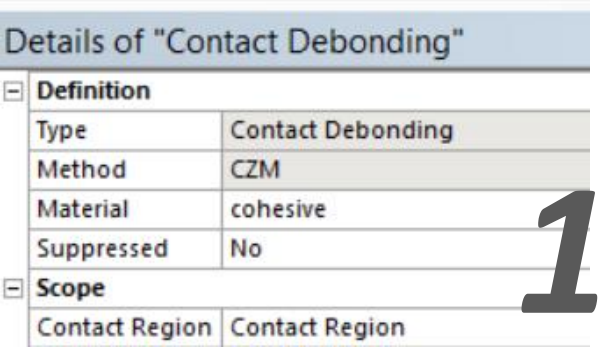


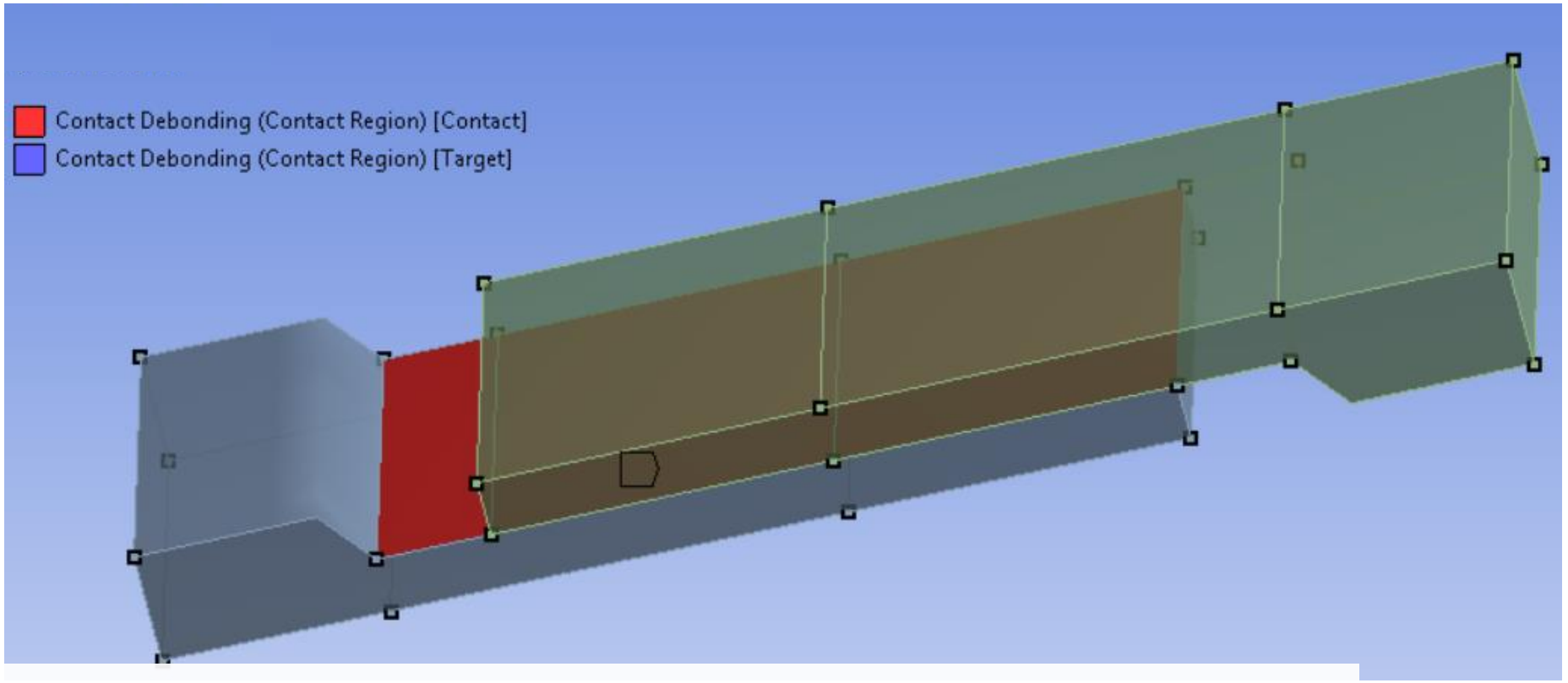
1. Check the bonded contact automatically created by Workbench, **modify the formulation to Pure Penalty**


© 2018 ANSYS, Inc.

