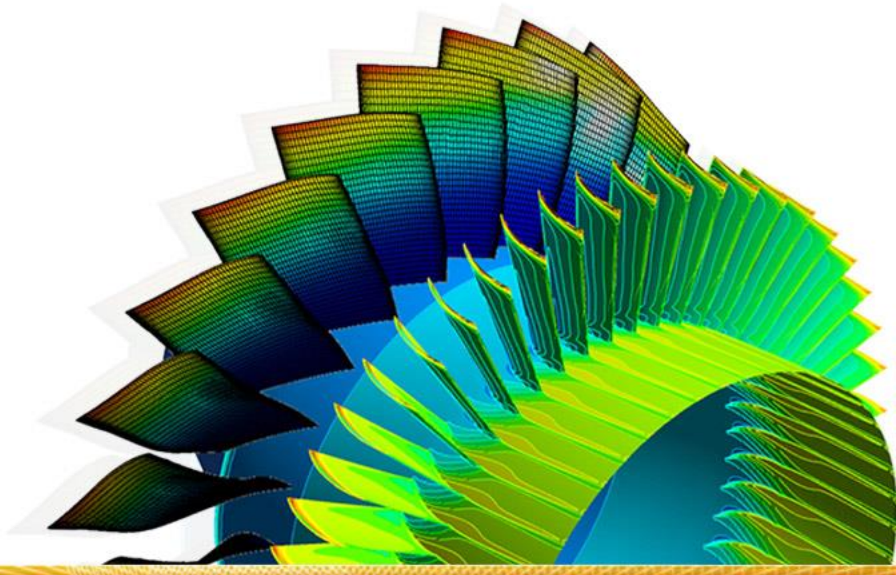
ANSYS Composite PrepPost 19.0

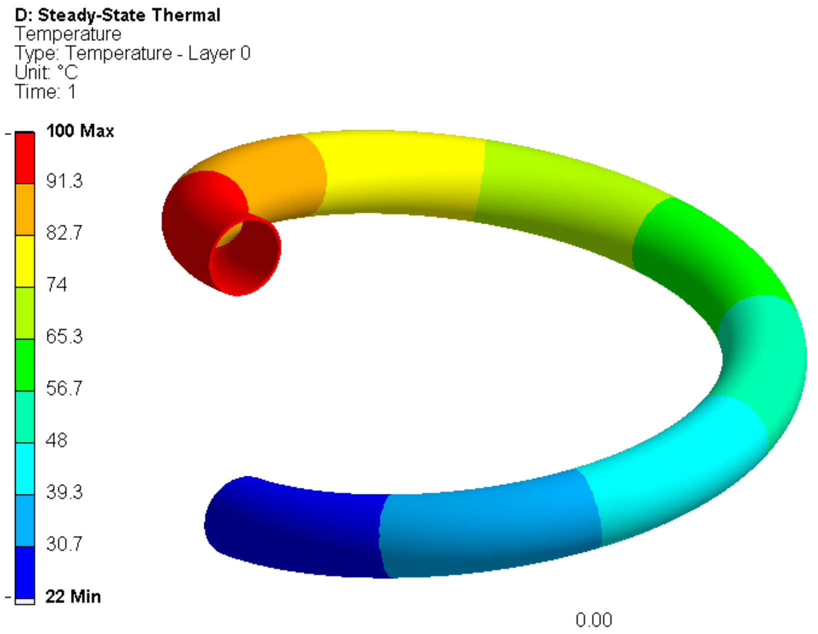
Workshop 10.5 – Temperature Dependent Material

