Workshop 01: C-Shape Profile Full Cure Simulation

Release 2021 R1

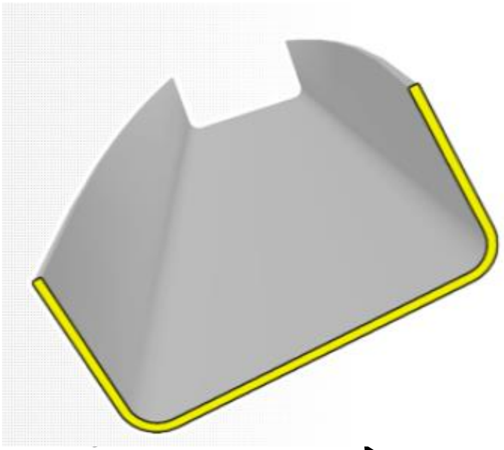
Workshop 01: C-Shape Profile Full Cure Simulation

- In this Workshop, we will model the curing of a C-shape composite profile made from Carbon fiber prepreg.
- Using ACCS, in conjunction with ACP, you will predict the mean process outputs like exothermal reaction, material state, degree of cure, glass transition temperature and heat of reaction.
- In a Static Structural analysis you will forecast the process induced distortions according to the thermal conditions of the curing cycle that the manufacturer recommends.

Background

C-Shape Profile: Problem Description

- During the curing of composite parts, residual stresses develop during the cure resulting in distorted components



Understanding the influence of the process parameters and if necessary adapting them accordingly, plays an important role in the shape and quality of the final part.

Workshop 01: C-Shape Profile Full Cure Simulation

Material Characterization

- During curing of thermosetting composites, the resin undergoes cross-linking reactions that lead to an increase of material density and reduction in volume.
- When monomers link into larger molecules they release energy in the form of heat. The exothermic heat of polymerization causes huge problems in processing especially in the case of thick laminates.
- Many thermo-mechanical properties are needed to solve the kinetic models that are available in ACCS.
- Differential Scanning Calorimetry (DSC)
  - Test performed to determine: Glass transition temperature (Tg), curing state (α) Total heat of cure (HR) and heat capacities (Cp)
  - Analytical models are obtained by empirical fitting of the DCS data.

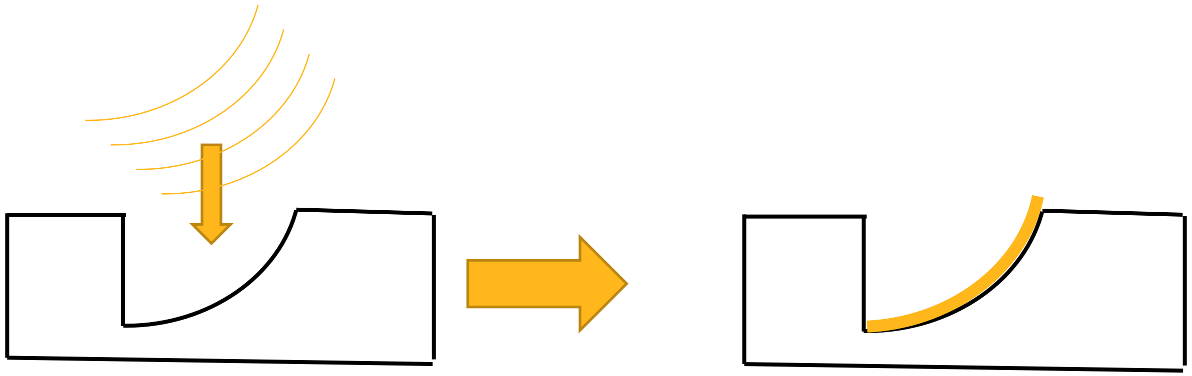
Workshop

Setting up the Transient Thermal Analysis

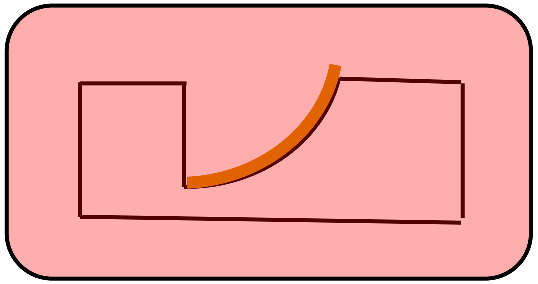
Workshop 01: C-Shape Profile Full Cure Simulation
Introduction

- This Workshop focuses on the full cure and distortion simulation of a quarter C-shape made of [0 90 90 0] AS4-8552 plies.
- The material properties needed for the curing simulation are complex and should be measured empirically. In this case we use a standard material of the ANSYS material library.
- The thermal analysis considers the part to be surrounded by air following the resin manufacturers recommended cure cycle.
- The convection coefficient is set to 200 $W/m^2°C$ on all free surfaces.
- The spatial and temporal temperature distribution obtained from the thermal analysis is then used to compute thermal expansion effects, material properties evolution and cure shrinkage.
- The mechanical analysis considers that the part is lying inside a cylindrical mold and the mold-part interface is modelled by a frictionless support condition.
- The Support Remover allows to simulate the opening of the mold by removing the frictionless support condition at the last step of the analysis. Additional constraints are applied to prevent rigid body motion of the part.

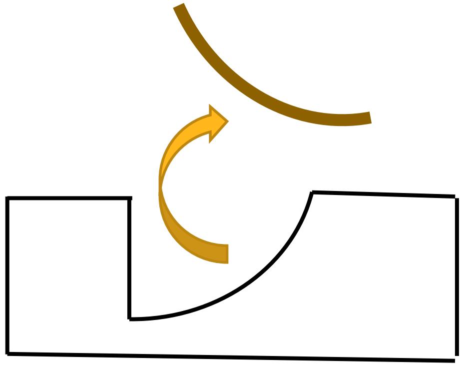
Workshop 01: C-Shape Profile Full Cure Simulation
Problem Introduction



1) Prepreg layers are placed on the negative mold



2) The mold with the prepreg
is put in the autoclave for cure

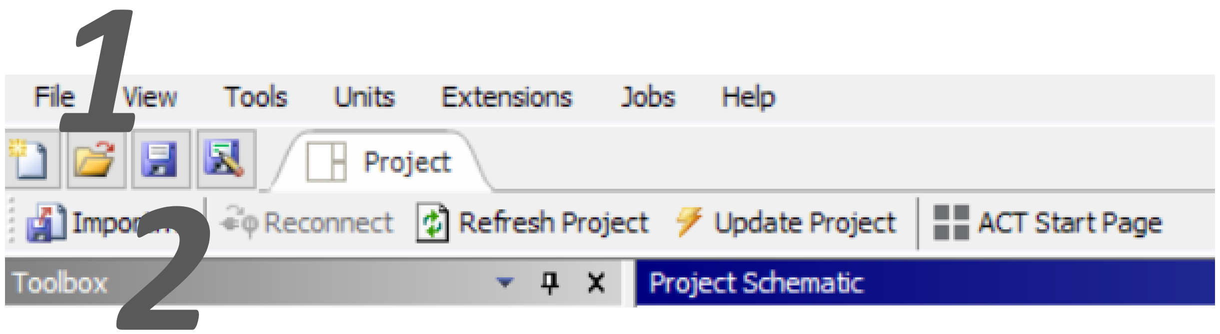


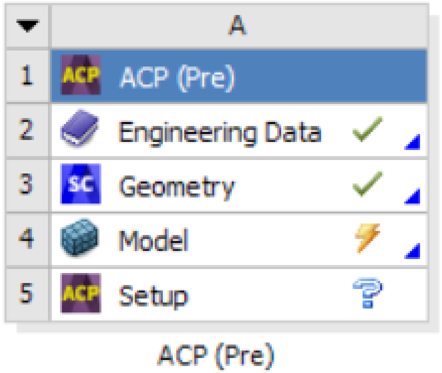
3) Out of the autoclave, the cured
part is released from the mold

Workshop 01: C-Shape Profile Full Cure Simulation

Opening the Project

Start ANSYS Workbench and Open Archive





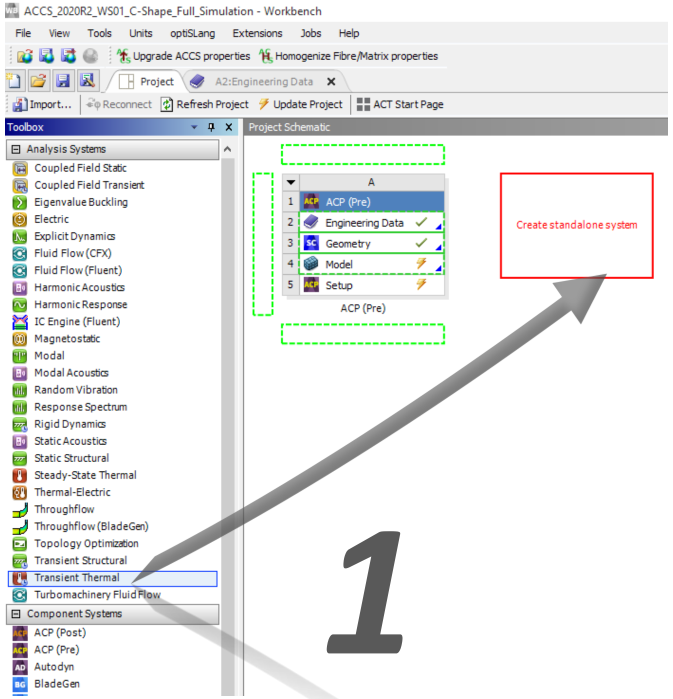
1. Start ANSYS Workbench and open the archive `ACCS_<Release>_WS01_C-Shape_Full_Simulation.wbpz`
2. Save the Workbench project
3. Update the project

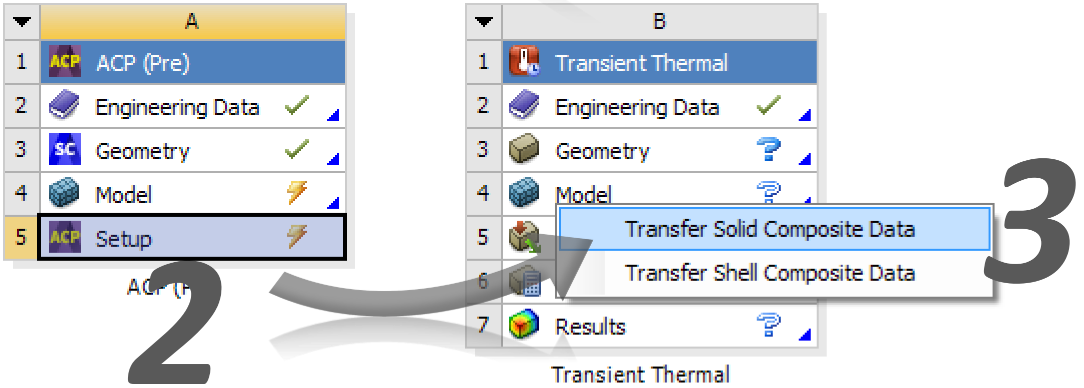
Workshop 01: C-Shape Profile Full Cure Simulation

Adding the Transient Thermal Analysis

The composite model is already built in. In order to learn how to setup a composite structure like this, please follow the ACP training courses.