# 11. Workshop Single Lap



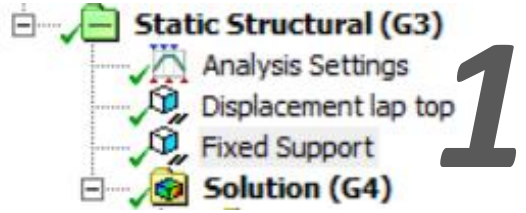




1. Insert *Contact-Debonding* in Fracture, select CZM method, the cohesive material defined in *Engineering Data*, and the *Contact Region* defined before by Workbench


© 2018 ANSYS, Inc.

# 11. Workshop Single Lap

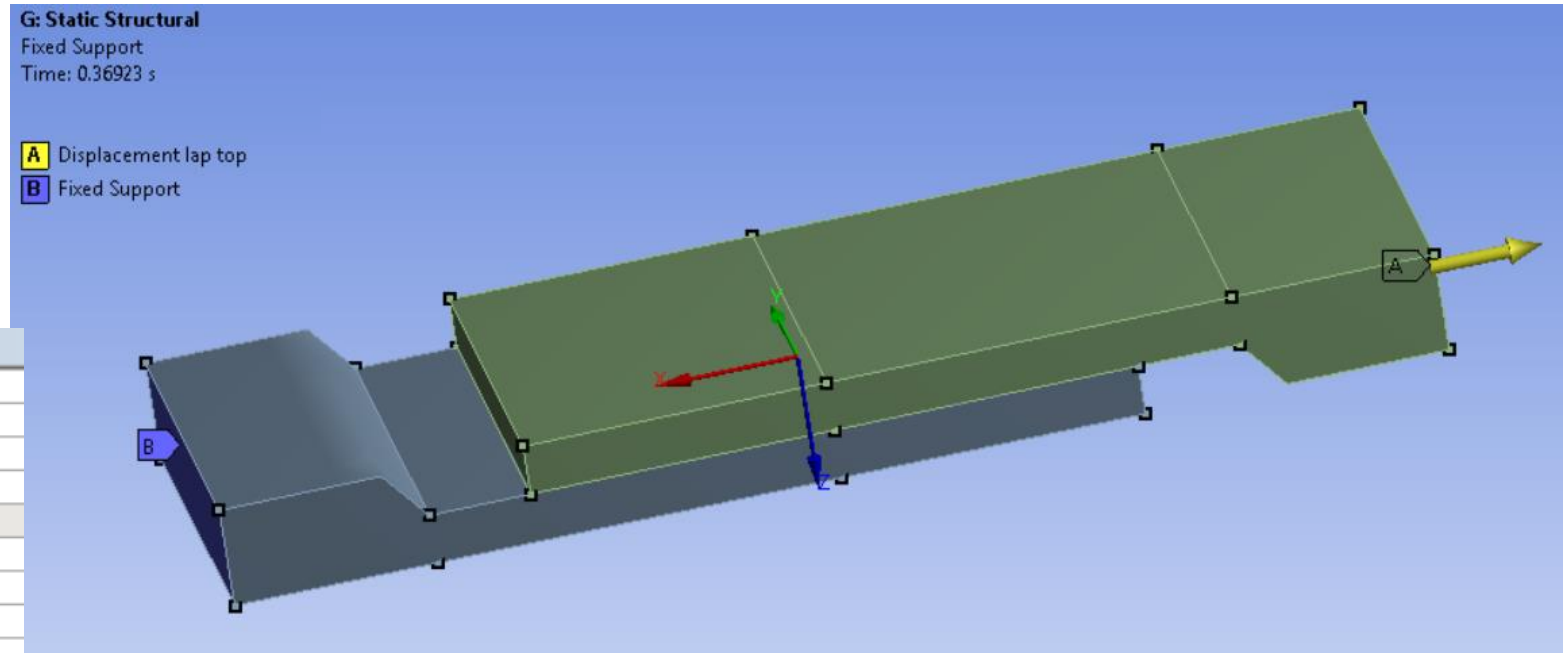



Details of "Fixed Support" 1

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

Details of "Displacement lap top" 1

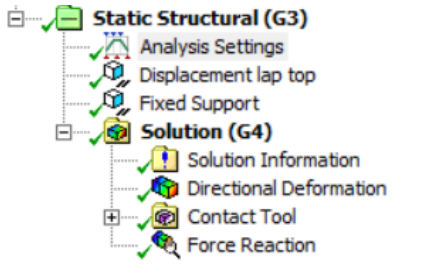
| | |
|---|---|
| **Scope** | |
| Scoping Method | Geometry Selection |
| Geometry | 1 Face |
| **Definition** | |
| Type | Displacement |
| Define By | Components |
| Coordinate System | Global Coordinate System |
| X Component | -0.8 mm (ramped) |
| Y Component | 0. mm (ramped) |
| Z Component | 0. mm (ramped) |
| Suppressed | No |




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


