Workshop 03: Tool Compensation

Release 2021 R1

Workshop 03: Tool Compensation

Workshop Overview

- In this workshop you will first learn the usage of ANSYS Composite Cure Simulation to capture the thermal-chemical reactions during composite manufacturing, to optimize the curing process (responsible for an exothermic peak) and how to design the heating and cooling system

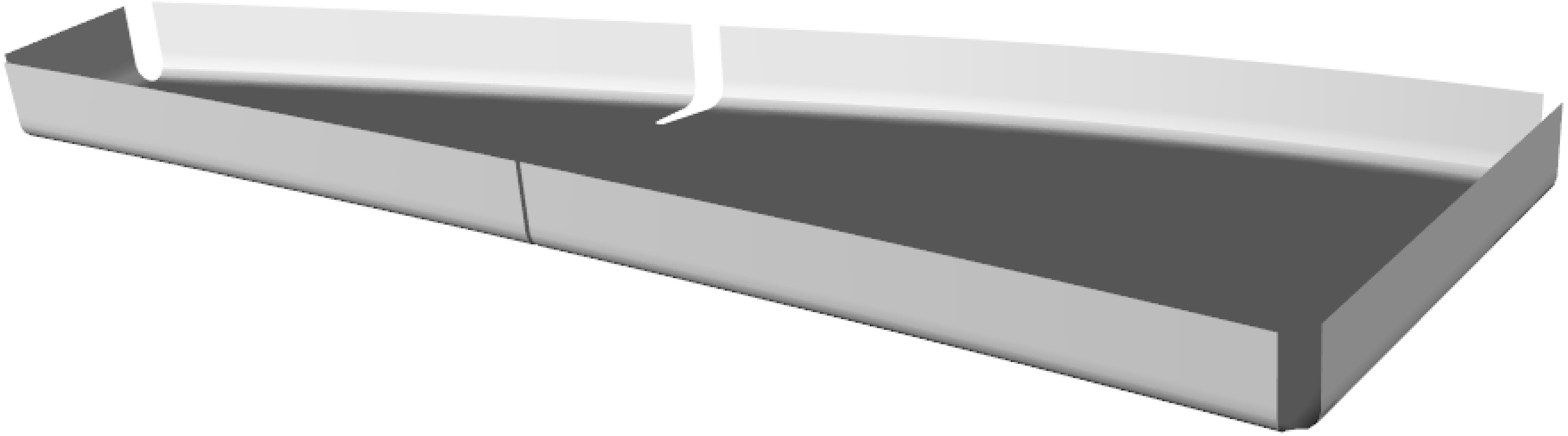
- You will predict the development of residual stresses and process induced distortions.

- You will understand how to compensate the tooling geometry to meet the required geometrical tolerances, resulting in an assembly process with the minimal built-in stresses

Workshop 03: Tool Compensation

Workshop Overview

- A composite frame will be used in this workshop to illustrate the usage of ACCS to predict the curing process and the methodology to compensate for associated process induced distortions



Overview to Process Induced Distortions

LMAT

Ansys

Workshop 03: Tool Compensation

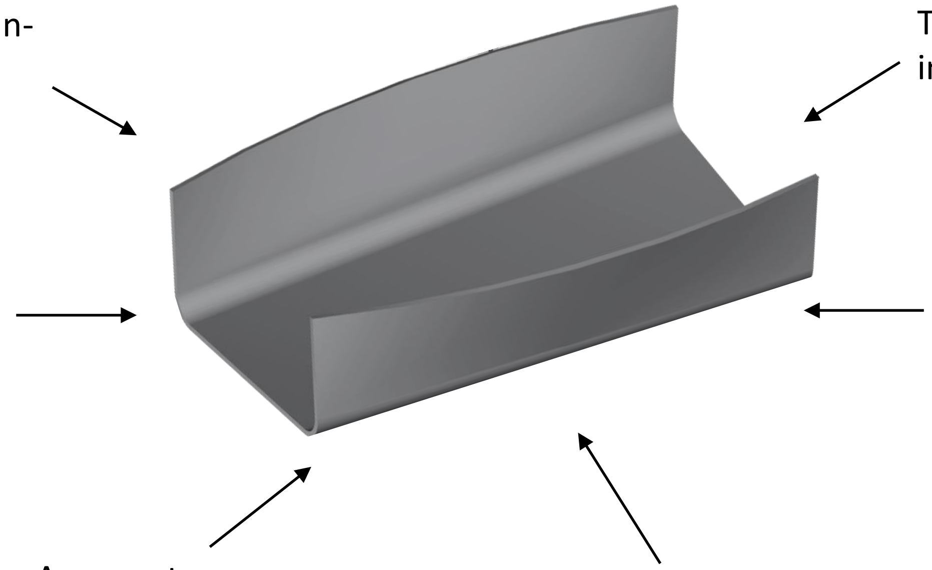
Process Induced Distortions Overview

- Process-induced distortions are a major concern when manufacturing with composite materials

Workshop 03: Tool Compensation

Causes of Process Induced Distortions and Failures

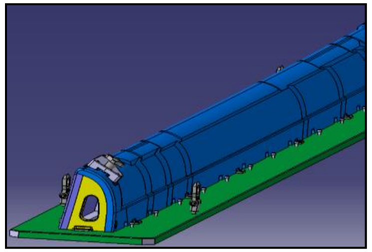
- Residual stresses that develop during cure result in distorted composite components



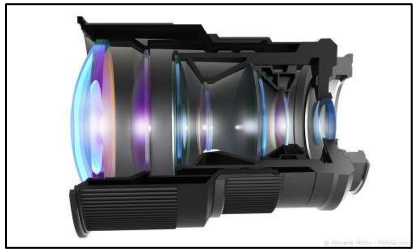
Workshop 03: Tool Compensation

Impact of the PID in the industry

- The traditional trial-and-error approach can work for simple geometries, but composite parts with complex shapes require more sophisticated models.



Aerospace: certification requires very tight tolerances. Subsequent corrections of the tooling to compensate PID are very expensive and can turn a profitable project into a deficit one.



Imaging: the relative alignment of the lenses is negatively affected by PID of the casing what reduces the accuracy of high performance optical systems.

Wind energy: the join between the blade and generator is very sensitive in terms of PID. An accurate prediction of the PID based on simulation saves time and money.

Automotive industry: initial crack due to residual stresses, for instance in ignition coils, is a phenomena that can cause high waste and reduces the lifetime of a product.

Workshop 03: Tool Compensation

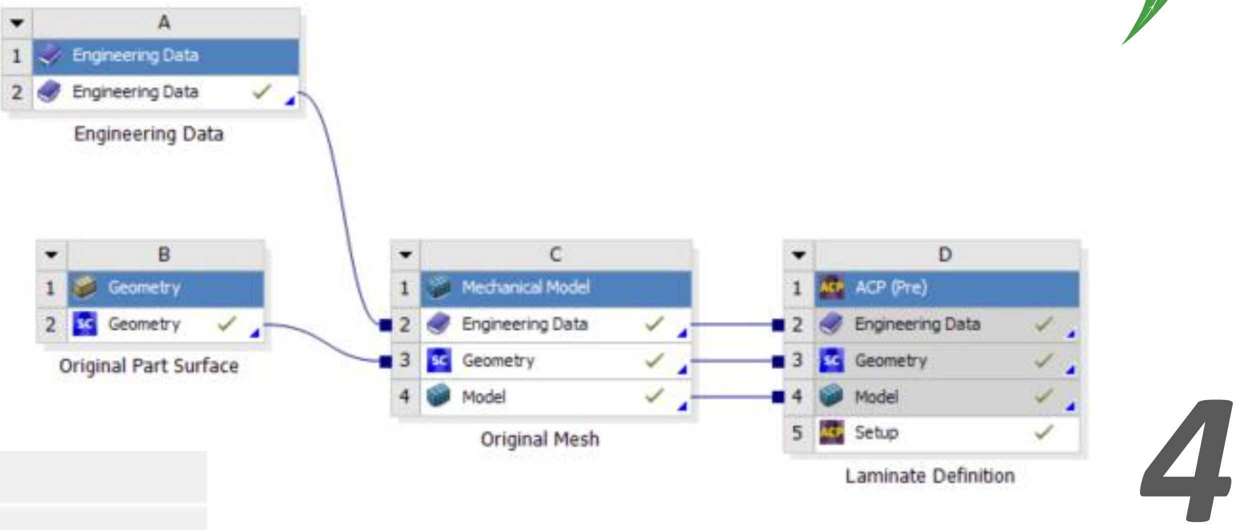
Compensation of PID

- Prediction of Process Induced Distortions after a curing process allows us to design a compensated geometry for our part to be closer to the nominal desired shape