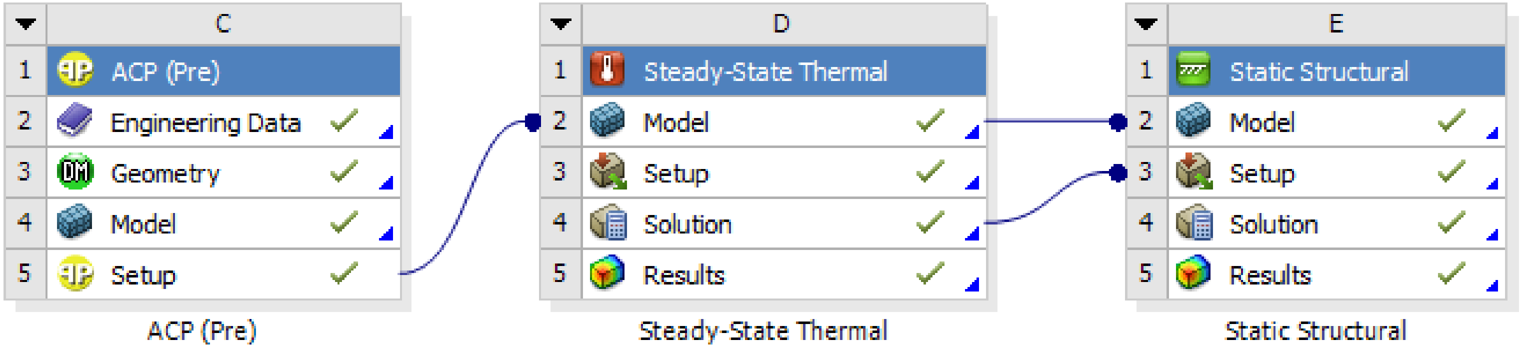


14. Workshop, Temperature Dependent Material

Agenda

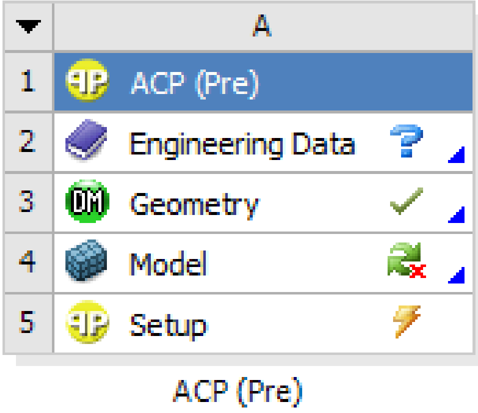
- Define Temperature dependent Material inside engineering data
- Set up thermal analysis and solve for temperature field
- Solve static model using temperature load





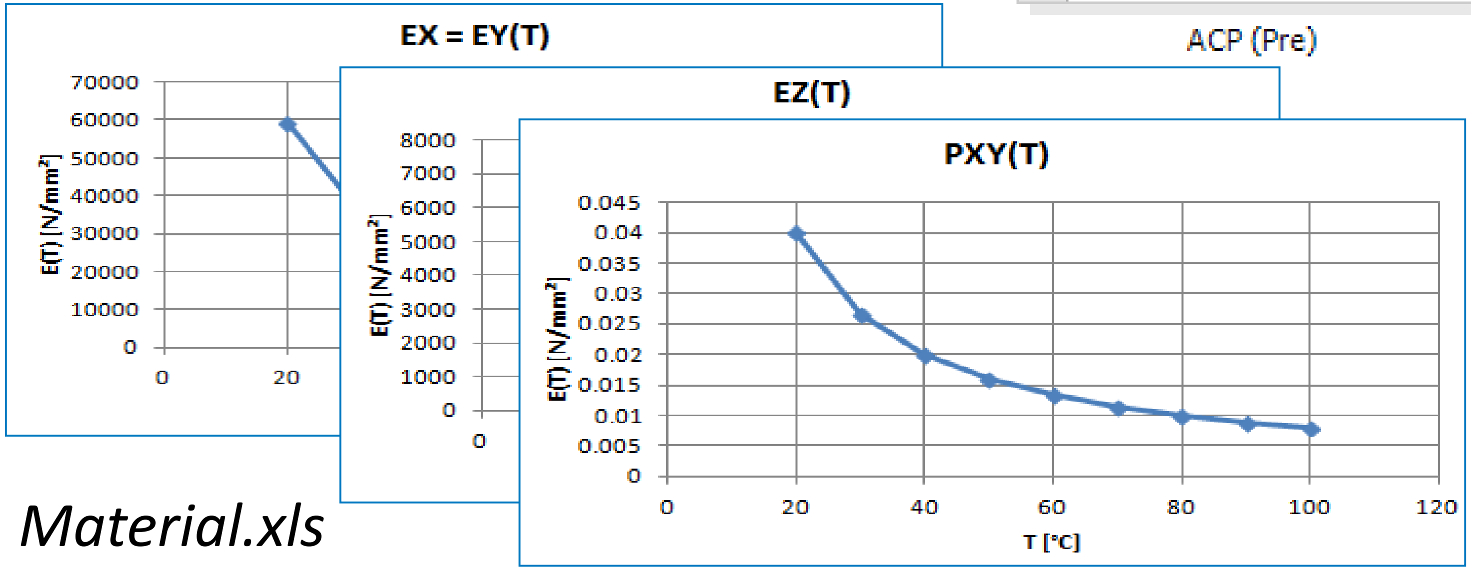
14. Workshop, Temperature Dependent Material