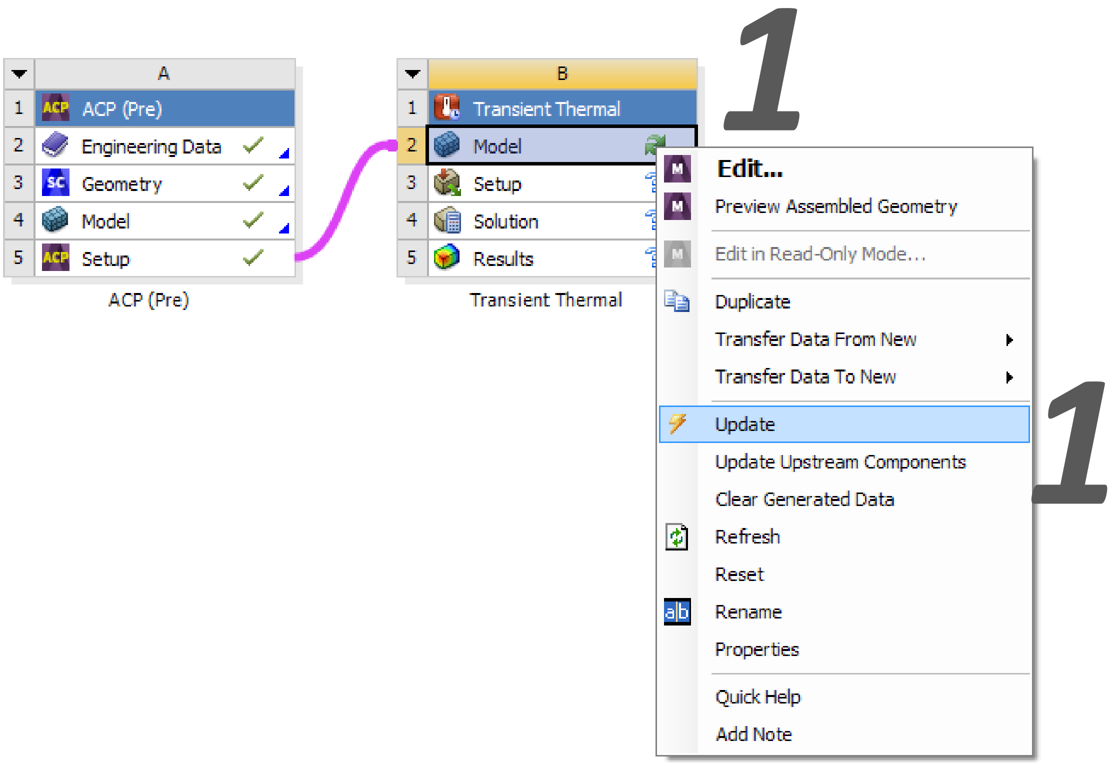
1. First, we need to analyze the thermal process. For that drag and drop a "Transient Thermal" Analysis System to the "Project schematic".
2. Drag and Drop Setup of "ACP (Pre)" onto Model of the new Transient Thermal
3. Select “Transfer Solid Composite Data”.





Workshop 01: C-Shape Profile Full Cure Simulation
Updating the Model

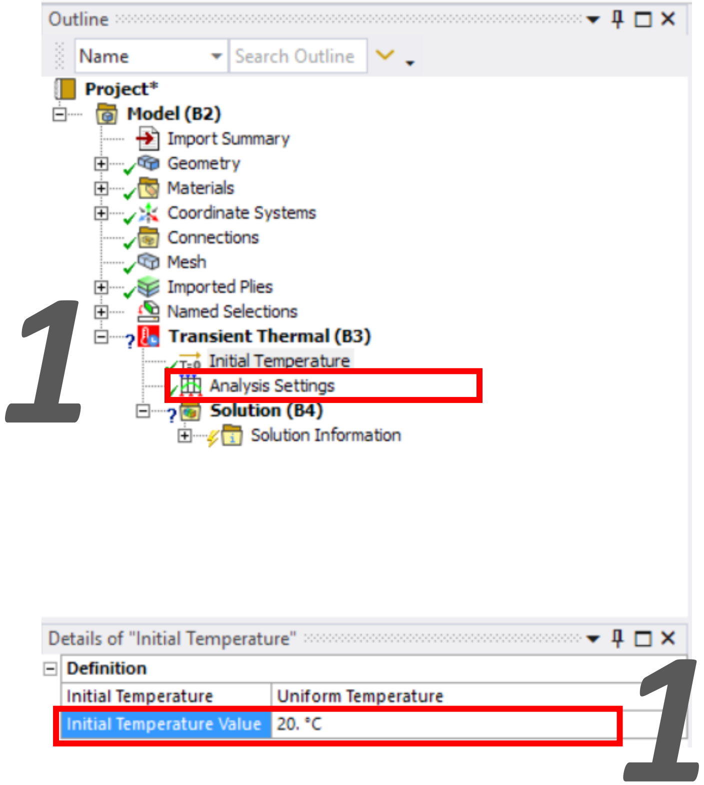
1. Right click in the Setup cell of the Transient Thermal and Update the project.



Workshop 01: C-Shape Profile Full Cure Simulation
Setting the Initial Temperature

The initial temperature must match with the initial temperature of the curing cycle. In this case is 20°C

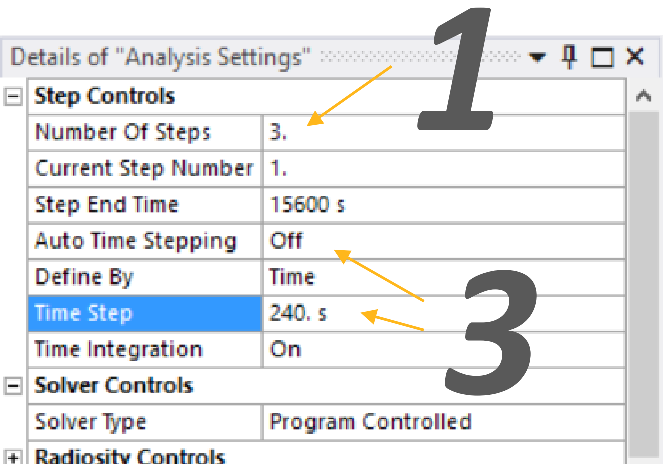
1. By clicking in the “Initial Temperature” tab you can set it to 20°C under the “Details of Initial Temperature”

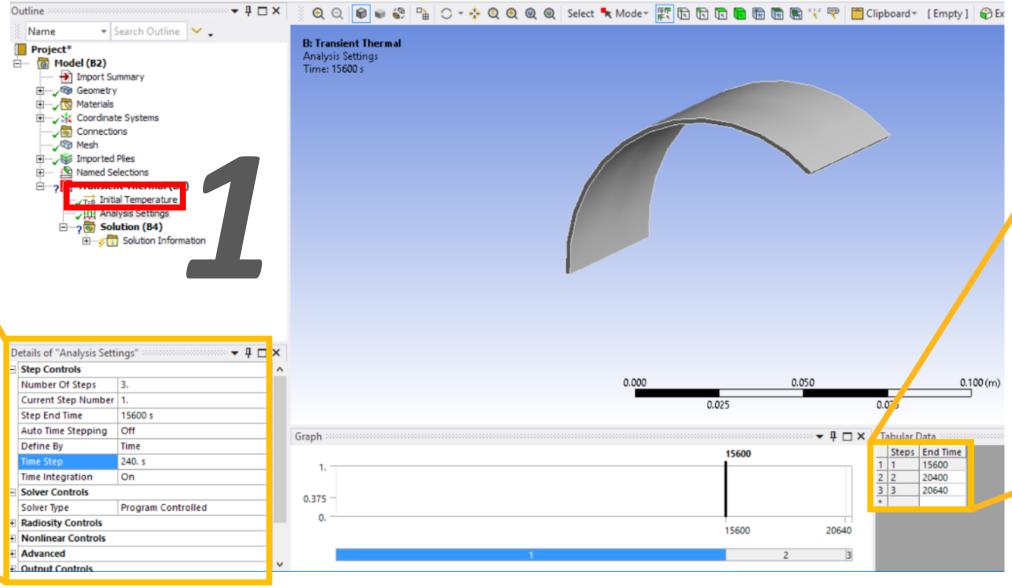


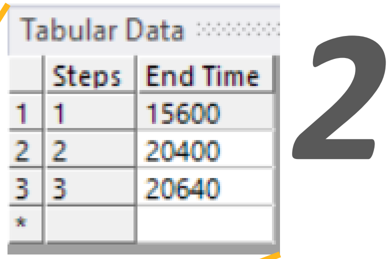
Workshop 01: C-Shape Profile Full Cure Simulation
Analysis Settings

Now we set the number of steps and the time steps of the curing cycle

1. Click in “Analysis Settings” and set the Number Of Steps to 3 which are the same as the no. of steps of the autoclave process.
2. Then in the “Tabular Data” tab, start editing the “End Time” of step 3 to 20640 s, edit then step 2 to 20400 s and finally step 1 to 15600 s.
3. <u>For all three steps</u>, set the “Auto Time Stepping” to “Off” and define the Time Step to 240 s. The time step must be small enough to catch the curing process.