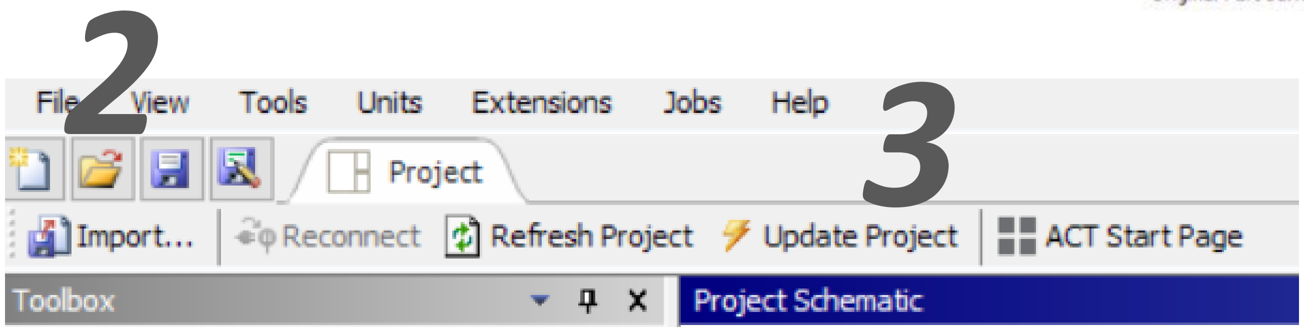
Workshop

Generating the Solid Model in ACP

Workshop 03: Tool Compensation

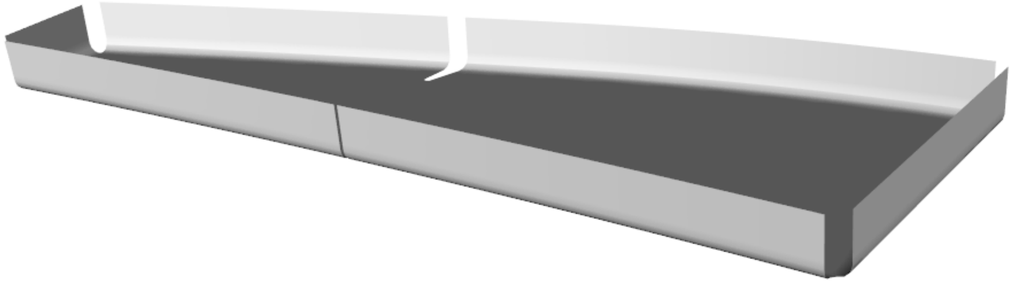


1. Start ANSYS Workbench



2. Open the archive
   `ACCS _<Release>_WS03_Rib_with_Tool_Compensation.wbpz`
3. Update the Workbench project
4. Edit Setup of the ACP Pre system to open ACP

Workshop 03: Tool Compensation

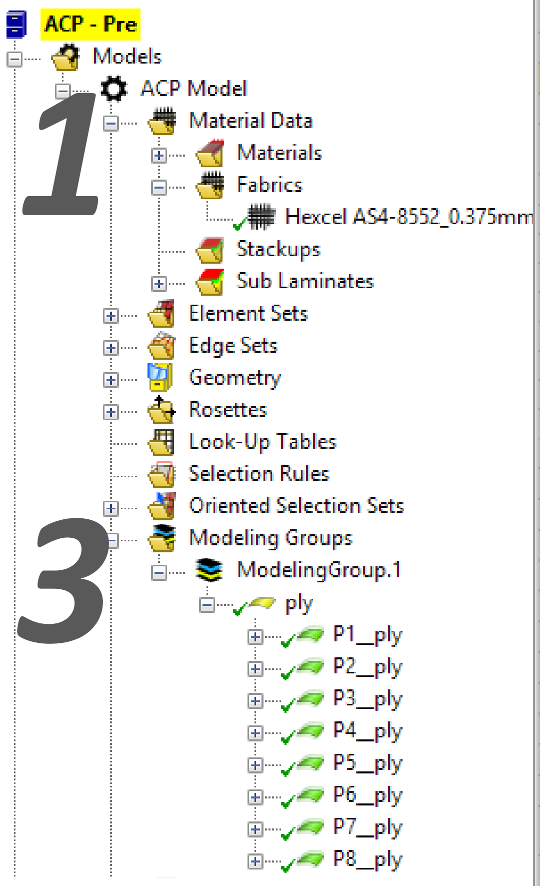


Composite Laminate in ACP

2

1

3

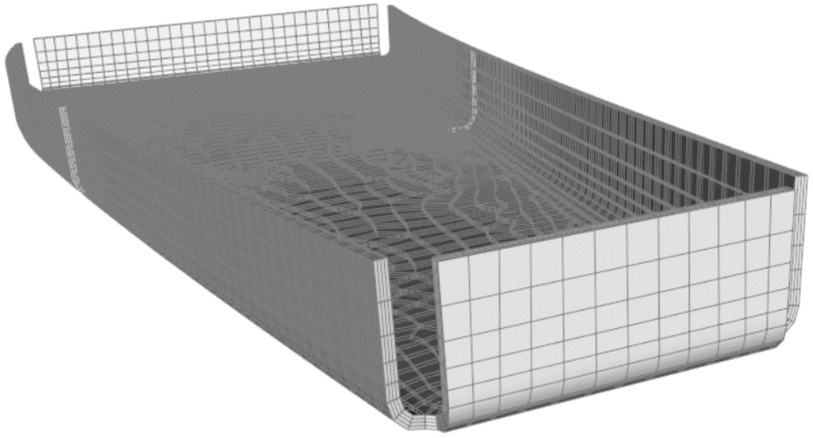


Properties of Outline Row 3: Hexcel AS4-8552 2

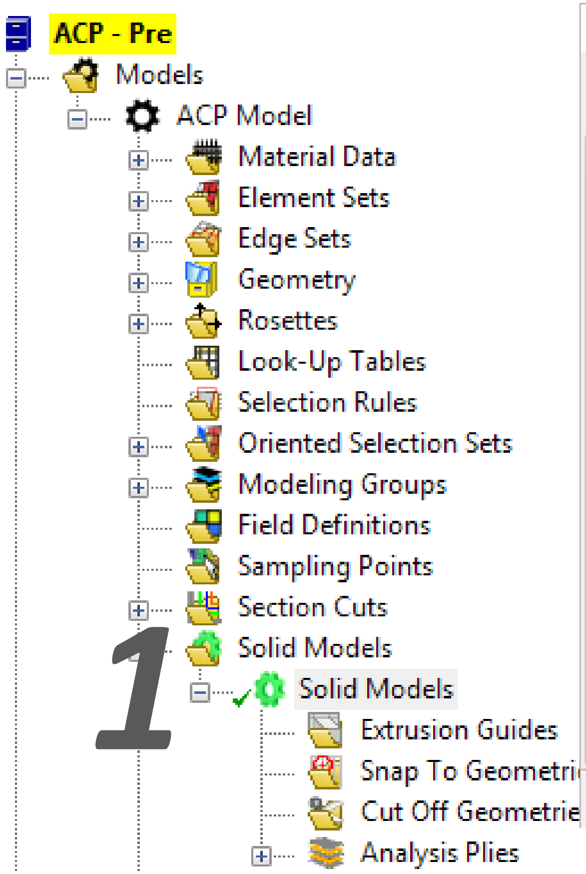
	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	1580	kg m^-3
4	Orthotropic Instantaneous Coefficient of Thermal Expansion		
8	Orthotropic Elasticity		
9	Young's Modulus X direction	1.35E+11	Pa
10	Young's Modulus Y direction	9.5E+09	Pa
11	Young's Modulus Z direction	9.5E+09	Pa
12	Poisson's Ratio XY	0.3	
13	Poisson's Ratio YZ	0.45	
14	Poisson's Ratio XZ	0.3	
15	Shear Modulus XY	4.9E+09	Pa
16	Shear Modulus YZ	3.27E+09	Pa
17	Shear Modulus XZ	4.9E+09	Pa
18	Orthotropic Thermal Conductivity		
22	Specific Heat, C_p	1300	J kg^-1 C...
23	Autocatalytic Cure Kinetic Equation		
28	Diffusion Limitation		
32	Resin Properties		
37	Total Heat of Reaction	5.4E+05	J
38	Glass Transition Temperature		
42	Orthotropic Cure Shrinkage	Tabular	
43	Material Properties Evolution		
46	Orthotropic Liquid Pseudo Elasticity	Tabular	
47	Orthotropic Rubbery Elasticity	Tabular	
48	Orthotropic Instantaneous Liquid Coefficient of Thermal Expansion		
52	Orthotropic Instantaneous Rubbery Coefficient of Thermal Expansion		