- Open workbench archive
  temperature_dependent_material_FROM_START_19.0.wbpz



- Open engineering data and from file menu import prepared material model
  Material_Temp_Dep.xml

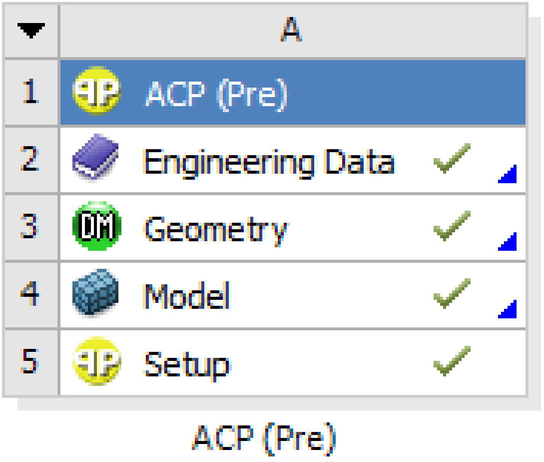
- This file contains a woven material with a temperature dependent stiffness



Material.xls

14. Workshop, Temperature Dependent Material

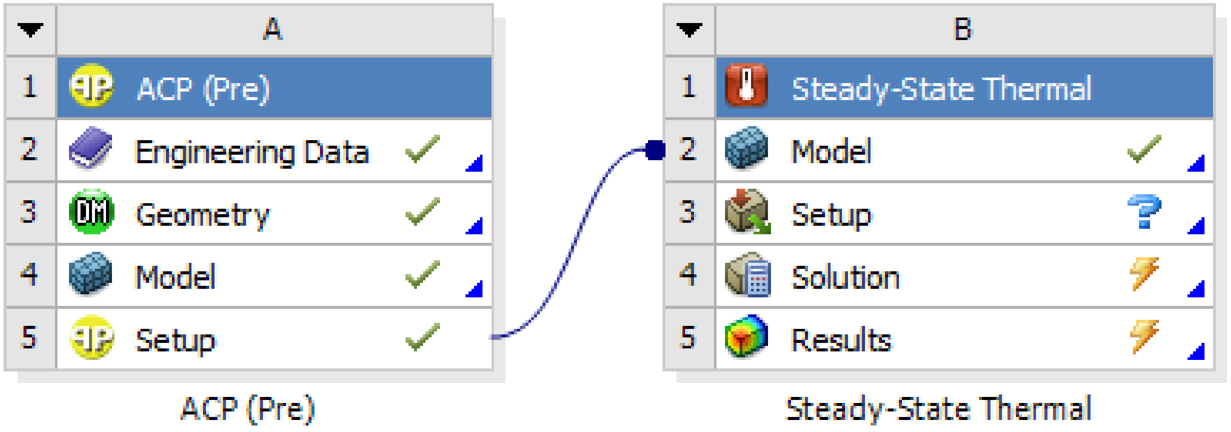
- Close engineering data and update / refresh project schematic



- Drag an *Steady-State Thermal* analysis system from toolbox onto project schematic.