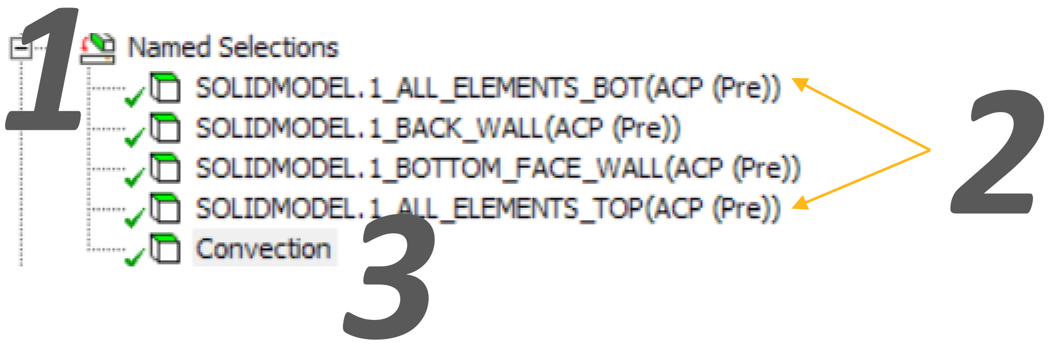


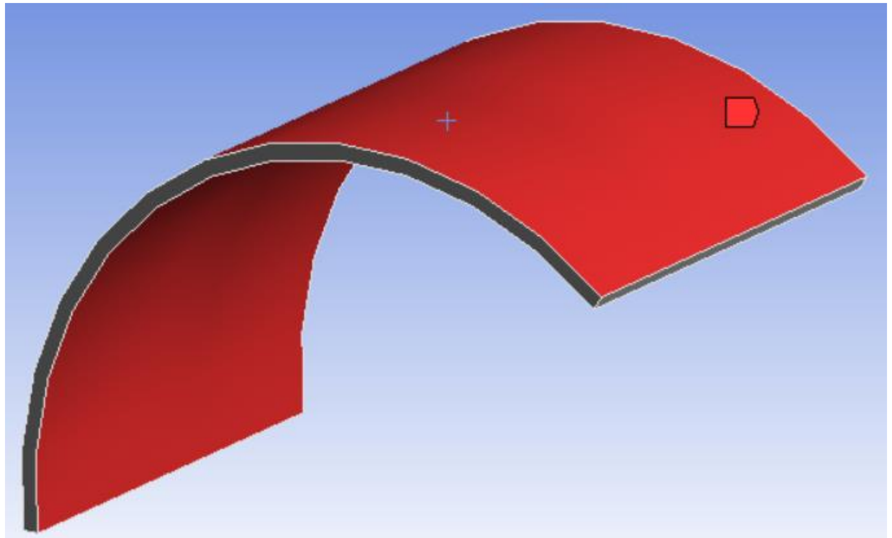


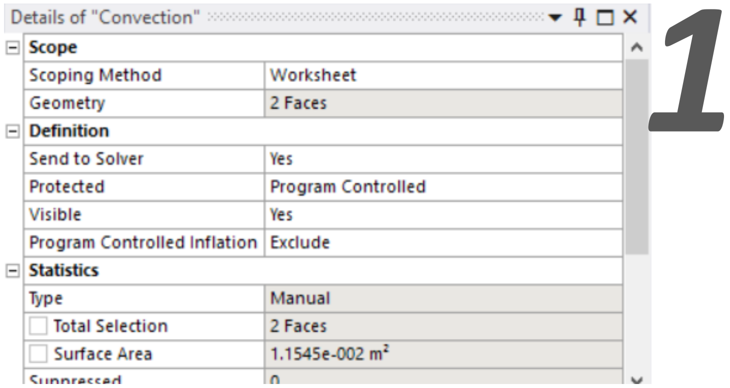


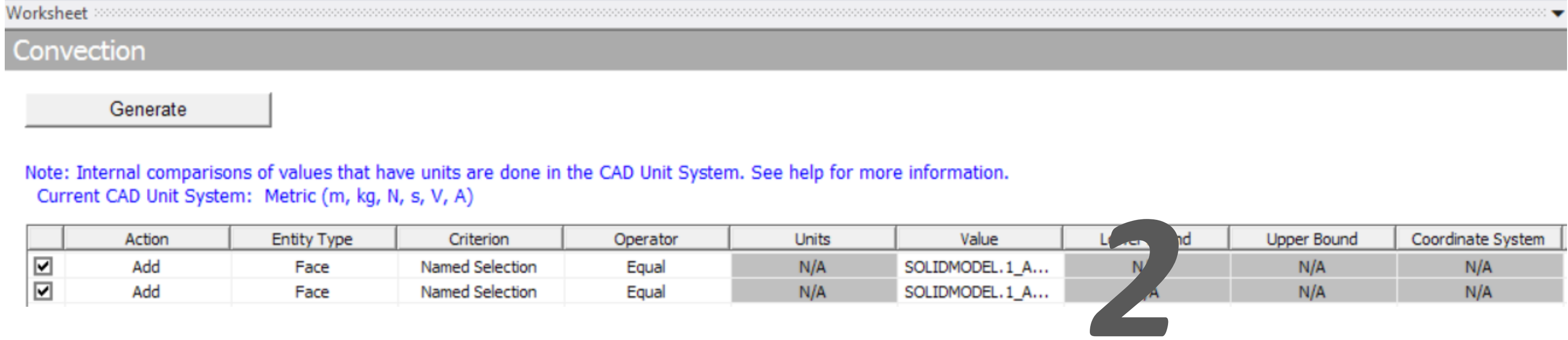
Workshop 01: C-Shape Profile Full Cure Simulation

Create a new Named Selection





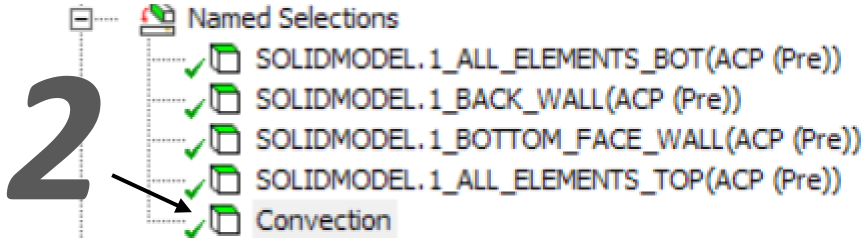


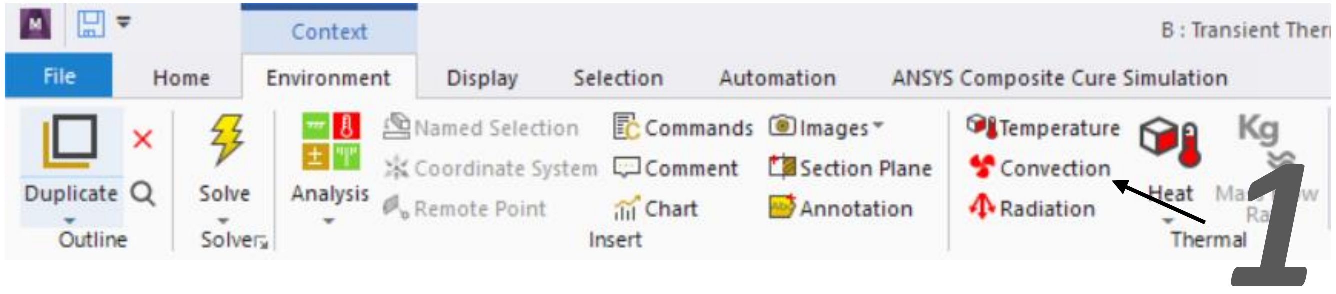


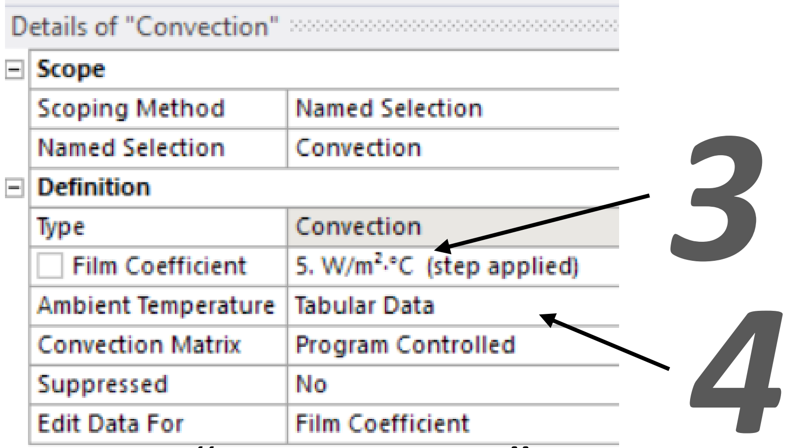
1. Create a new worksheet based Named Selection
2. Add two new actions with the Named Selection Criterion and select the “SOLIDMODEL.1_ALL_ELEMENTS_BOT” and “…_TOP”)
3. Rename the Named Selection as “Convection”

Workshop 01: C-Shape Profile Full Cure Simulation

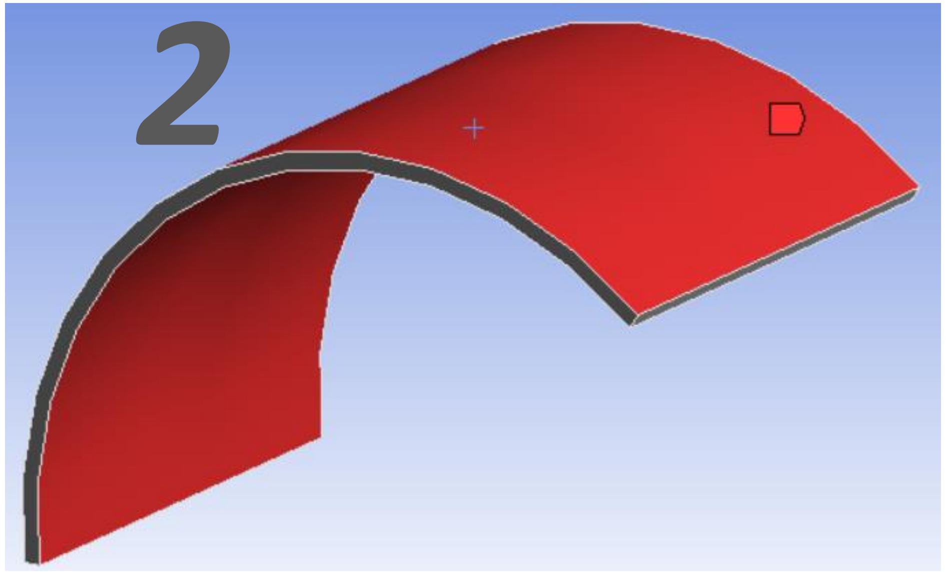
Boundary Conditions






Details of "Convection"

Scope	
Scoping Method	Named Selection
Named Selection	Convection
Definition	
Type	Convection
Film Coefficient	5. W/m²·°C (step applied)
Ambient Temperature	Tabular Data
Convection Matrix	Program Controlled
Suppressed	No
Edit Data For	Film Coefficient

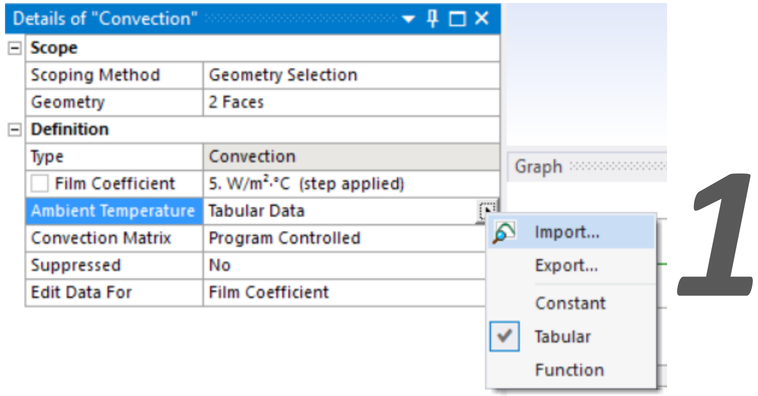


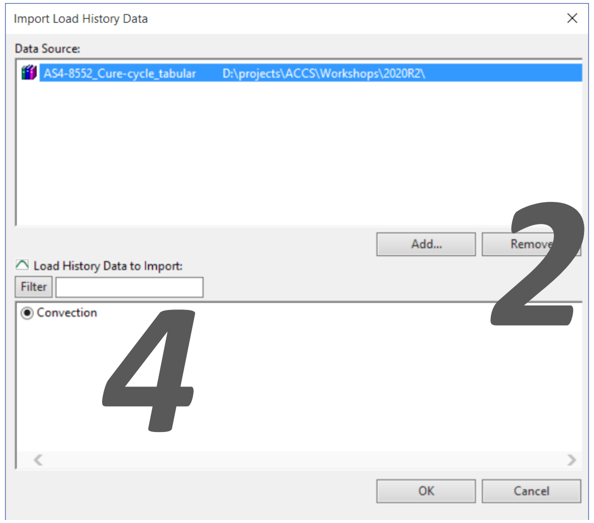
1. Insert a "Convection" environment,
2. select the "Convection" Named Selection
3. change the film coefficient to 5 W/m^2 °C, typical value for natural air convection.
4. and select "Tabular" for the "Ambient Temperature".