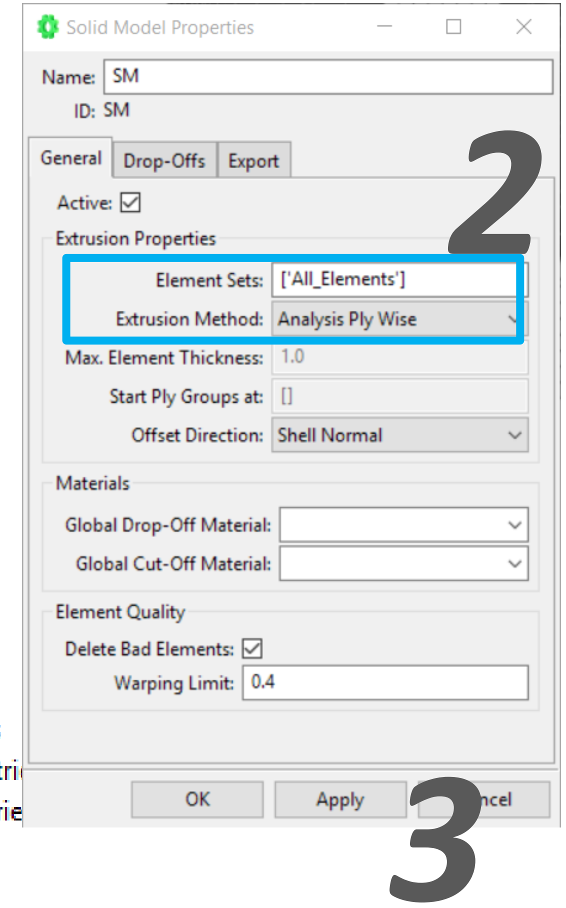
1. Expand *Material Data* to check the fabric type present in the model
2. The material for the fabric specified through *Engineering Data* in the project schematic already contains all the properties required by ACCS to model the resin curing process
3. The laminate definition is already present in ACP. Expand the *Modeling Groups* tab to check the plies applied to the part

Workshop 03: Tool Compensation



Create a Solid Model in ACP





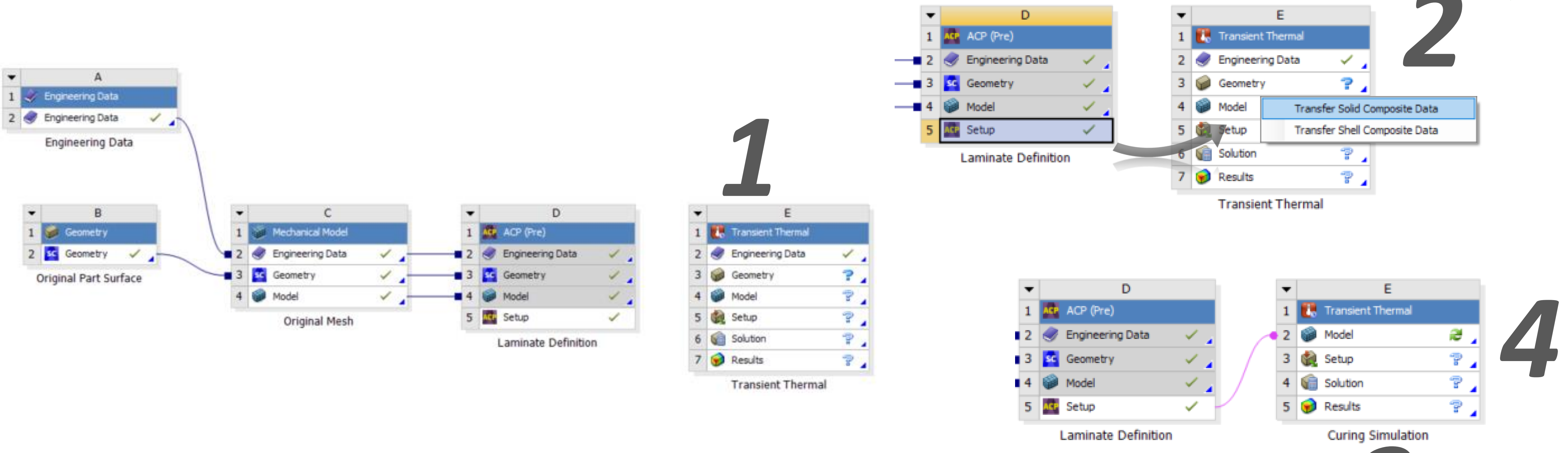
1. Create a new Solid Model (Right Mouse button on *Solid Models* → Create Solid Model)
2. Select All Elements in element sets and extrusion method to Analysis Ply Wise
3. Press Apply to create the solid model by extrusion and return to the project schematic

Note: in curved composite parts, it is recommended to use about 5 elements through the thickness in order to get accurate results.

Workshop

Setting up the Transient Thermal Analysis

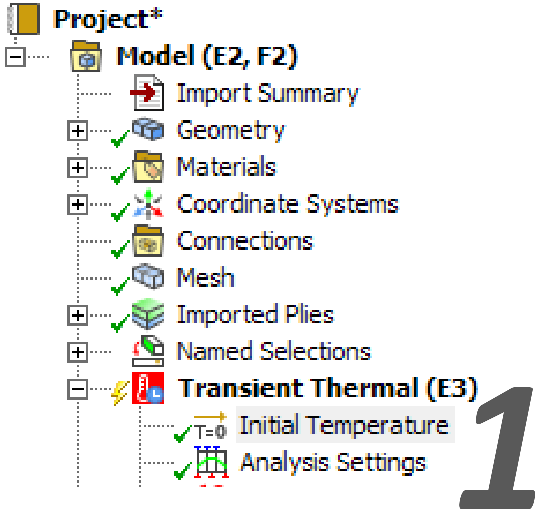
Workshop 03: Tool Compensation

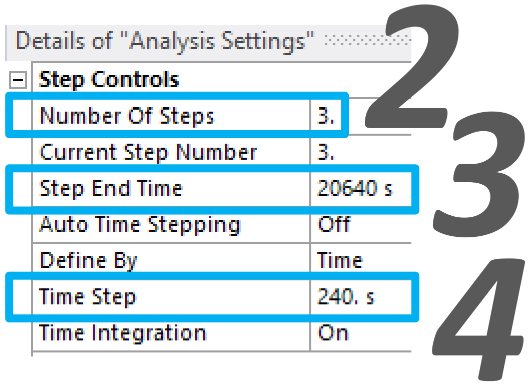


1. Drag and drop *Transient Thermal* onto the Project Schematic
2. Drag and Drop *Setup* of ACP (Pre) onto *Model* of the new *Transient Thermal* cell (select Transfer Solid Composite Data)
3. Rename the new analysis system as *Curing Simulation*
4. Right Mouse button on *Model* → Edit to open Mechanical