© 2018 ANSYS, Inc.

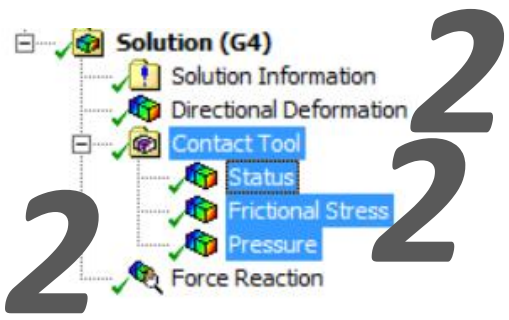
# 11. Workshop Single Lap




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 |
| Display Time | 0.25 s |
| Coordinate System | Global Coordinate System |
| Calculate Time History | Yes |
| Identifier | |
| Suppressed | No |
| **Results** | |
| Minimum | -0.20257 mm |
| 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** | |
| Minimum | -0.8 mm |

Details of "Force Reaction"

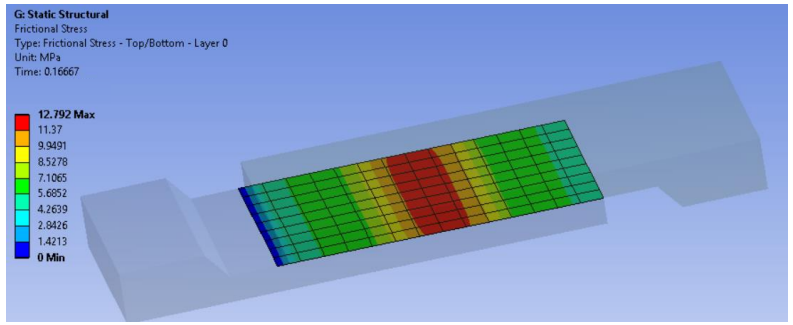
| | |
|---|---|
| **Definition** | |
| Type | Force Reaction |
| Location Method | Boundary Condition |
| Boundary Condition | Displacement lap top |
| Orientation | Global Coordinate System |
| Suppressed | No |
| **Options** | |
| Result Selection | All |
| Display Time | 0.35 s |
| **Results** | |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

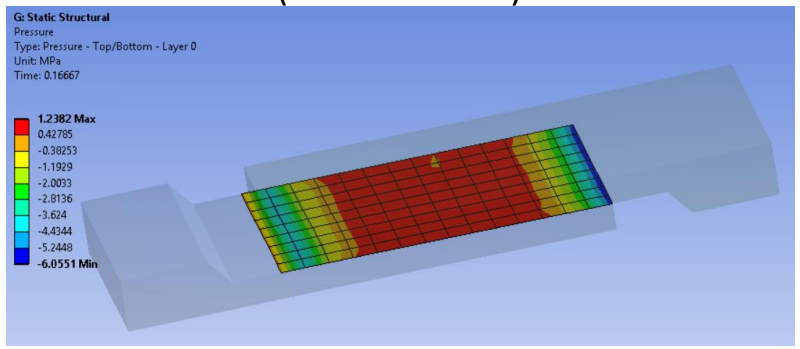

© 2018 ANSYS, Inc.