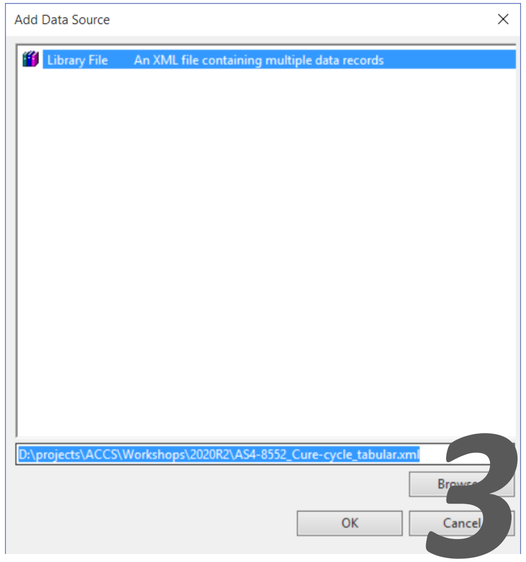
Workshop 01: C-Shape Profile Full Cure Simulation
Configurating the Cure Cycle

Now we are selecting the curing cycle specified by the resin manufacturer.

1. In the “Tabular Data” of the “Ambient Temperature” tab, select “Import…”
2. Click on “Add” and then “Browse” and select the file “AS4-8552_Cure-cycle_tabular.xml”.
3. Click on “Open” and “OK”
4. Select the “Convection” option and then “OK”. A graph of the temperature gradient will be shown in the Convection boundary condition.





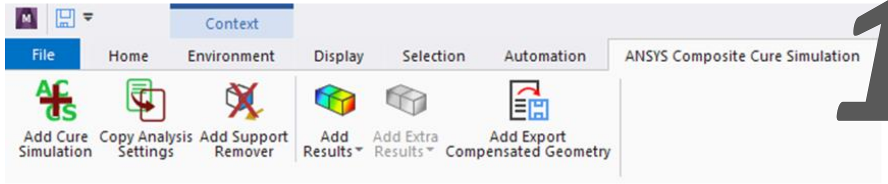


Workshop 01: C-Shape Profile Full Cure Simulation
Adding the ACCS Solver Routines

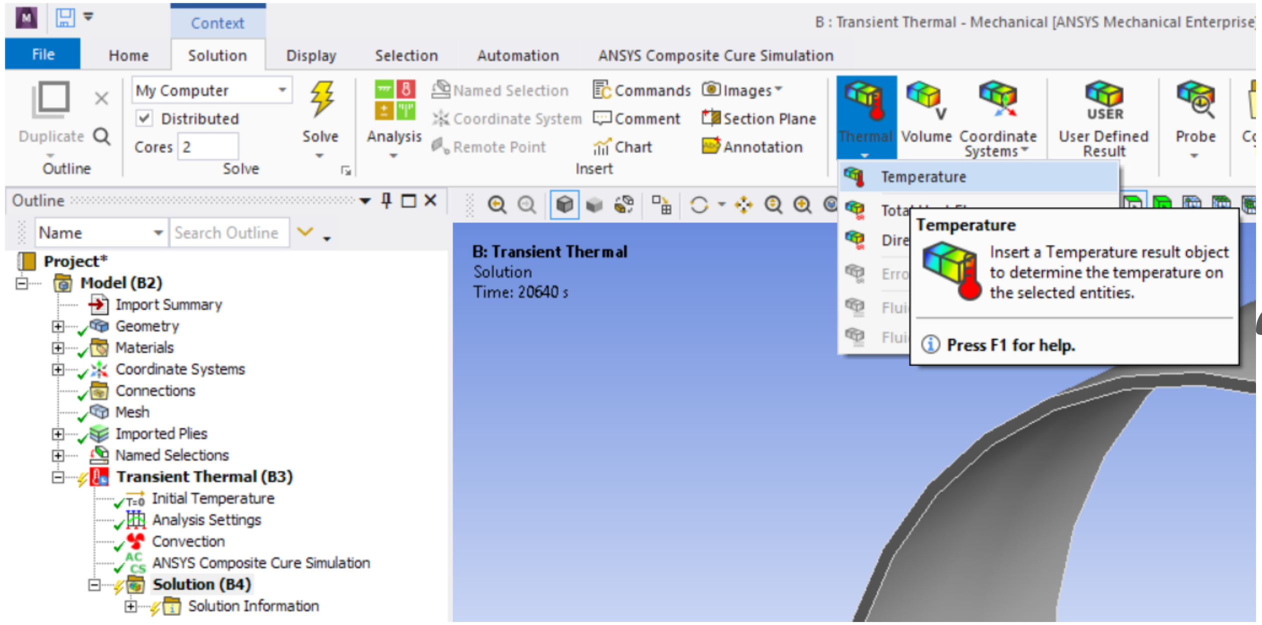
Now we are going to add the ANSYS solver with chemical cure and cure shrinkage routines for materials with defined cure kinetic properties within the **Engineering Data** module by adding the “ACCS Cure Simulation” solution.