Workshop 03: Tool Compensation

Transient Analysis Settings

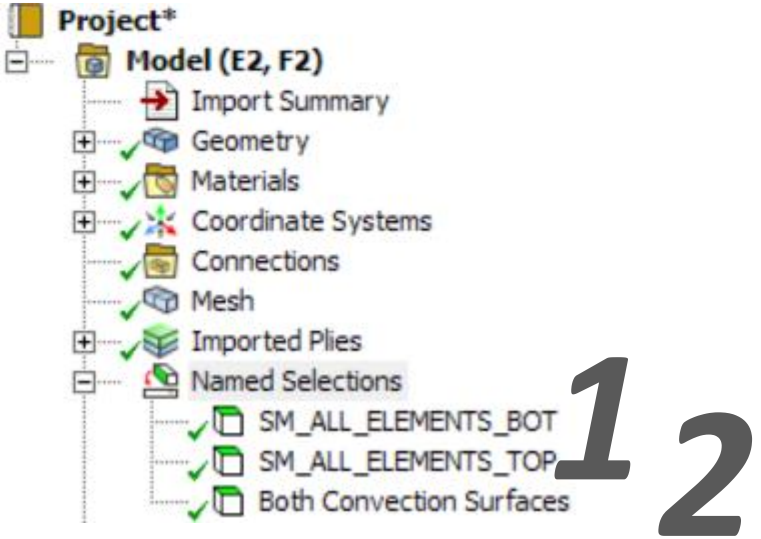


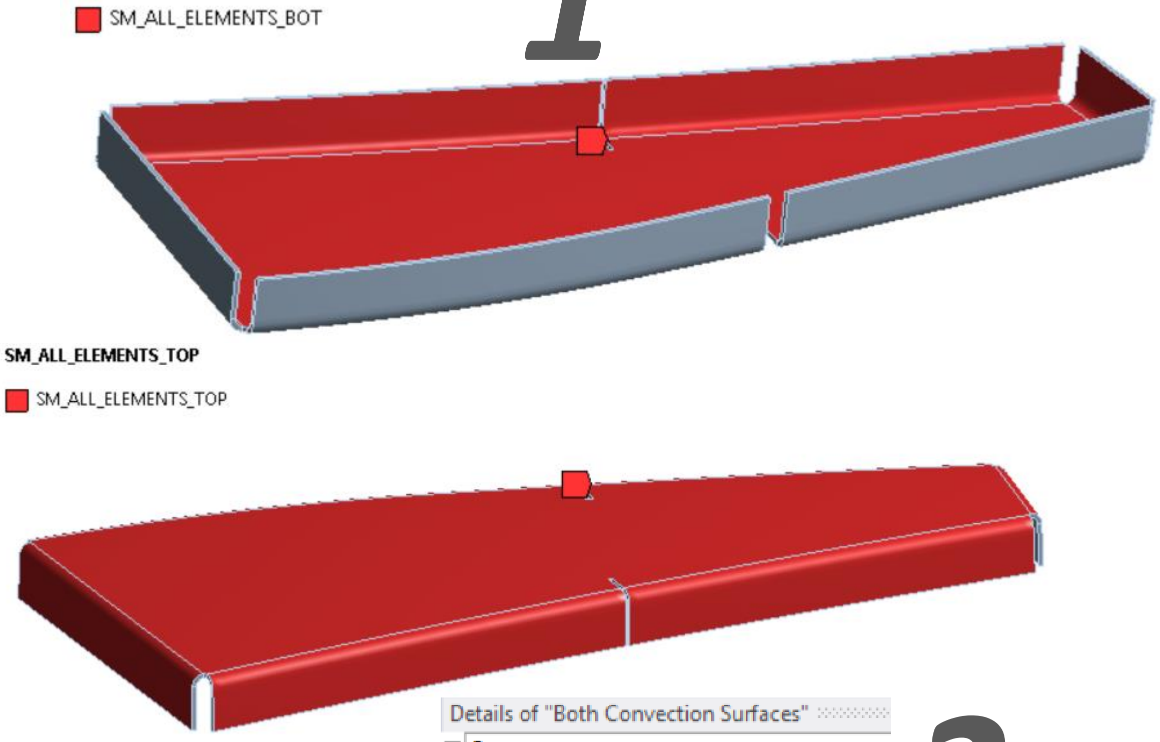


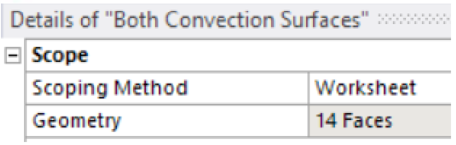
1. Select *Analysis Settings* in the Mechanical tree
2. Choose three as the *Number Of Steps*
3. By selecting the appropriate *Current Step Number*, adjust the *Step End Time* to 15600s, 20300s and 20640s for each step of thermal cycle specified later
4. A *Time Step* of 240s is appropriate for solving this transient thermal analysis

Workshop 03: Tool Compensation

Create Named Selections



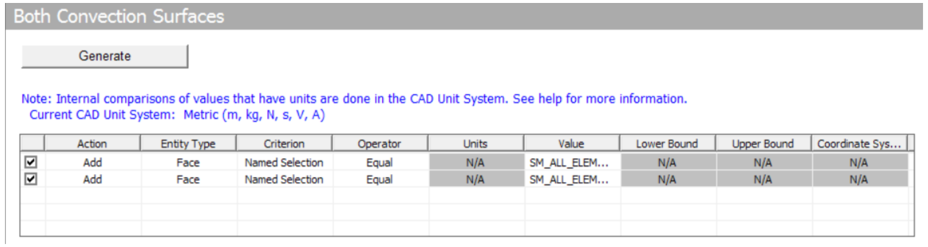




Details of "Both Convection Surfaces"

Scope	
Scoping Method	Worksheet
Geometry	14 Faces

2



Both Convection Surfaces

Generate

Note: Internal comparisons of values that have units are done in the CAD Unit System. See help for more information.
Current CAD Unit System: Metric (m, kg, N, s, V, A)

	Action	Entity Type	Criterion	Operator	Units	Value	Lower Bound	Upper Bound	Coordinate Sys...
☑	Add	Face	Named Selection	Equal	N/A	SM_ALL_ELEM...	N/A	N/A	N/A
☑	Add	Face	Named Selection	Equal	N/A	SM_ALL_ELEM...	N/A	N/A	N/A

1. Add named selections in the mechanical tree for faces at the bottom and top of the composite frame
2. Combine the previous named selections using a *Worksheet*

Workshop 03: Tool Compensation

Set the Thermal Cycle

- Project*
  - Model (E2, F2)
    - Import Summary
    - Geometry
    - Materials
    - Coordinate Systems
    - Connections
    - Mesh
    - Imported Plies
    - Named Selections
    - Transient Thermal (E3) **1**
      - Initial Temperature
      - Analysis Settings
      - Convection **2**

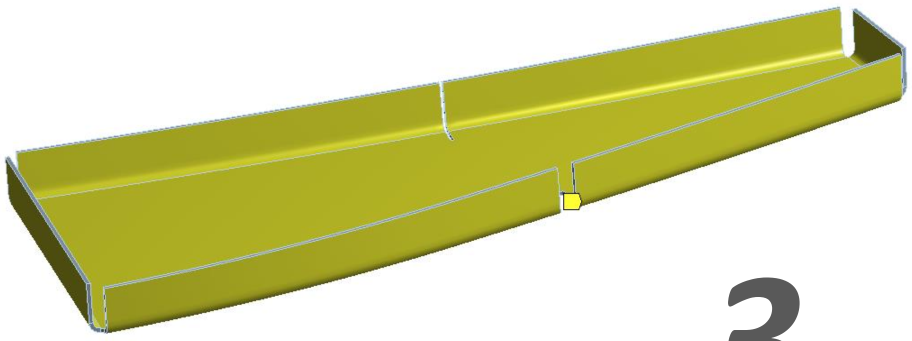
Details of "Initial Temperature" **1**

Definition	
Initial Temperature	Uniform Temperature
Initial Temperature Value	20. °C

Details of "Convection"

Scope	
Scoping Method	Named Selection
Named Selection	Both Convection Surfaces
Definition	
ID (Beta)	81
Type	Convection
Film Coefficient	Tabular Data
Ambient Temperature	Tabular Data
Convection Matrix	Program Controlled
Suppressed	No
Edit Data For	Film Coefficient
Tabular Data	
Independent Variable	Time

2 **3**



3

Tabular Data

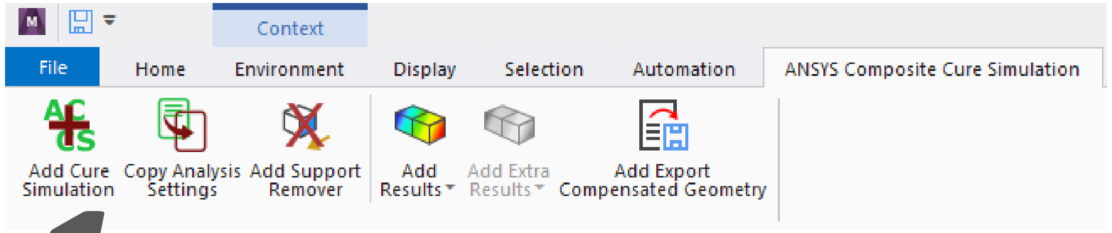
	Steps	Time [s]	Convection Coefficient [W/mm²·°C]	Temperature [°C]
1	1	0.	= 1.e-004	20.
2	1	3300.	1.e-004	120.
3	1	6600.	1.e-004	120.
4	1	8400.	1.e-004	180.
5	1	15600	1.e-004	180.
6	2	20400	= 1.e-004	20.
7	3	20640	= 1.e-004	= 20.

1. Select *Initial Temperature* in the Mechanical tree and specify a starting temperature of 20°C
2. Add a *Convection* boundary condition to the named selection previously defined
3. Specify the *Convection Coefficient* and the ambient *Temperature* using a tabular data to set the thermal cycle for the curing process

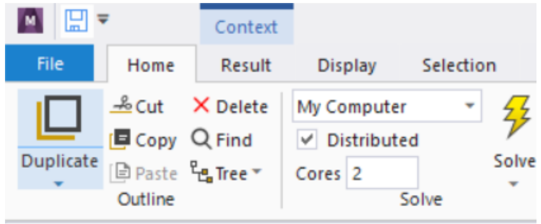
Ansys

Workshop 03: Tool Compensation

Run the Curing Simulation



1

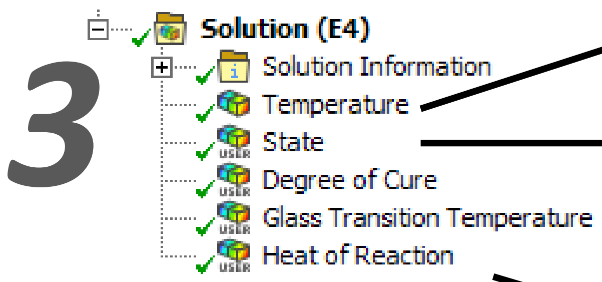


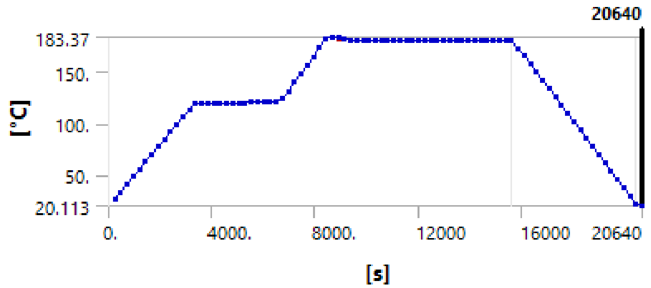
2

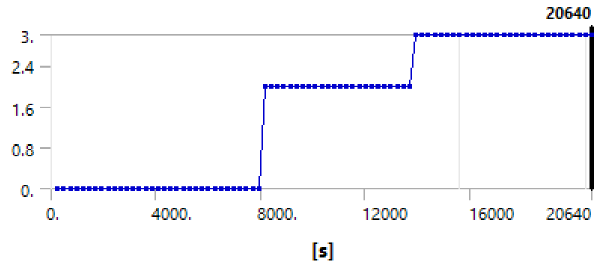
1. Use the ribbon bar to enable the chemical-thermal solver by adding the ANSYS Composite Cure Simulation object to the analysis and to add results of interests
2. Solve the model