# 11. Workshop Single Lap

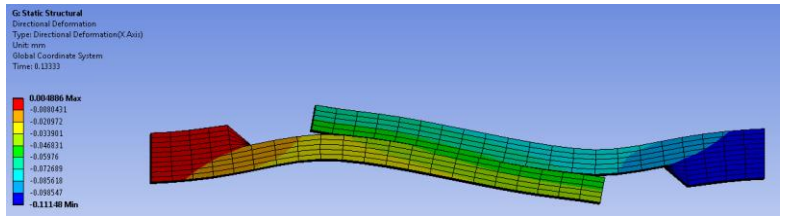
Frictional (Shear) Stress at Peak



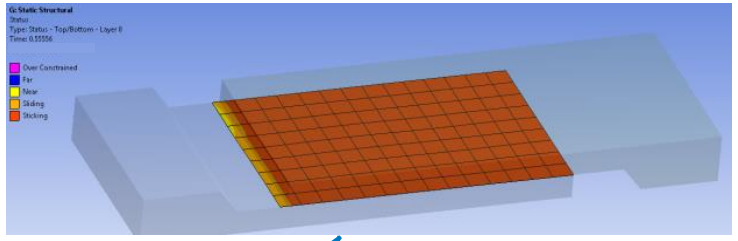

Pressure (Normal Stress) at Peak

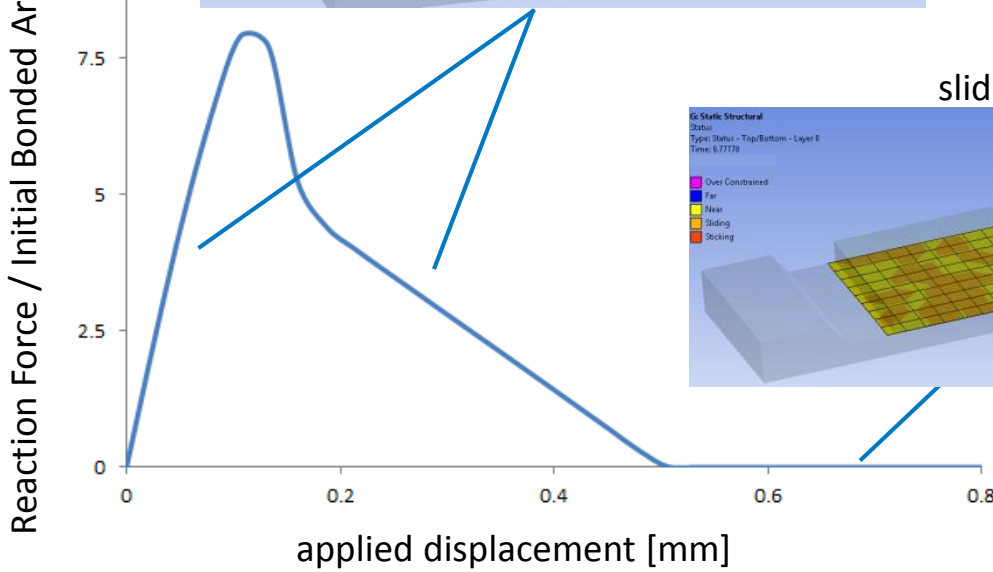



Deformed Shape (57X) at Peak

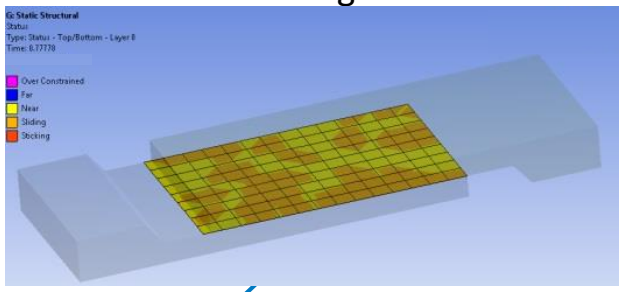


Nonlinear Bonding (sticking)






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


© 2018 ANSYS, Inc.