1. In the main menu bar, go to the “ANSYS Composite Cure Simulation” tab and click in “Add Cure Simulation”.

2. In the main menu go to the Solution tab and then add a “Temperature” result.



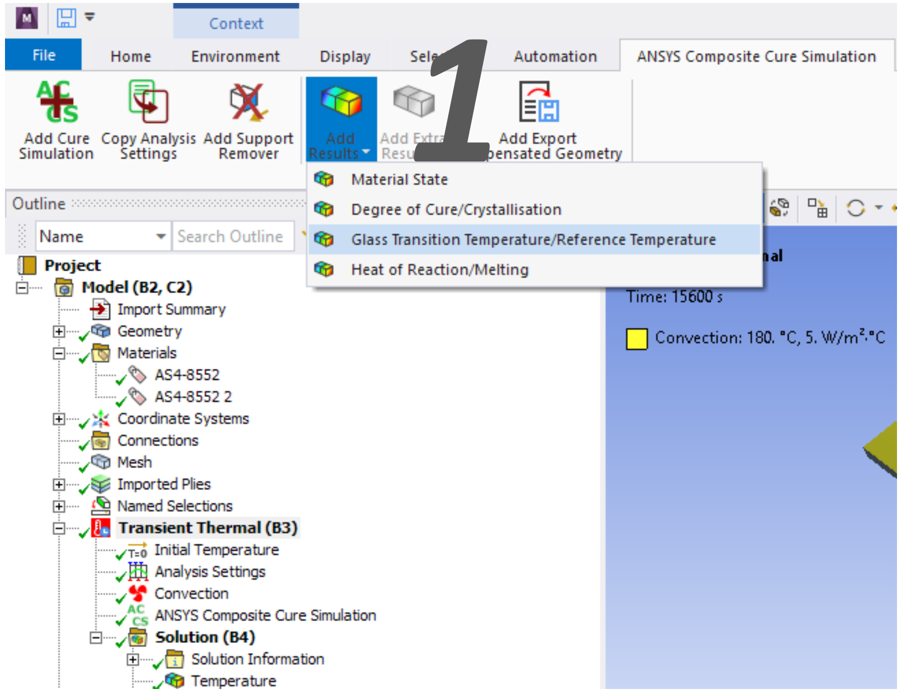
1

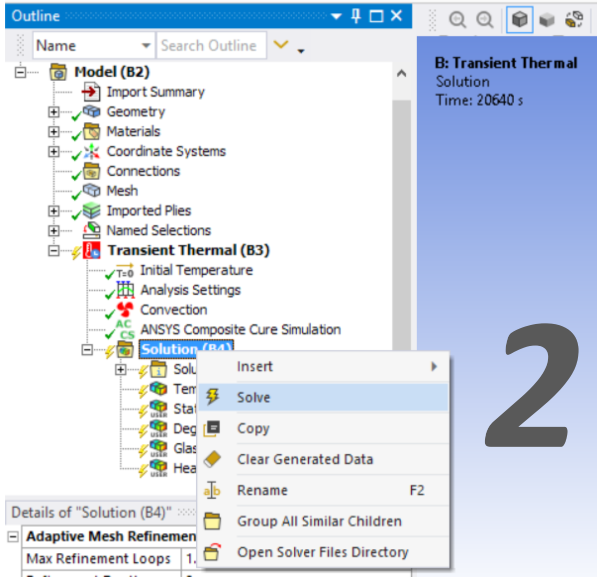


2

Workshop 01: C-Shape Profile Full Cure Simulation
Adding the ACCS Results

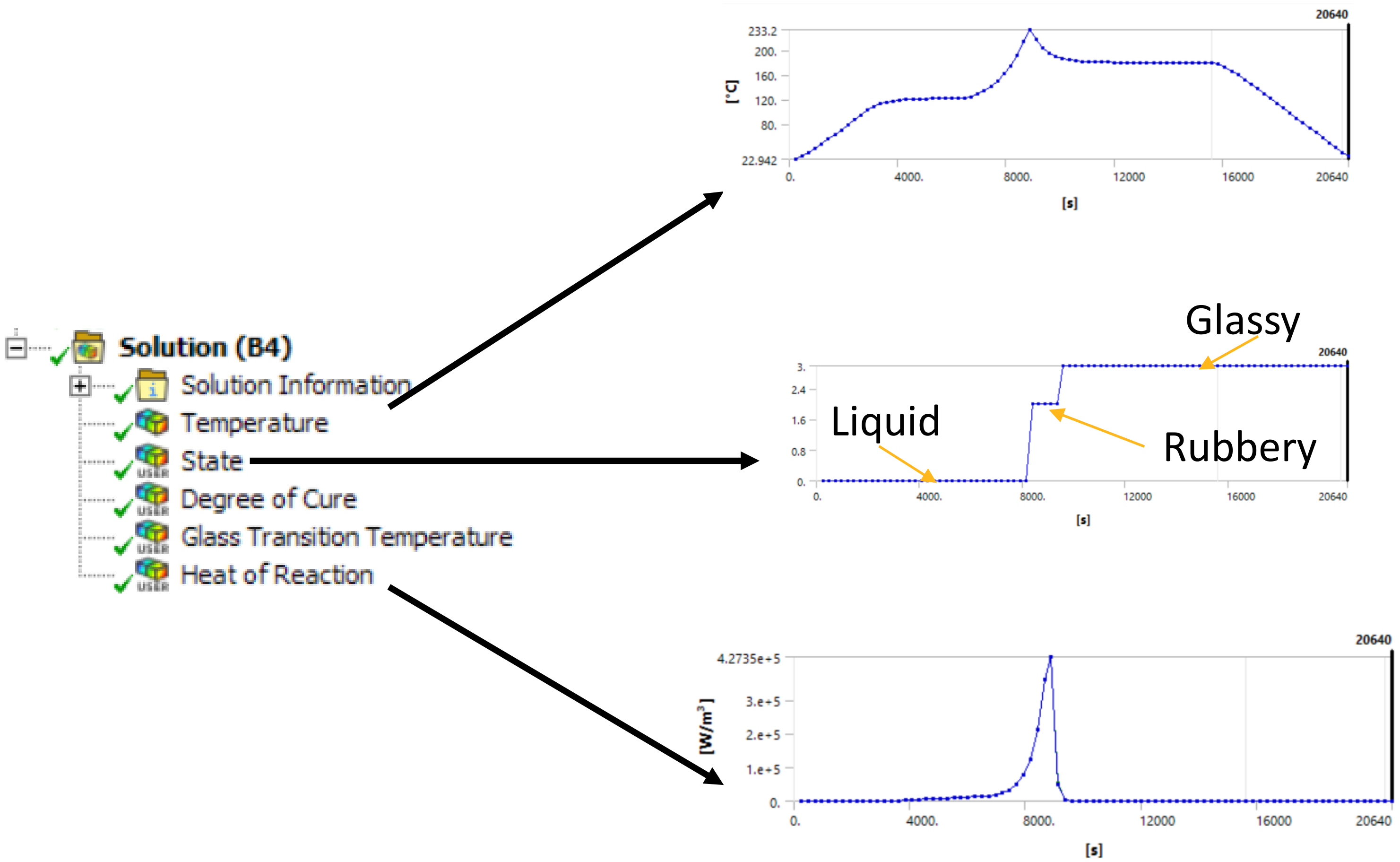
1. Add the ANSYS Composite Cure Simulation results: Material State, Degree of Cure, Glass Transition Temperature, Heat of Reaction.
2. Run the simulation by clicking the solve button.





Workshop 01: C-Shape Profile Full Cure Simulation
Thermal Results

- Once the model is solved, you can review the results by clicking on them. Observe the exothermal reaction in the temperature profile. In many cases, though, the exothermic reaction represents a risk to the process. Excessive heat can cause inhomogeneous cure (check the Degree of Cure Result) and can damage the structural materials (mainly the resin itself, but also polymeric sandwich cores; or heat-sensitive fibers, such as natural fibers), as well as the auxiliary materials (e.g. the vacuum bag).



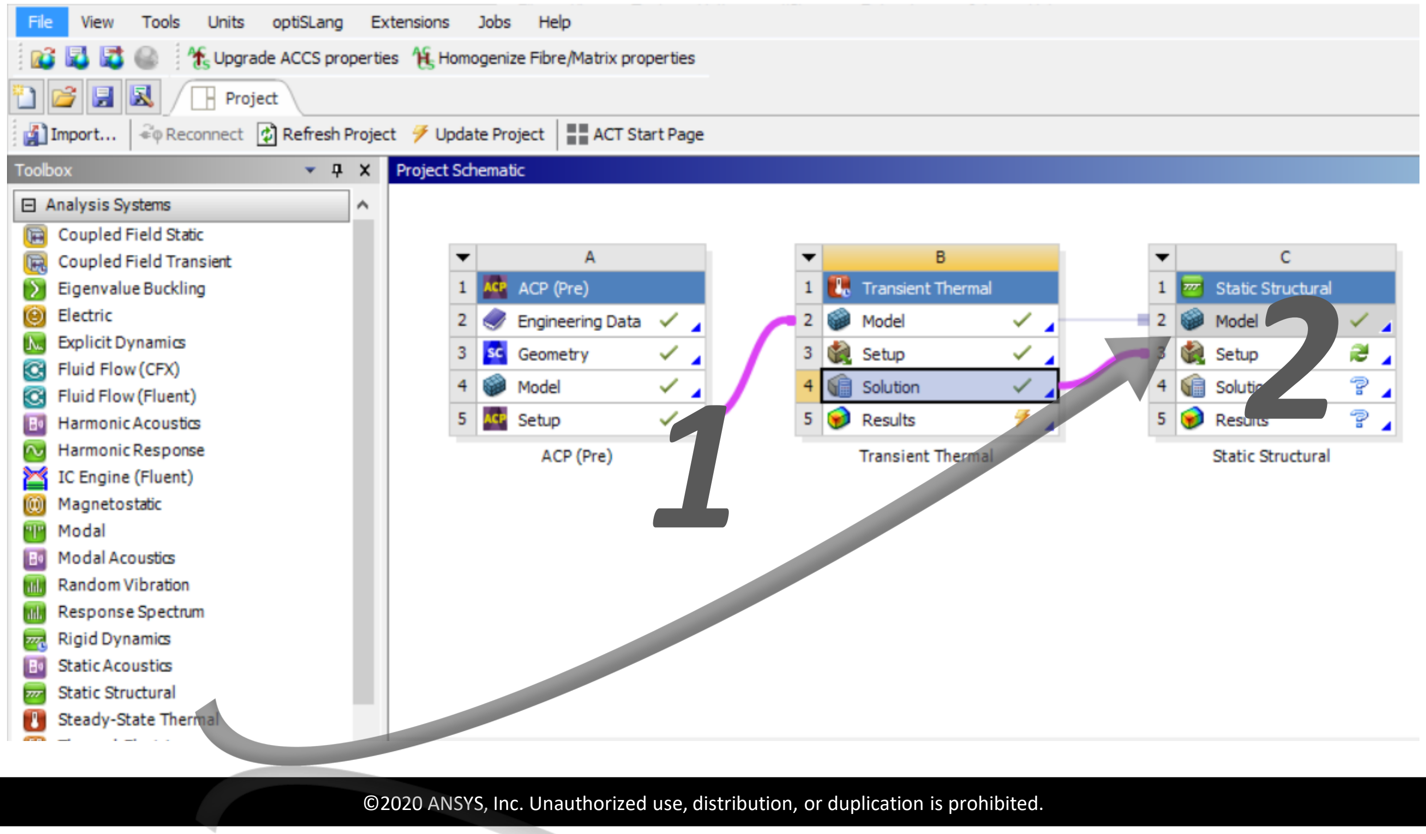
Workshop

Setting up the Static Structural

Workshop 01: C-Shape Profile Full Cure Simulation
Adding the Static Structural Analysis

To analyze the deformations of the part due to the residual stresses of the curing process, it is necessary to add a Static Structural Analysis System and couple it to the thermal analysis.

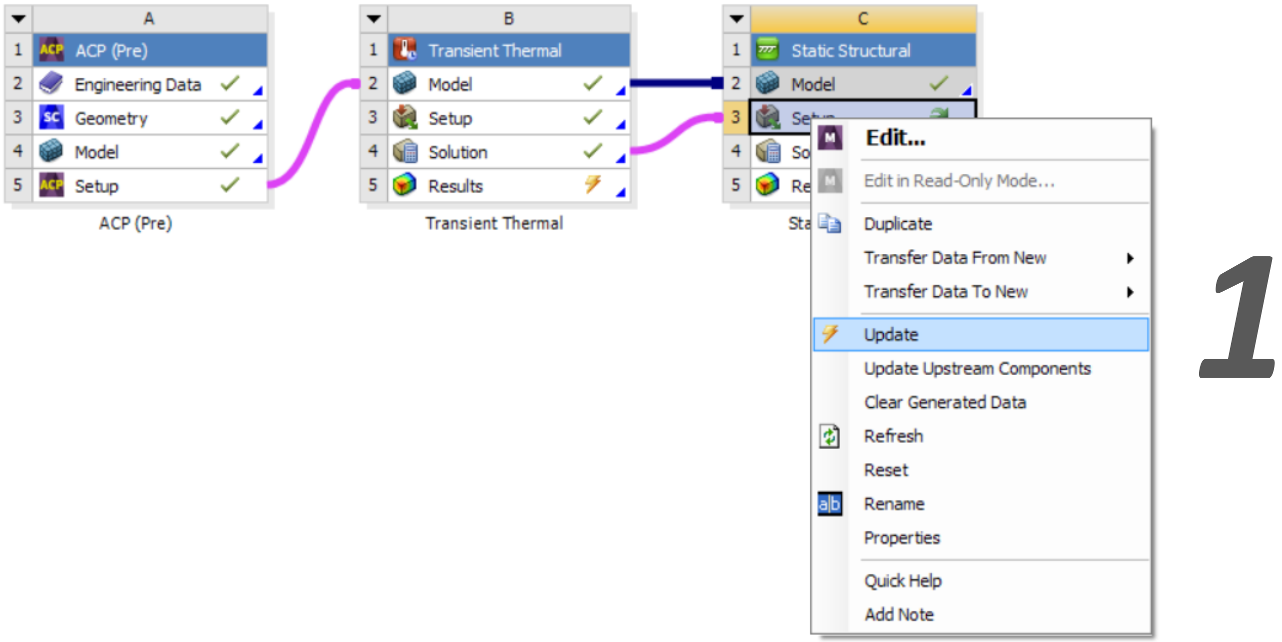
1. Drag and drop a “Static Structural” to the “Model” cell (B2). A blue connection line should be shown.
2. Connect the “Solution” cell (B4) of the Transient Thermal system to the “Setup” cell of the Static Structural system (C3)



Workshop 01: C-Shape Profile Full Cure Simulation

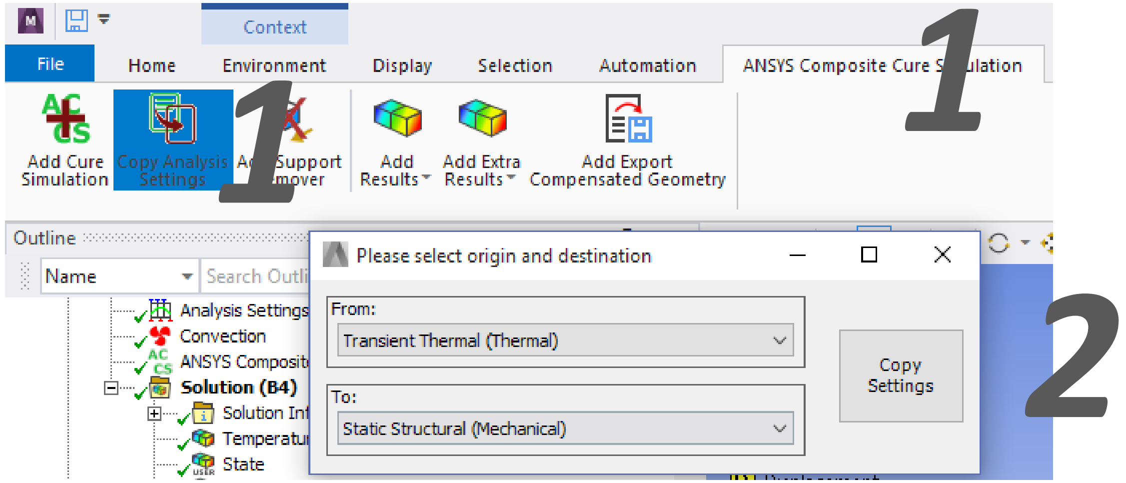
Updating the Static Structural Analysis

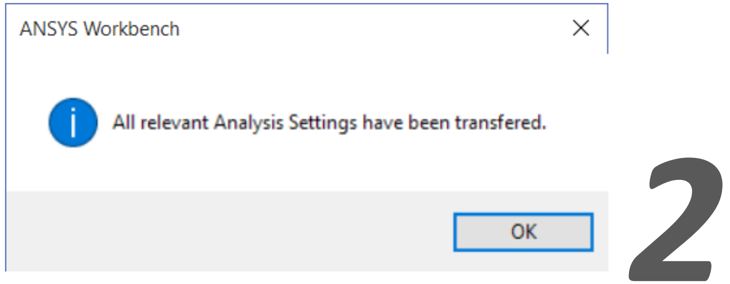
1. RMB in the “Model” cell (C2) of the Static Structural System and select “Update”.
2. Open the “Setup” cell by double clicking in the C3 cell.