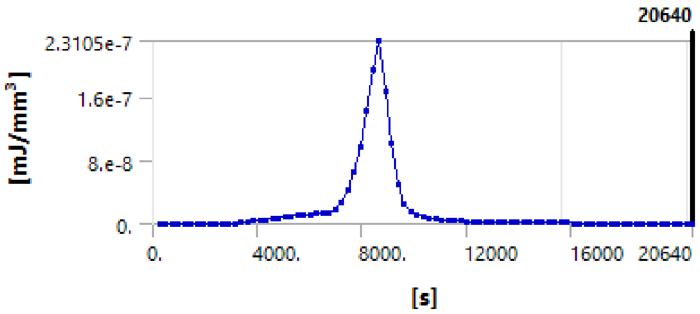
Workshop 03: Tool Compensation

Run the Curing Simulation









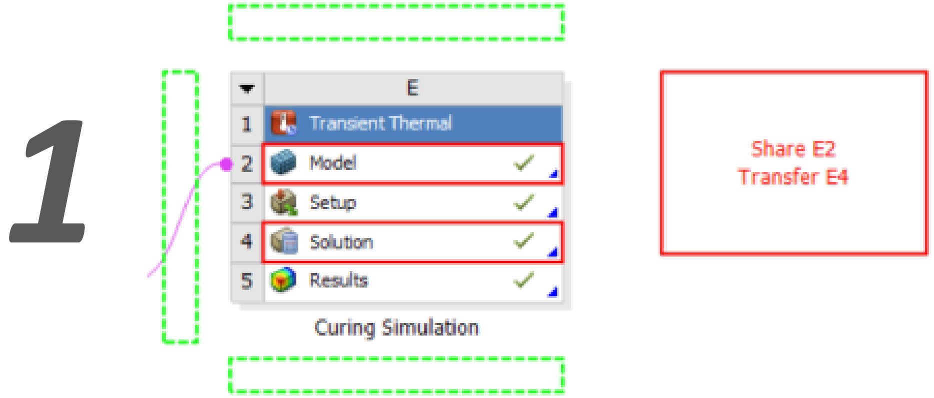
By monitoring the exothermal peak, the maximum temperature in the part, the degrees of curing and state of the material we can optimize the thermal cycle of the curing simulation

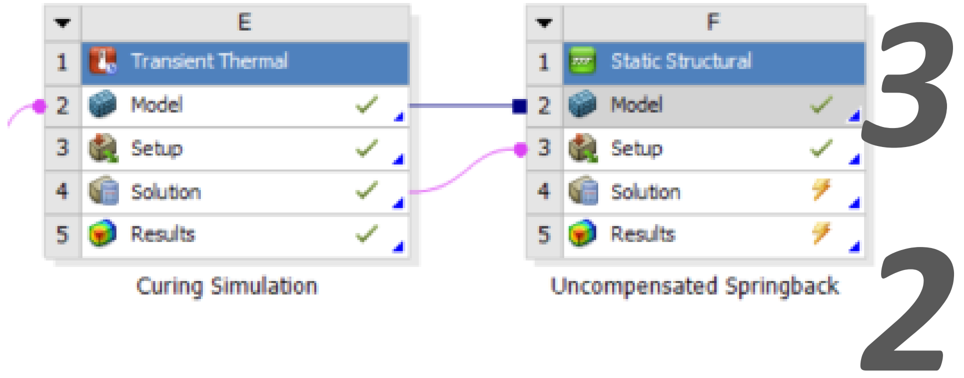
1. When the model is solved, examine the maximum temperature recorded on the composite part during the curing process, the transition state (liquid = 0 , rubbery = 2, and glassy = 3), and the heat released during the reaction

Workshop

Setting up the Static Structural

Workshop 03: Tool Compensation

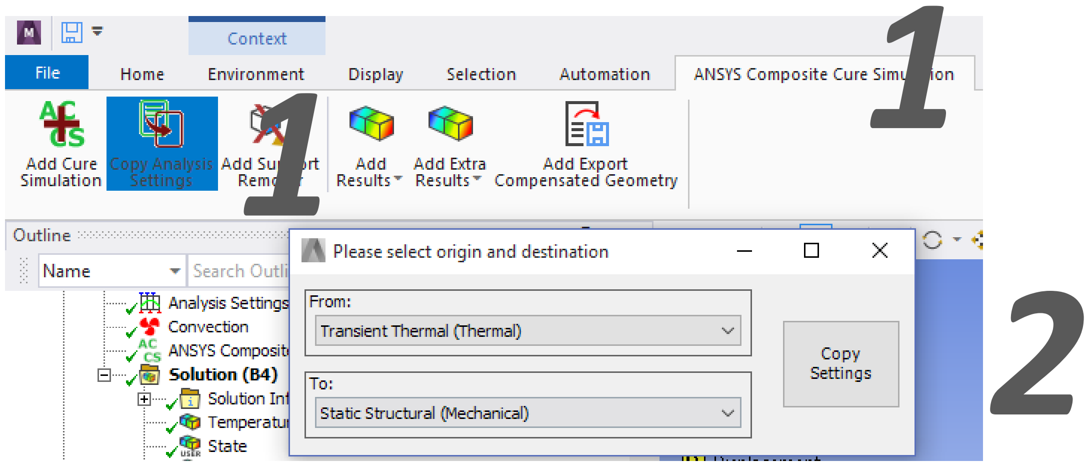


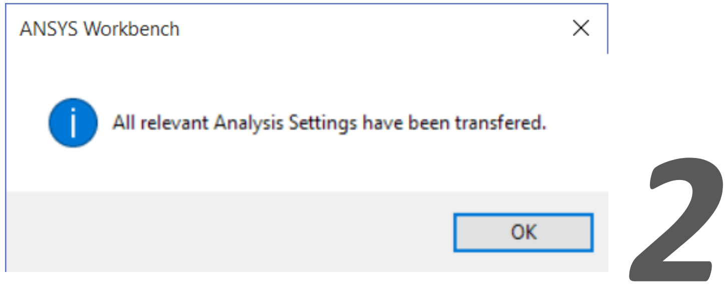


1. Go back to the Workbench project schematic and a drag and drop onto the *Thermal Analysis* a *Static Structural* cell
2. Rename the *Static Structural* cell as *Uncompensated Springback*
3. Return to Mechanical to evaluate the deformation induced on the composite part by the curing process

Workshop 03: Tool Compensation

Copy Analysis Settings

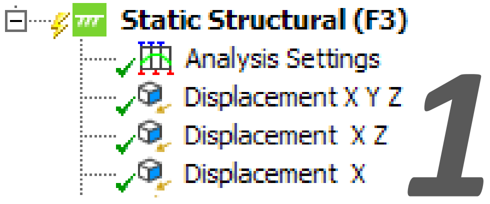




1. In the main menu tab, go to ANSYS Composite Cure Simulation and select “Copy Analysis Settings”. This allows the user to copy the relevant analysis settings between the two analyses.
2. Click on “Copy Settings” and then “OK” in the notification that pops up.

Workshop 03: Tool Compensation

Restrain Arbitrary Rotations of the Part



Details of "Displacement X Y Z"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
ID (Beta)	105
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	0. mm (ramped)
Y Component	0. mm (ramped)
Z Component	0. mm (ramped)
Suppressed	No

Details of "Displacement X Z"

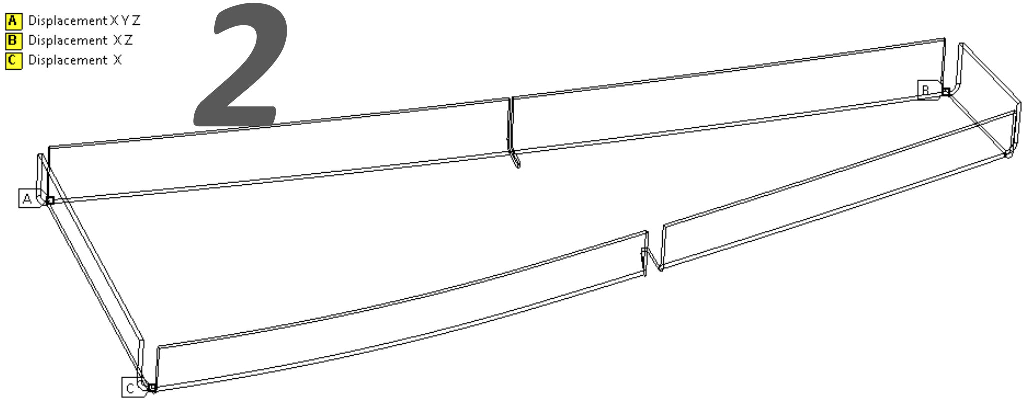
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
ID (Beta)	107
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	0. mm (ramped)
Y Component	Free
Z Component	0. mm (ramped)
Suppressed	No