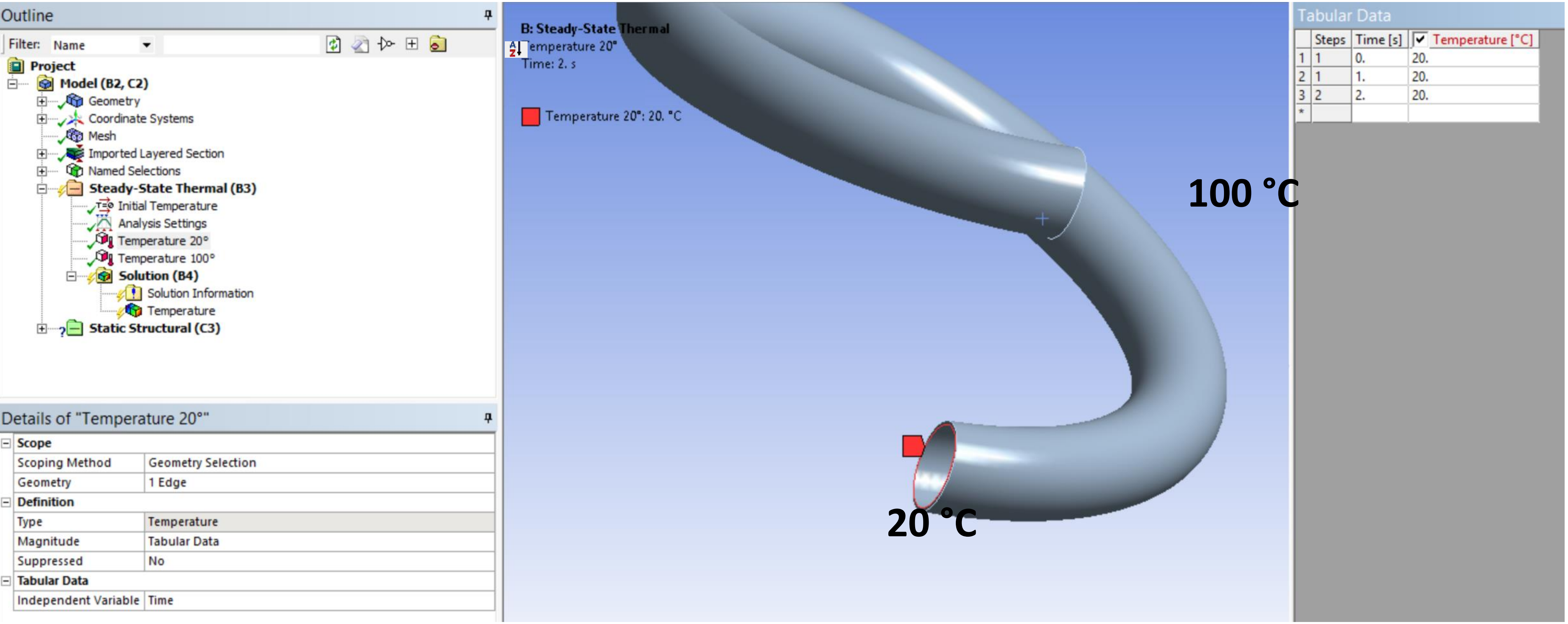
- Link the Setup Cell from ACP-Pre with the Model cell from *Steady-State Thermal*

- Update and refresh project schematic.
 A Solid Composite model will be generated and transferred



14. Workshop, Temperature Dependent Material

Open the thermal model and define 2 Load Steps in Analysis Systems:



	Steps	Time [s]	Temperature [°C]
1	1	0.	20.
2	1	1.	20.
3	2	2.	20.
*			

14. Workshop, Temperature Dependent Material

Open the thermal model and define 2 Load Steps in Analysis Systems:

- Apply Temperature Boundary Conditions on the lower and on the upper end as shown

- Lower end:

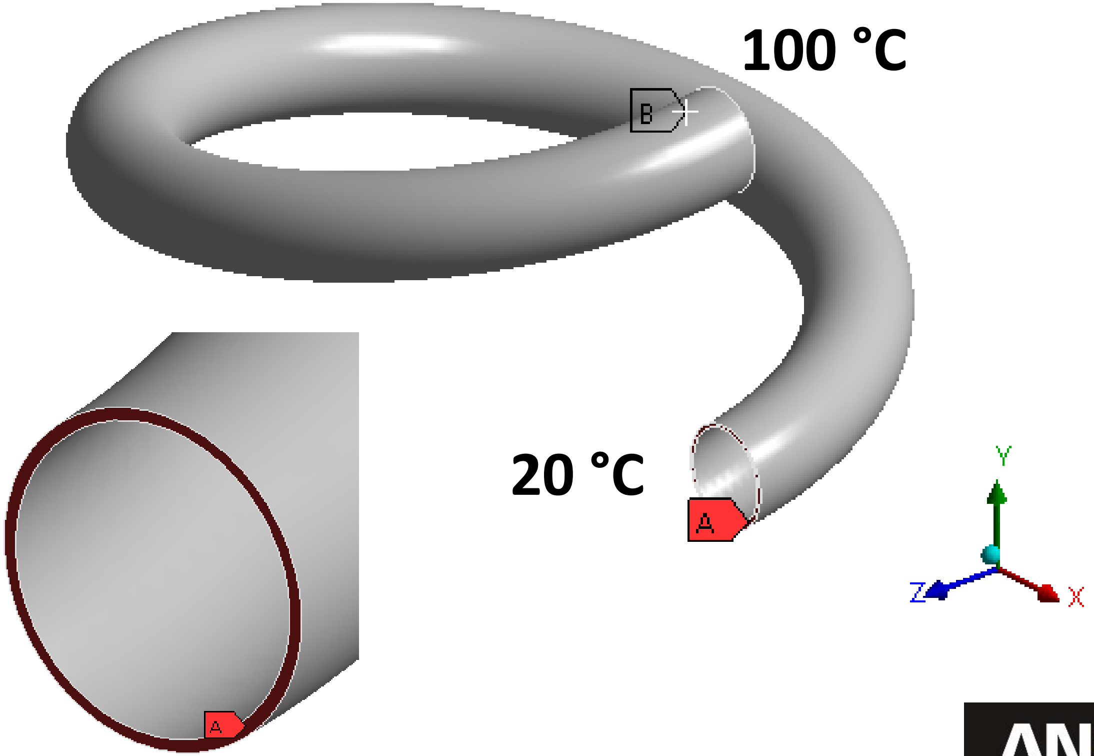
Tabular Data

	Steps	Time [s]	✔ Temperature [°C]
1	1	0.	20.
2	1	1.	20.
3	2	2.	20.

- Upper end:

Tabular Data

	Steps	Time [s]	✔ Temperature [°C]
1	1	0.	20.
2	1	1.	20.
3	2	2.	100.



14. Workshop, Temperature Dependent Material

- With these boundary conditions we hold the Temperature constant at 20°C during the first load step. Afterwards we increase temperature to 100°C at the upper end.

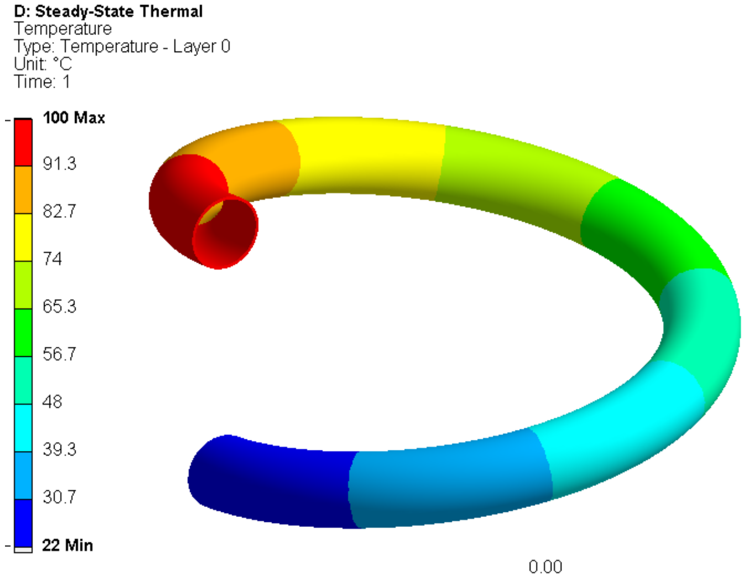
- Go to Analysis Settings and choose time stepping settings as shown below (switch between the different current time steps with the Tabular Data menu in the side of the main view):

Details of "Analysis Settings"

Step Controls	
Number Of Steps	2.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	On
Define By	Substeps
Initial Substeps	5.
Minimum Substeps	5.
Maximum Substeps	10.