Workshop 01: C-Shape Profile Full Cure Simulation
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.



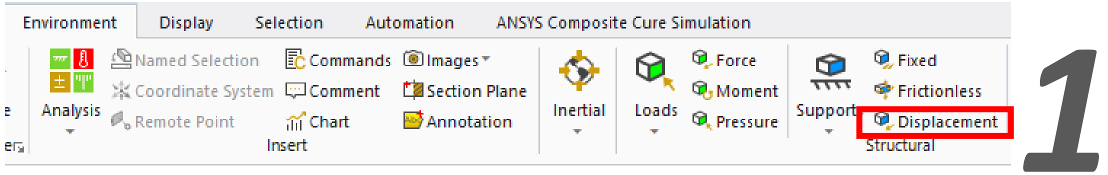


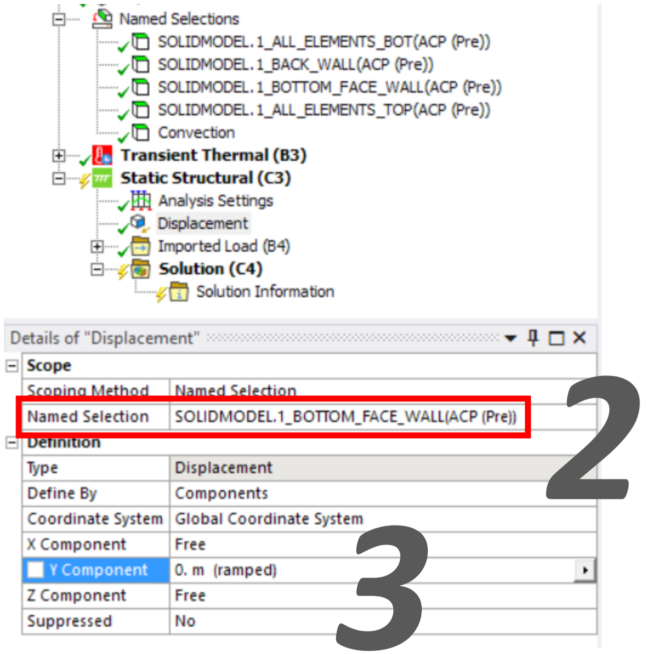
Workshop 01: C-Shape Profile Full Cure Simulation
Adding Boundary Conditions

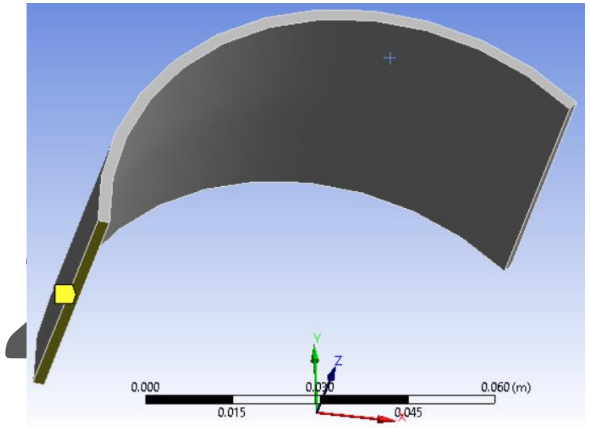
Now the degrees of freedom (DOFs) should be defined according to the process. The part is fist laminated in a mold and then released from it.

Fix the displacement in Y direction:

1. Add a “Displacement” condition.
2. Scope the Named Selection named SOLIDMODEL.1_BOTTOM_FACE_WALL
3. Specify the “Y Component” to 0.



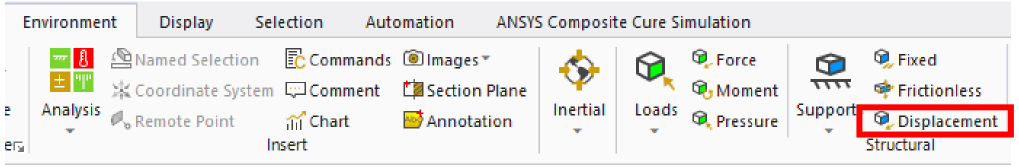




Workshop 01: C-Shape Profile Full Cure Simulation
Adding Boundary Conditions

Fix the displacement in Z direction:

1. Add another “Displacement” condition.
2. Select the Named Selection named SOLIDMODEL.1_BACK_WALL
3. Specify the “Z Component” to 0.



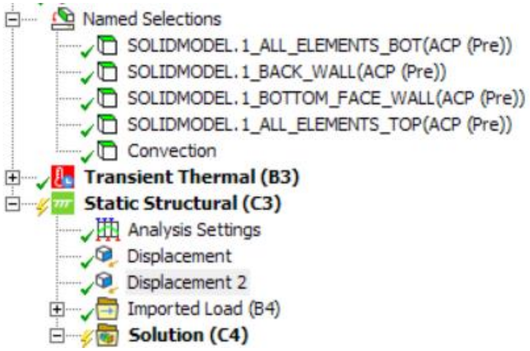
1

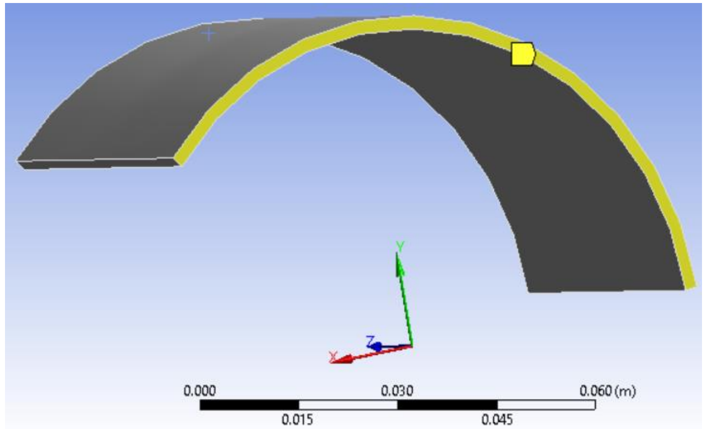
Details of "Displacement 2"

Scope	
Scoping Method	Named Selection
Named Selection	SOLIDMODEL.1_BACK_WALL(ACP (Pre))
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
Y Component	Free
Z Component	0. m (ramped)
Suppressed	No

2

3

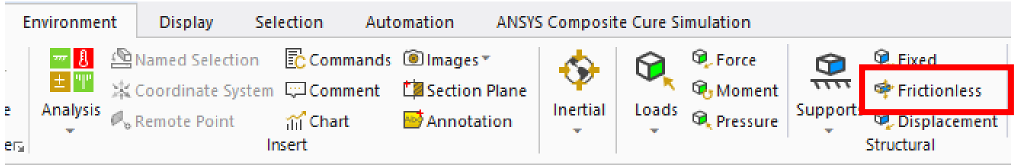




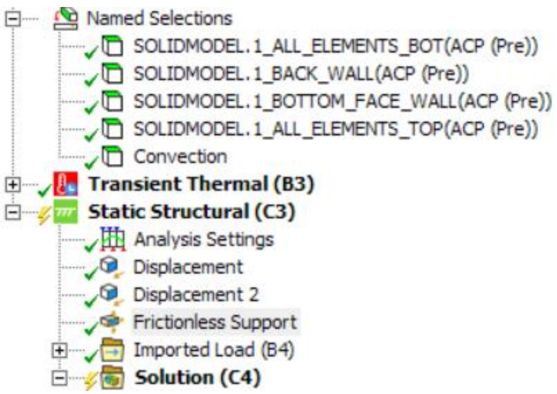
Workshop 01: C-Shape Profile Full Cure Simulation
Adding Boundary Conditions

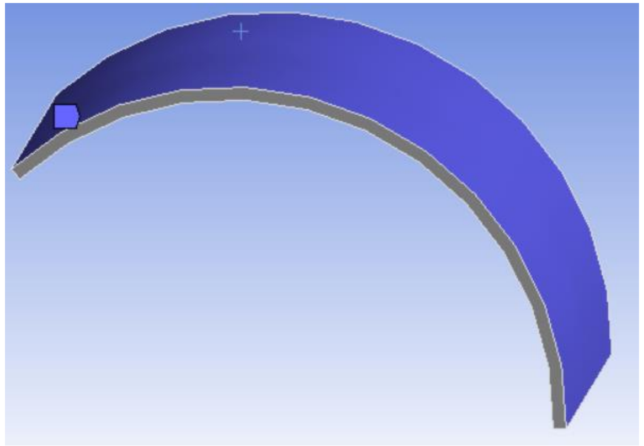
Now we are defining the tool surface. Since the tool surface has a special treatment to enable an easy extraction of the part, the best choice of BC is a frictionless support:

1. Add a “Frictionless Support” condition
2. Select the Named Selection named SOLIDMODEL.1_ALL_ELEMENTS_BOT



1





Details of "Frictionless Support"

Scope	
Scoping Method	Named Selection
Named Selection	SOLIDMODEL.1_ALL_ELEMENTS_BOT(ACP (Pre))
Definition	
Type	Frictionless Support
Suppressed	No