Details of "Displacement X"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
ID (Beta)	109
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	0. mm (ramped)
Y Component	Free
Z Component	Free
Suppressed	No

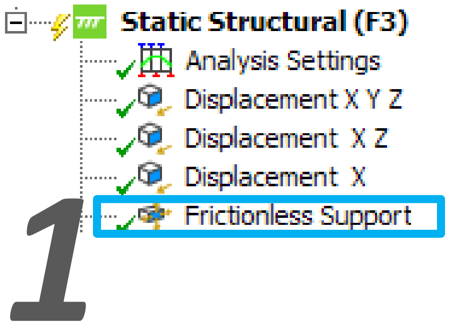
The displacement of the part must be restricted to prevent arbitrary rigid body motions:

1. Add displacement boundary conditions in the Mechanical tree
2. Select in *Scope* a different vertex for each displacement boundary condition
3. Constrain all the degree of freedom of the first vertex, while progressively freeing other two degrees of freedom for the remaining vertices



Workshop 03: Tool Compensation

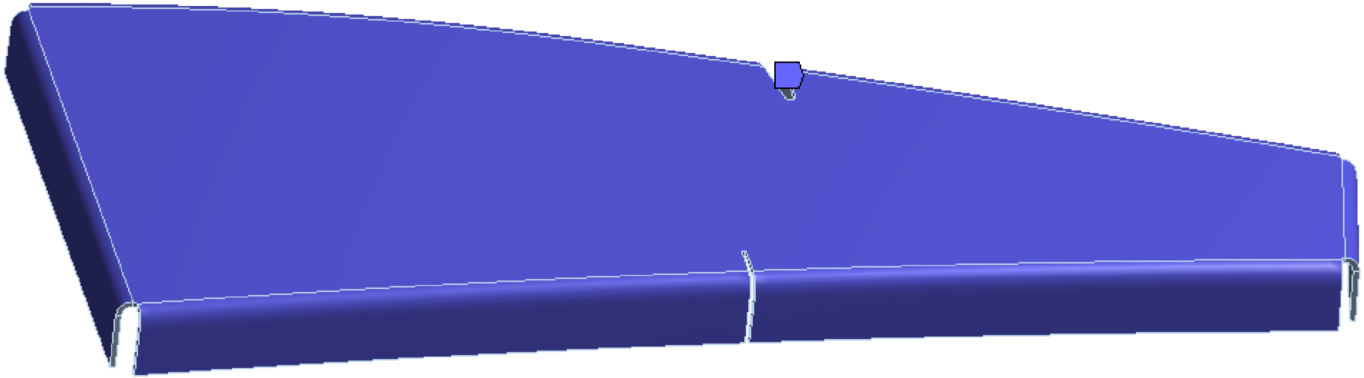
Simulate Mold Interaction



Details of "Frictionless Support"

Scope	
Scoping Method	Named Selection
Named Selection	SM_ALL_ELEMENTS_TOP
Definition	
ID (Beta)	111
Type	Frictionless Support
Suppressed	No

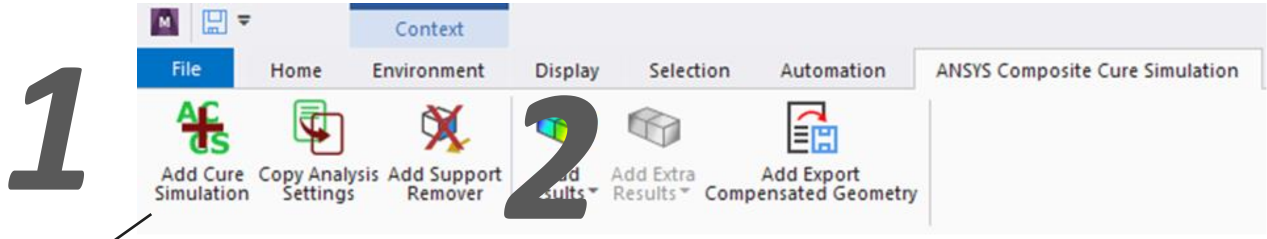
2

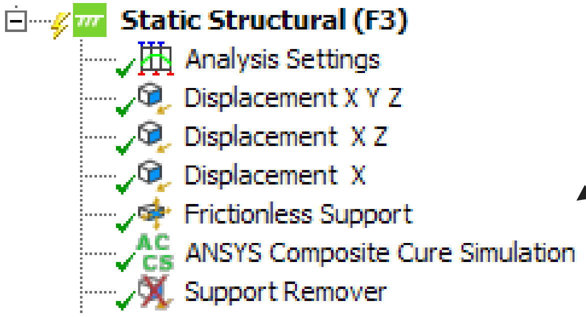


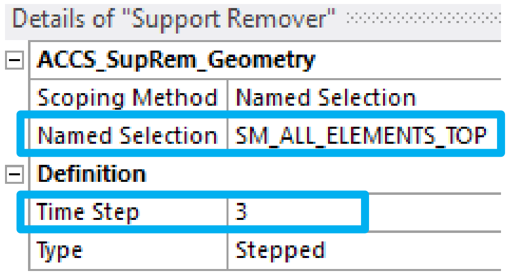
1. Add a *Frictionless Support* in the Mechanical tree to simulate the contact with the mold
2. Apply the support to the top of the laminate, use the named selections defined previously

Workshop 03: Tool Compensation

Simulate Demolding





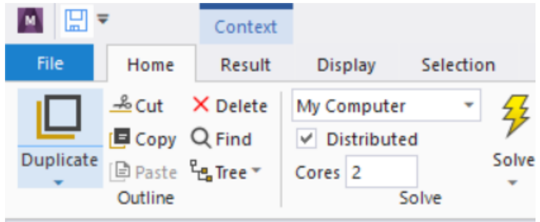


1. Use the ribbon bar to add the ANSYS Composite Cure Simulation object to the analysis

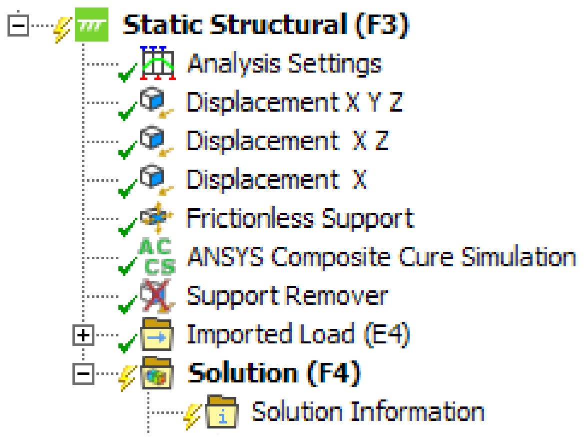
2. Insert in the tree a *Support Remover* object from the curing tool, to simulate the demolding of the composite part from the tooling, demolding is applied in the last step time step of curing process to those faces where the frictionless contact was previously applied to simulate contact with the mold

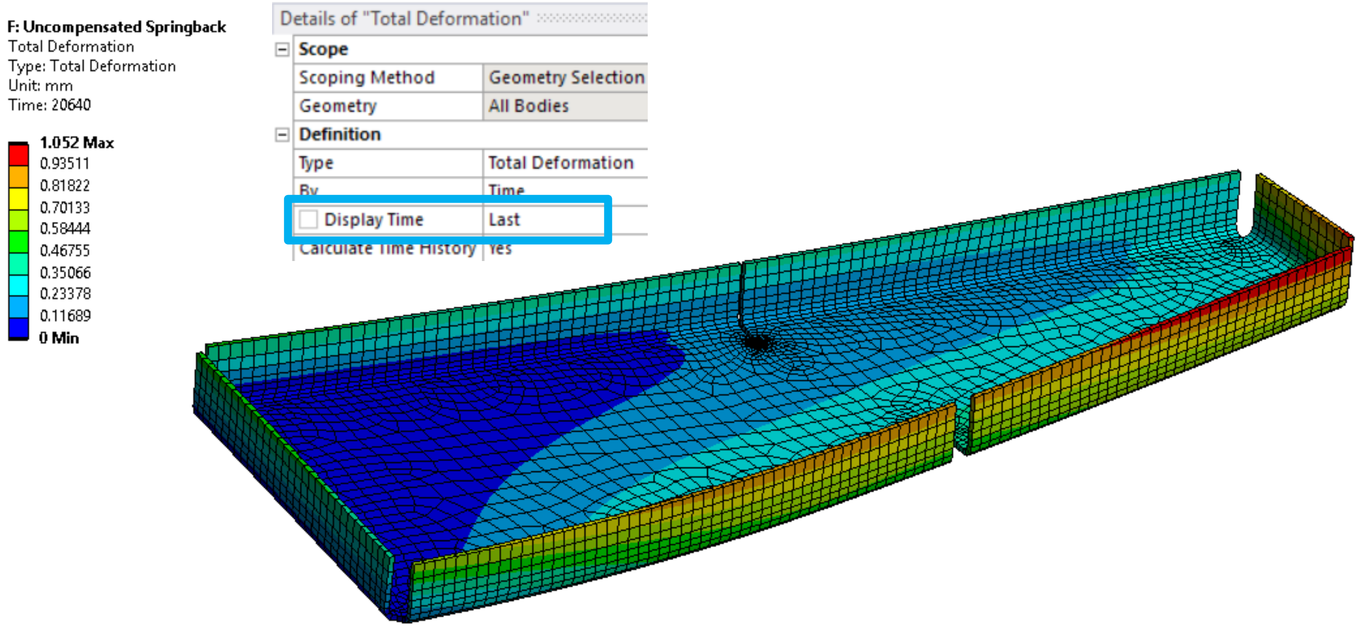
Workshop 03: Tool Compensation

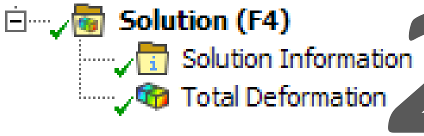
Residual Deformation at the End of the Curing Process