Details of "Analysis Settings"

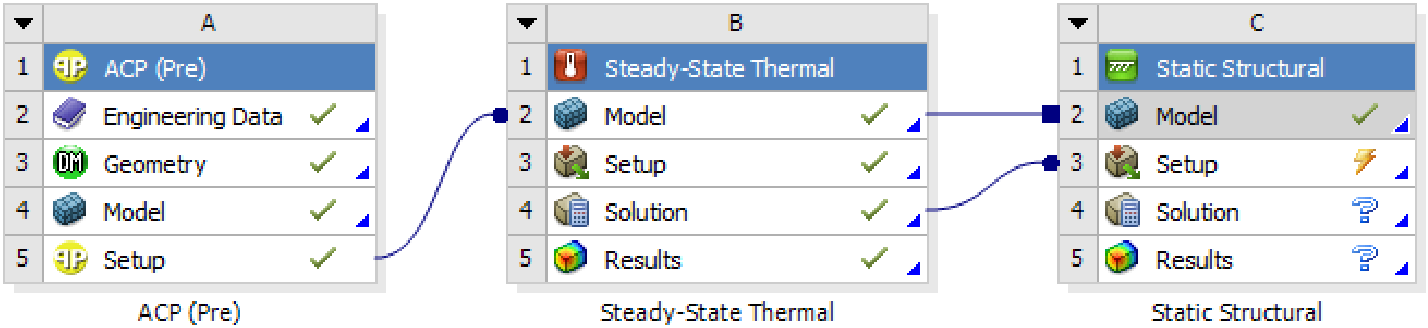
Step Controls	
Number Of Steps	2.
Current Step Number	2.
Step End Time	2. s
Auto Time Stepping	On
Define By	Substeps
Carry Over Time Step	Off
Initial Substeps	5.
Minimum Substeps	5.
Maximum Substeps	10.



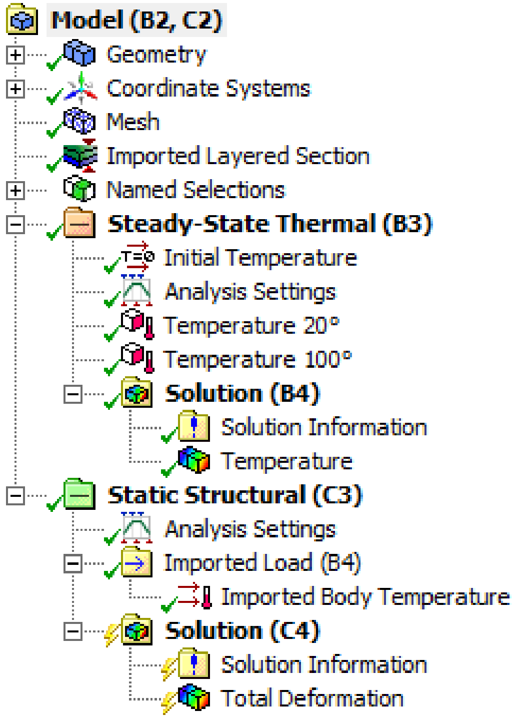
- Solve the model, save and close Mechanical

14. Workshop, Temperature Dependent Material

- Now we will use the calculated temperature fields as a boundary condition for a static structural analysis. Prepare the project as shown



- Open Mechanical to add structural boundary conditions to the model



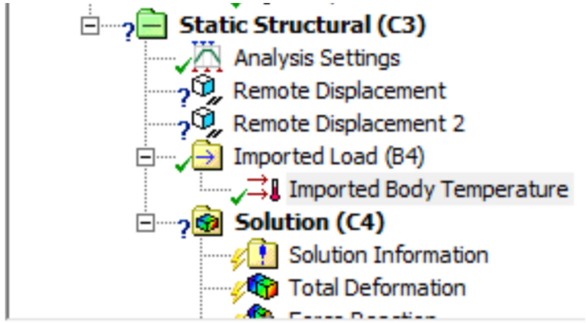
- The prior calculated temperature field is shown as imported load object in tree, *Imported Body Temperature*

- Evaluate imported loads and check temperatures

14. Workshop, Temperature Dependent Material

1) Go to Analysis Settings and define two load steps and substep settings.