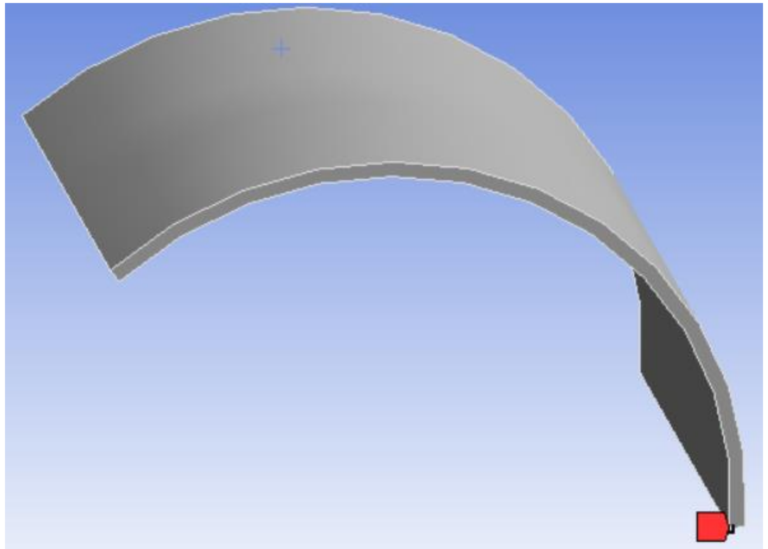
2

Workshop 01: C-Shape Profile Full Cure Simulation

Adding Boundary Conditions

Now all DOF are selected, but once the part is removed from the mold in the last step, the x direction will be undefined, so it is necessary to define it in such way that when the part can deform freely for the last step and at the same time restrict the DOF. That’s why the bottom vertex should be scoped as follow:

1. Add a Named Selection based in Worksheet
2. specify the vertex at the bottom as shown in the worksheet and name it “Vertex_Bottom”



Named Selections
- SOLIDMODEL.1_ALL_ELEMENTS_BOT(ACP (Pre))
- SOLIDMODEL.1_BACK_WALL(ACP (Pre))
- SOLIDMODEL.1_BOTTOM_FACE_WALL(ACP (Pre))
- SOLIDMODEL.1_ALL_ELEMENTS_TOP(ACP (Pre))
- Convection
- Vertex_Bottom

2

Details of "Vertex_Bottom"

Scope	
Scoping Method	Worksheet
Geometry	1 Vertex
Definition	
Send to Solver	Yes
Protected	Program Controlled
Visible	Yes
Program Controlled Inflation	Exclude
Statistics	
Type	Manual
Total Selection	1 Vertex
Suppressed	0
Used by Mesh Worksheet	No

1

Worksheet

Vertex_Bottom

Generate

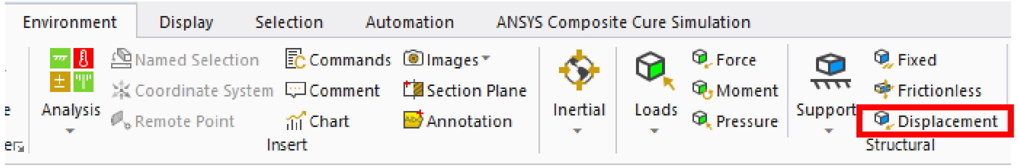
2

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)

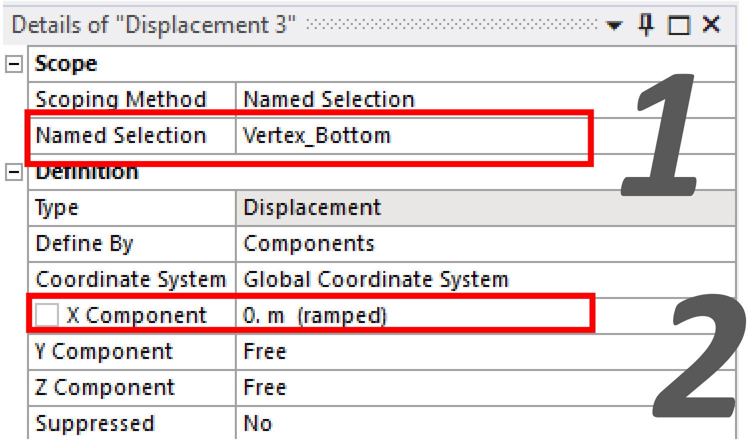
	Action	Entity Type	Criterion	Operator	Units	Value	Lower Bound	Upper Bound	Coordinate System
☑	Add	Vertex	Location X	Equal	m	-4.8e-002	N/A	N/A	Global Coordinate System
☑	Filter	Vertex	Location Z	Range	m	N/A	0.	1.e-003	Global Coordinate System

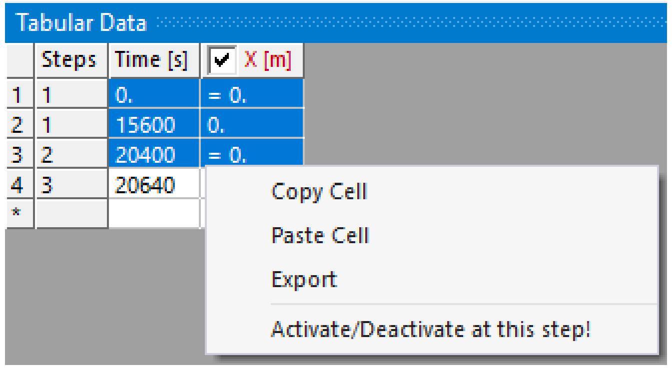
Workshop 01: C-Shape Profile Full Cure Simulation
Adding the ACCS Solver Routines

1. Add a "Displacement" and scope it to the "Vertex_Bottom"
2. Specify the X component to "0".
3. Open the Tabular Data tab, select the first three lines, press the right click button, and select "Activate/Deactivate at this step!".



1

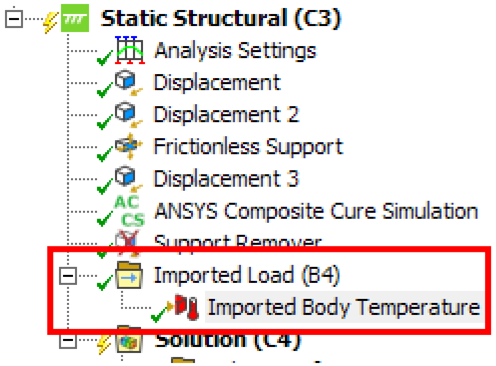




3

Workshop 01: C-Shape Profile Full Cure Simulation
Synchronizing the Imported Load

1. In the tree, open the Imported Load folder
2. In the Source Time, select “All” instead of “Worksheet”. This is necessary for correctly importing the thermal load to the static structural system.



1

Details of "Imported Body Temperature"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Type	Imported Body Temperature
Tabular Loading	Program Controlled
Suppressed	No
Source Environment	Transient Thermal (B3)
Source Time	All
Graphics Controls	
By	Active Row
Active Row	1

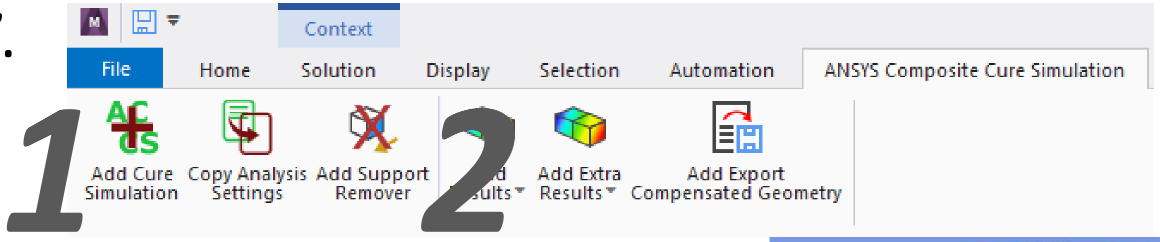
2

3

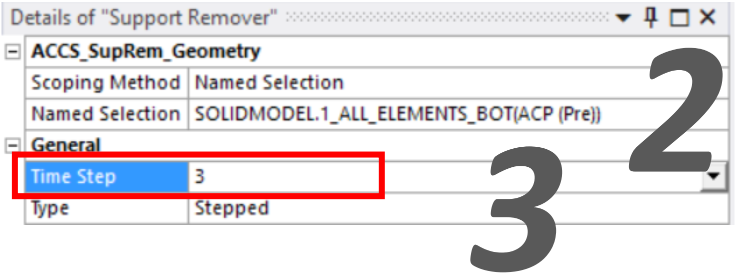
Workshop 01: C-Shape Profile Full Cure Simulation
Adding the ACCS Solver Routines

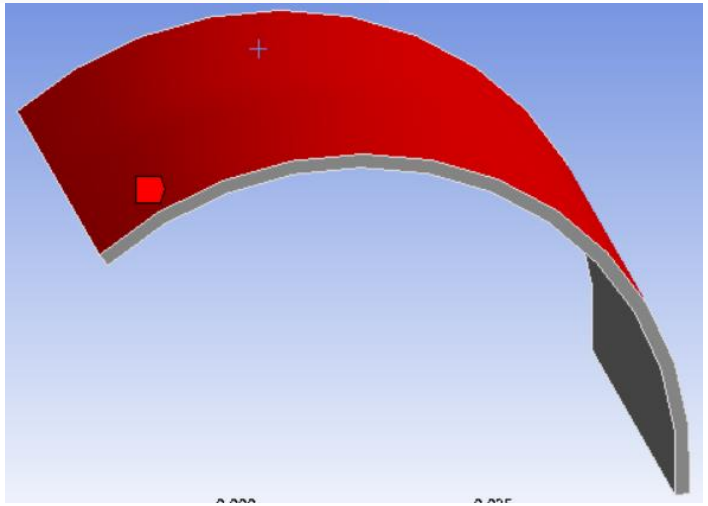
Now we are adding the ACCS material models to calculate the effect of the curing in the final shape of the part.

1. Add the "ACCS" module and make sure "Full" is selected in "Analysis Type".
2. Add a "Support Remover" condition on the Named Selection "ALL_ELEMENTS_BOT"
3. set the "Time Step" to "3".



Note: the Support Remover will remove all the BC that are defined in the nodes of the mapped region. This is important when more BCs are defined on the same nodes; the model might become structurally undefined.