1







2

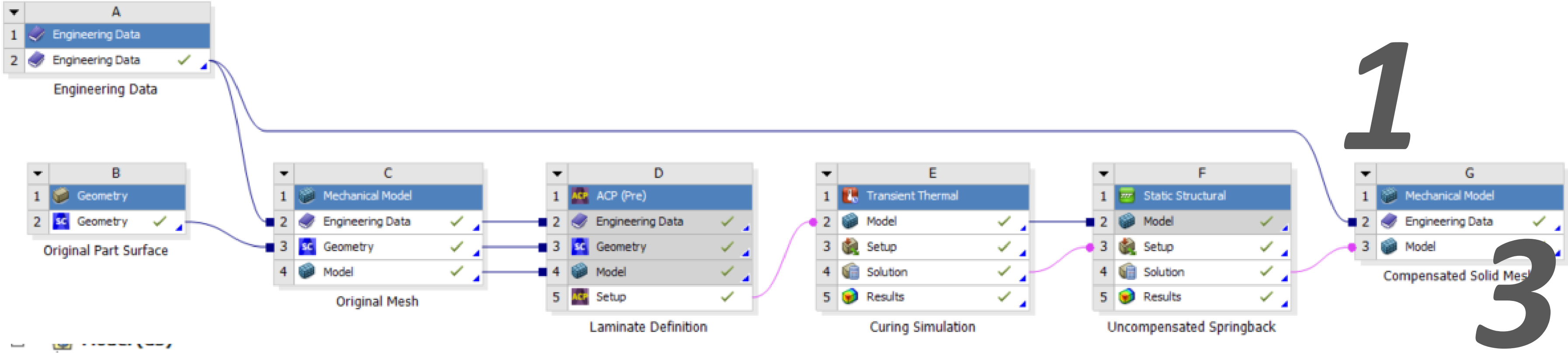
1. Solve the model
2. When the model is solved, add a *Total Deformation* plot to the solution to examine the residual deformation in the part at the end of the curing process
3. Go back to the project schematic in Workbench

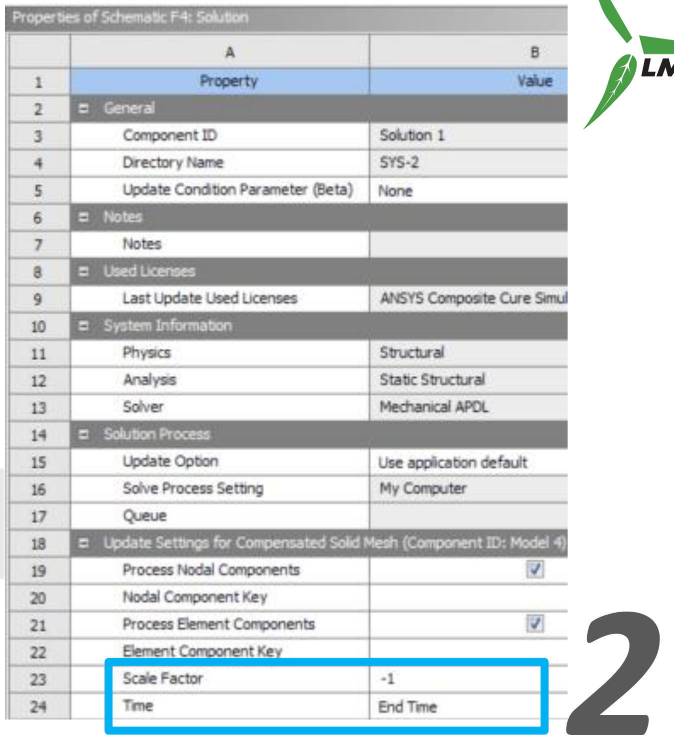
Workshop

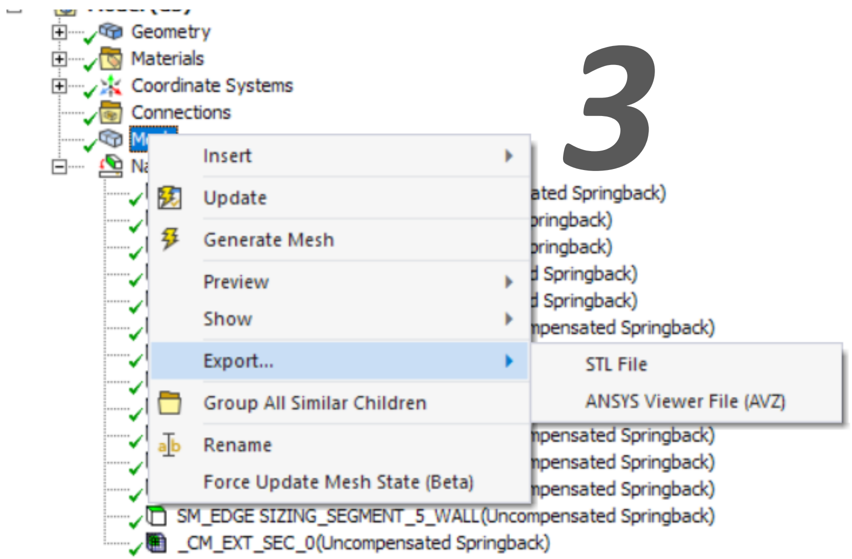
Compensating the Tool Geometry

Workshop 03: Tool Compensation

Create a Compensated Mesh of the Tool



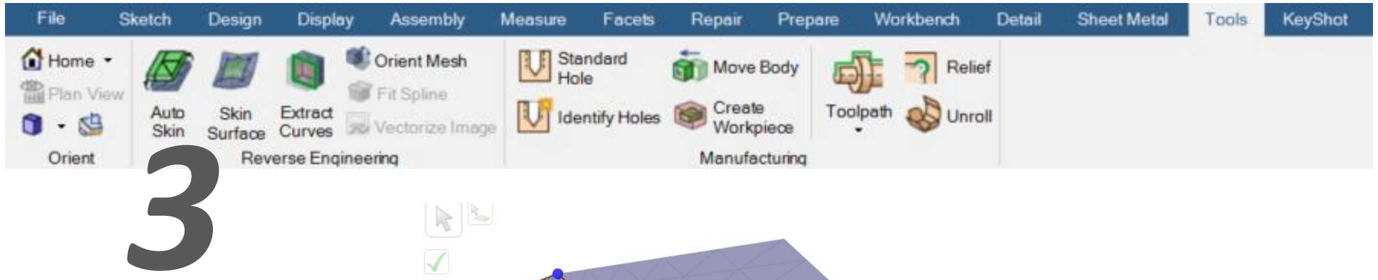


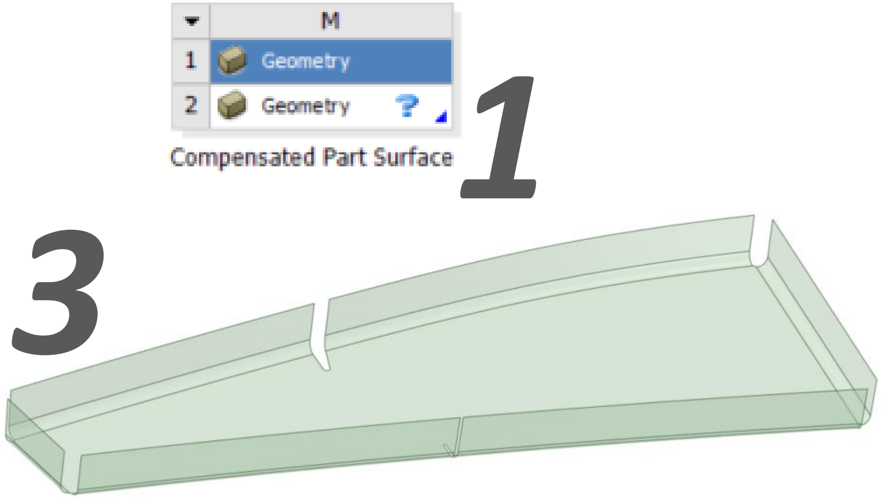


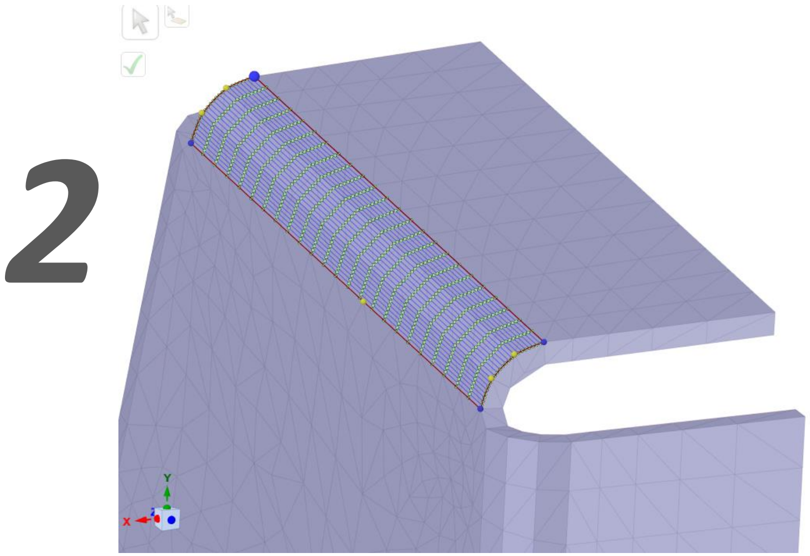
1. Drag and drop *Mechanical Model* onto the project schematic and drag and drop the *Solution* of the *Static Structural* onto *Model* of the new *Mechanical Model*, rename the system as *Compensated Solid Mesh*
2. Change the Scale factor of the Solution of the Static Structural system to “-1”: this will transfer the compensated geometry to the *Mechanical Model*
3. Right Mouse button on *Model* → Edit to open a new Mechanical environment where you can export the compensated mesh in *STL* format

Workshop 03: Tool Compensation

Prepare the Compensated Geometry

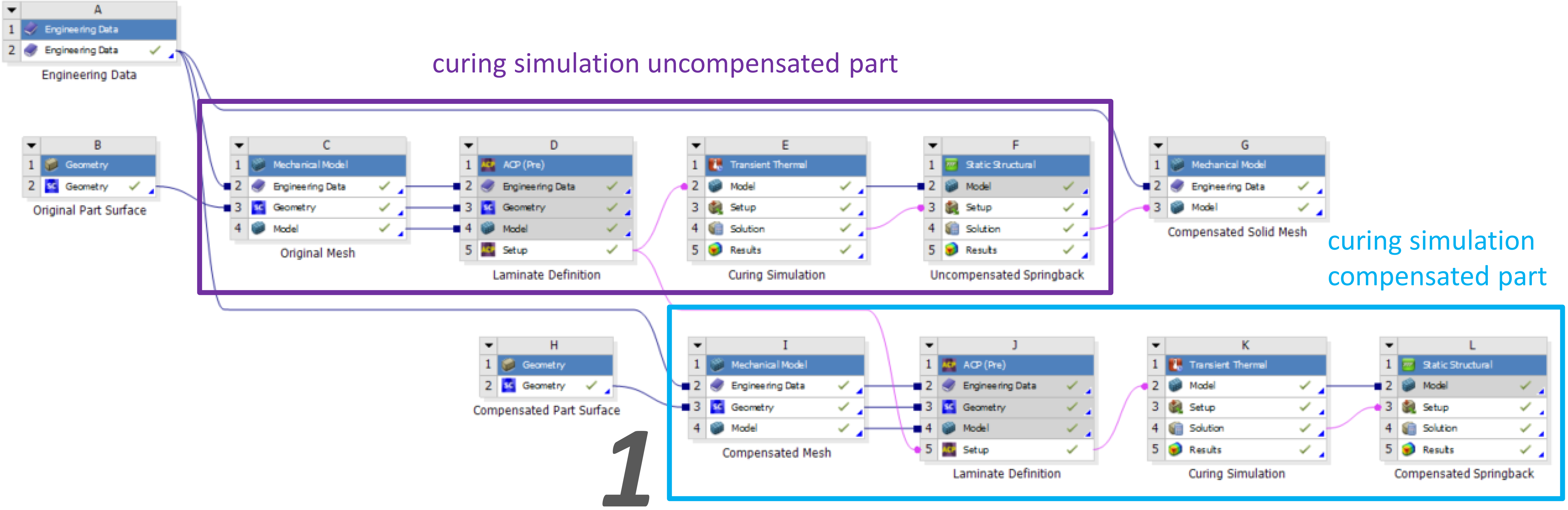






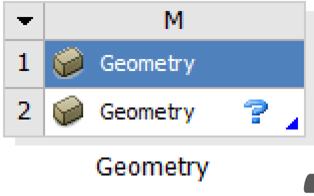
1. Drag and drop a new *Geometry* system in the project schematic, RMB on *Geometry* and select to edit geometry in SpaceClaim
2. Load the compensated mesh in STL format
3. Use *Skin Surface among* the *Reverse Engineering* tools on the SpaceClaim ribbon bar to create a new surface using the mesh as a reference. Refer to SpaceClaim training material for additional details on the usage of *Skin Surface*

Workshop 03: Tool Compensation

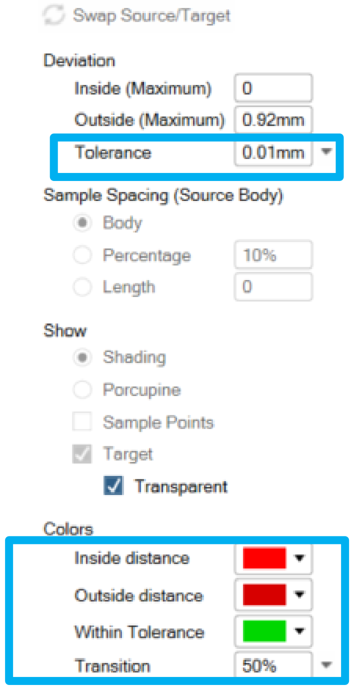


1. Go back to the project schematic and repeat the same steps of this workshop to realize a new curing simulation with the compensated geometry, the laminate information and material properties can be transferred directly from the previous analysis

Workshop 03: Tool Compensation
Validate the Compensation

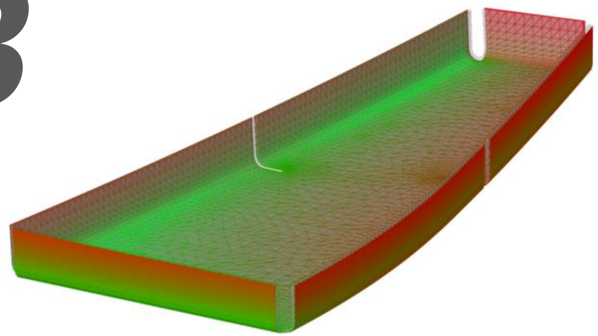


1

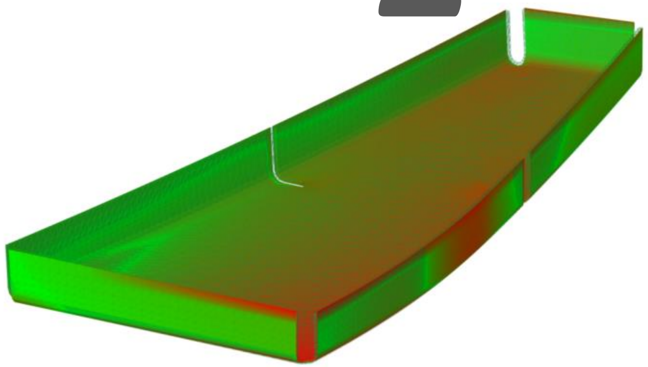


3

2



Target Geom.: undeformed outer surface
Source Geom.: deformed mesh with <u>*no*</u> compensation



Target Geom.: undeformed outer surface
Source Geom.: deformed mesh with compensation

1. Drag and drop a new *Geometry* system in the project schematic, RMB on *Geometry* and select to edit geometry in SpaceClaim
2. Use *Deviation* among the *Quality* tools of the SpaceClaim ribbon bar to compare the original undeformed surface used for the first curing simulation to the deformed shapes obtained at the end of curing with and without compensation
3. The visualization and tolerance value of the deviation options can be adjusted: curing with a compensated geometry results in a final shape closer to nominal shape sought for the frame

Workshop 03: Tool Compensation

Workshop Summary

- In this workshop you have learnt the usage of ANSYS Composite Cure Simulation to capture and monitor the thermal-chemical reactions during composite manufacturing, you can use this information to control and optimize your curing process
- You saw how to predict the development process induced distortions
- You understood the process to compensate the tooling geometry to meet the required geometrical tolerances, resulting in an assembly process with the minimal built-in stresses

End of presentation