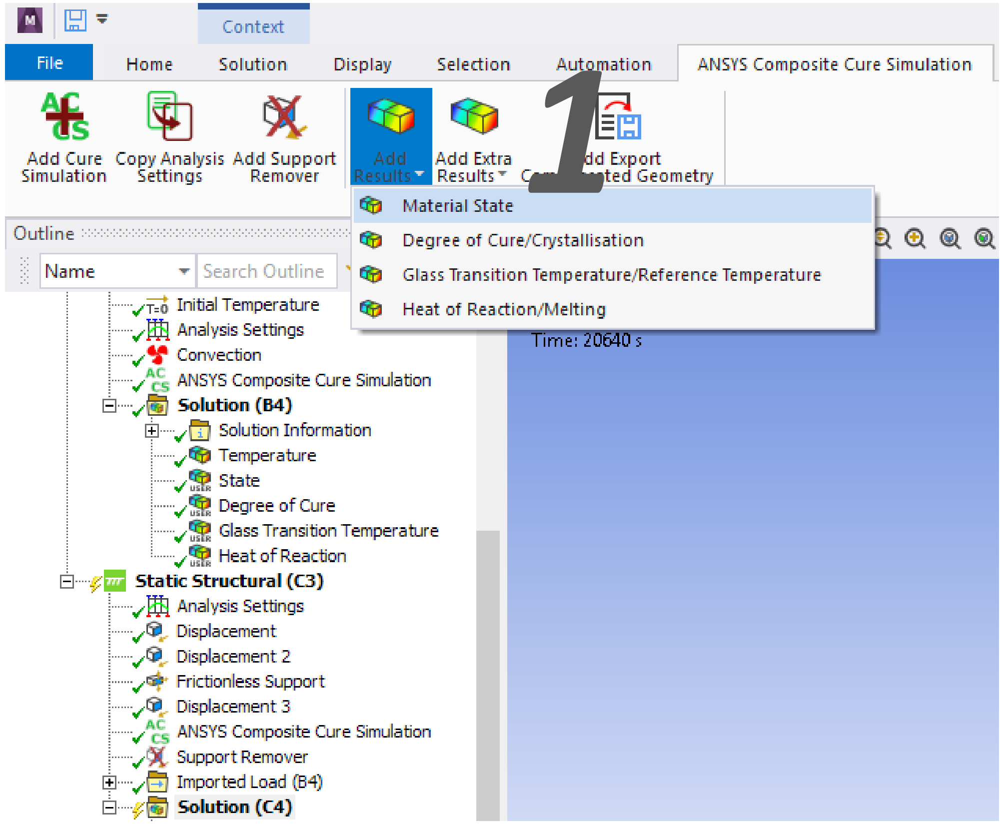


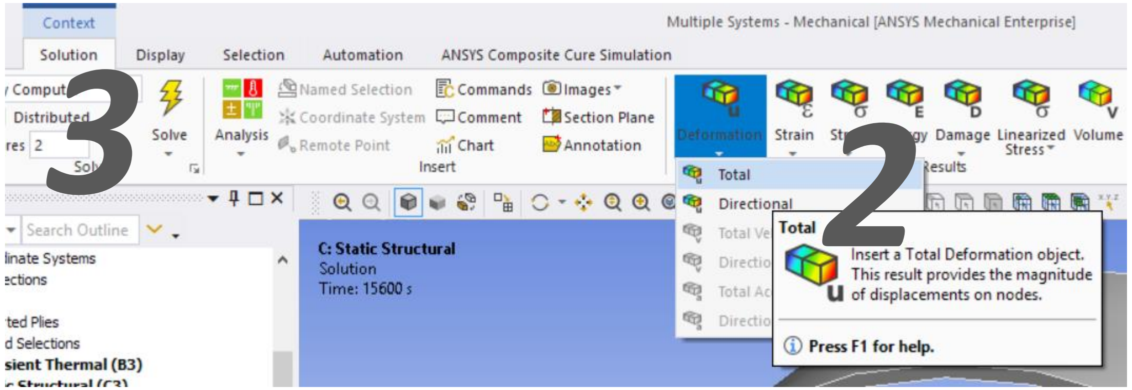


Workshop 01: C-Shape Profile Full Cure Simulation

Adding the ACCS Results

1. Add the ACCS results: Material State, Degree of Cure, Glass Transition Temperature, Heat of Reaction. result.
2. Add also a “Total Deformation”
3. Run the simulation by clicking the "Solve" button.



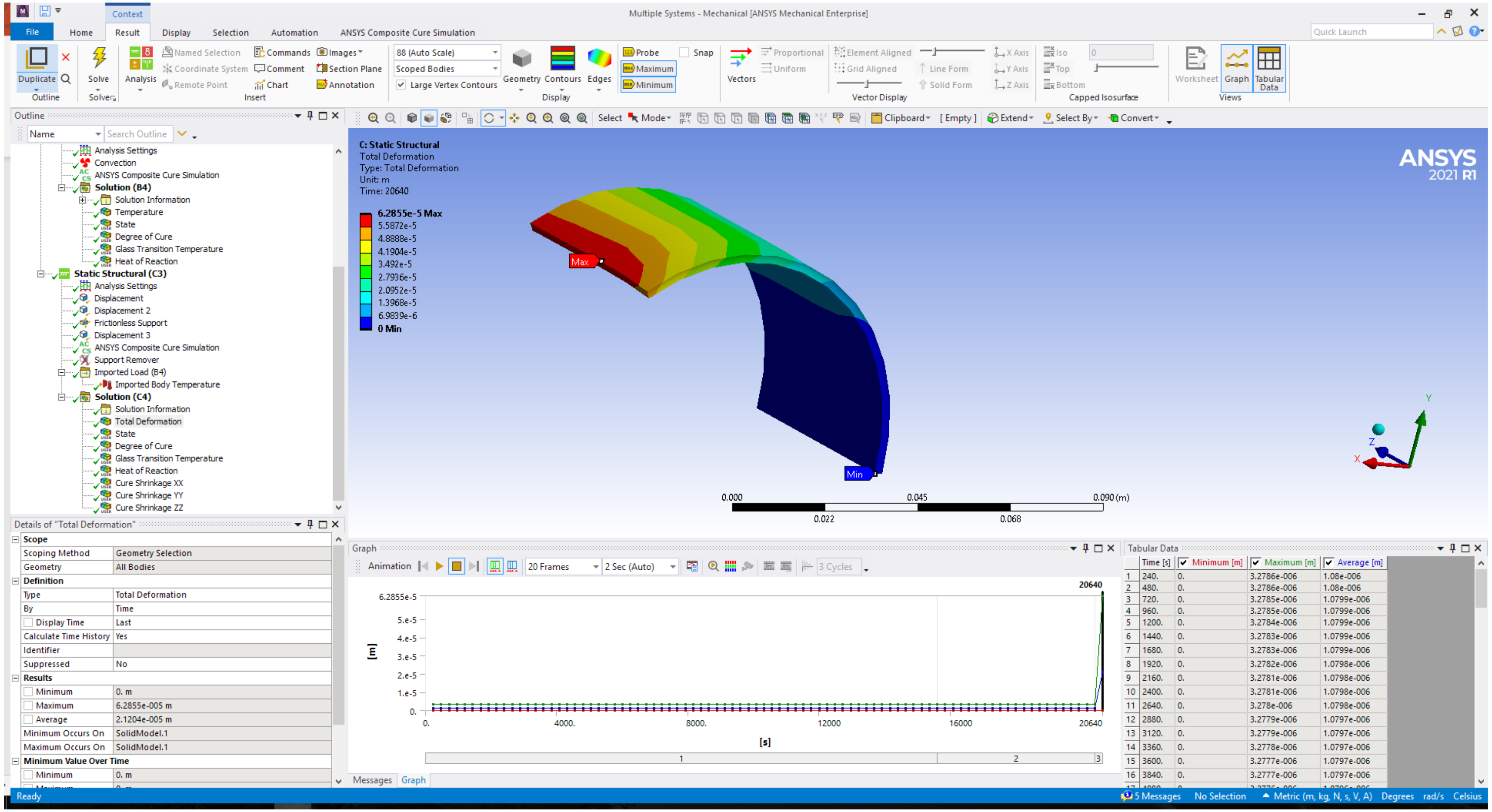


Workshop

Result Analysis

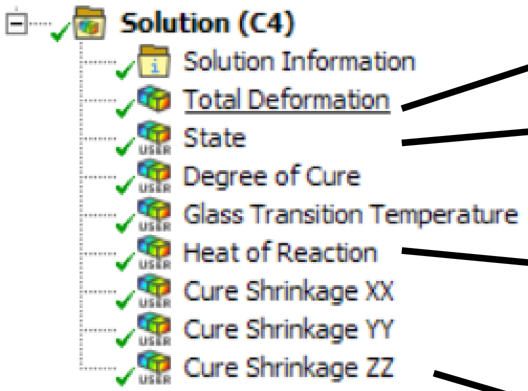
Result Analysis

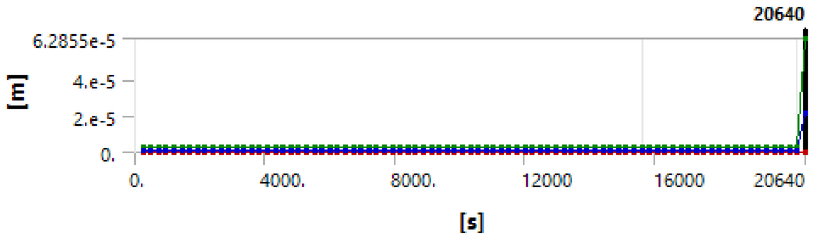
As you may noticed, the deformations occurred in the last step, when the part is released from the mold. You can try to change the curing parameters to see what has a mayor effect in the process induced distortions (PID).

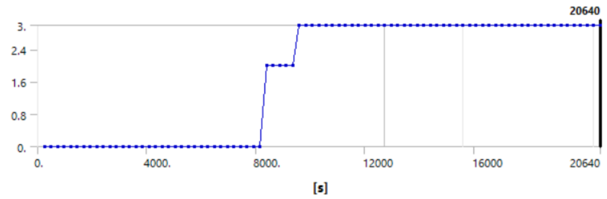


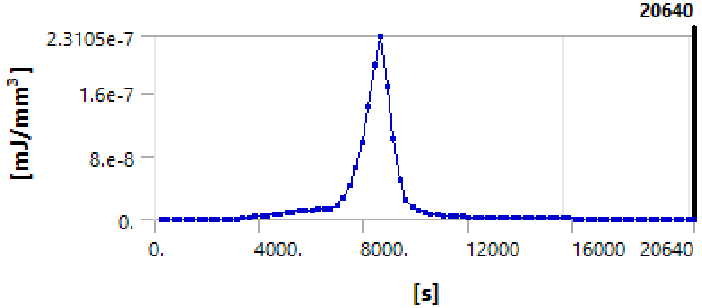
Result Analysis

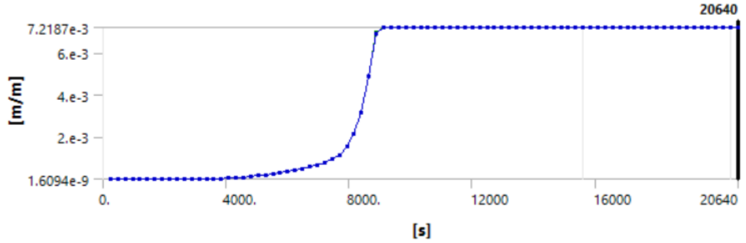
In the Static Structural Solution tree, both thermal and structural can be viewed











One can see that the heat of reaction peaks when the phase change occurs. The cure shrinkage is significant in the transition between the rubbery (state 2) and glassy (state 3) states.

Summary

Summary

In this workshop you learnt …

- How to analyze the thermal behavior of the curing process in a composite model built in ACP (Pre)
- That ACCS solutions use many material properties that can be obtained by a DSC test.
- To set the curing cycle and the time stepping.
- To properly connect a Static Structural Analysis System to the Transient Thermal System, in order to analyze the distortions induced by the heating cycle.
- To define the DOFs so the part is free to deform once it is released from the mold.
- The PID, exotherms, Material State, Degree of Cure, Glass Transition Temperature and Heat of reaction of an epoxy composite structure can be simulated using the ACCS module.