2) Afterwards set the time history of the imported temperature load. We will apply the results of the first thermal load step to the first mechanical load step.



Details of "Imported Body Temperature"

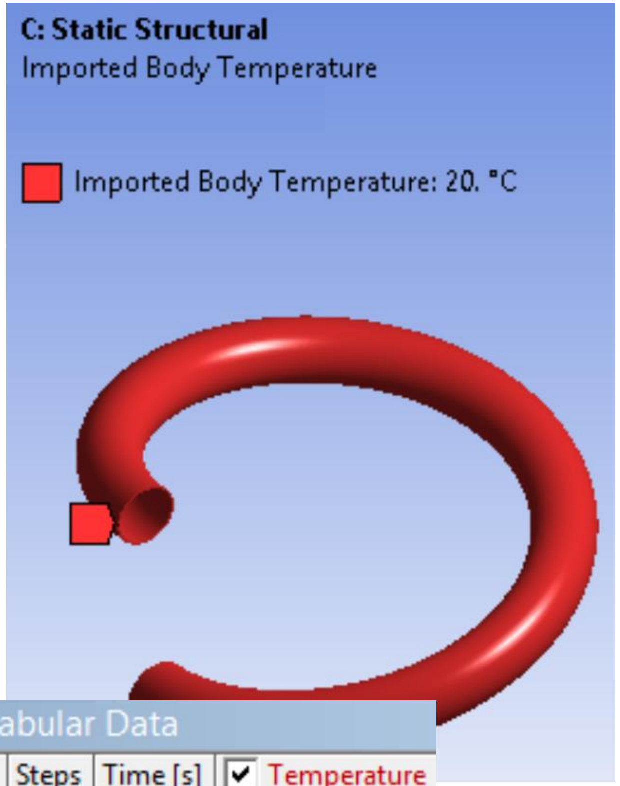
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Type	Imported Body Temperature
Tabular Loading	Program Controlled
Suppressed	No
Source Environment	Steady-State Thermal (B3)
Source Time	Worksheet
Graphics Controls	
By	Active Row
Active Row	1

Details of "Analysis Settings"

Step Controls	
Number Of Steps	2.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	On
Define By	Substeps
Initial Substeps	3.
Minimum Substeps	3.
Maximum Substeps	10.
Solver Controls	
Solver Type	Direct
Weak Springs	Off
Large Deflection	On

Details of "Analysis Settings"

Step Controls	
Number Of Steps	2.
Current Step Number	2.
Step End Time	2. s
Auto Time Stepping	On
Define By	Substeps
Carry Over Time Step	Off
Initial Substeps	5.
Minimum Substeps	5.
Maximum Substeps	10.
Solver Controls	
Solver Type	Direct
Weak Springs	Off
Large Deflection	On



Tabular Data

	Steps	Time [s]	✔ Temperature
1	1	1.	Row_1
2	2	2.	Row_2

14. Workshop, Temperature Dependent Material

- Add a Remote Displacements at each end of the pipe using following setting.

Lower end

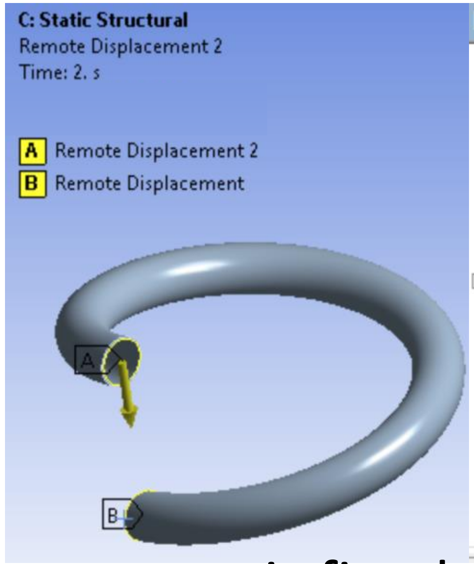
Tabular Data

	Steps	Time [s]	X [mm]	Y [mm]	Z [mm]	RX [°]	RY [°]
1	1	0.	0.	0.	0.	0.	0.
2	1	1.	0.	0.	0.	0.	0.
3	2	2.	= 0.	= 0.	= 0.	= 0.	= 0.

Upper end

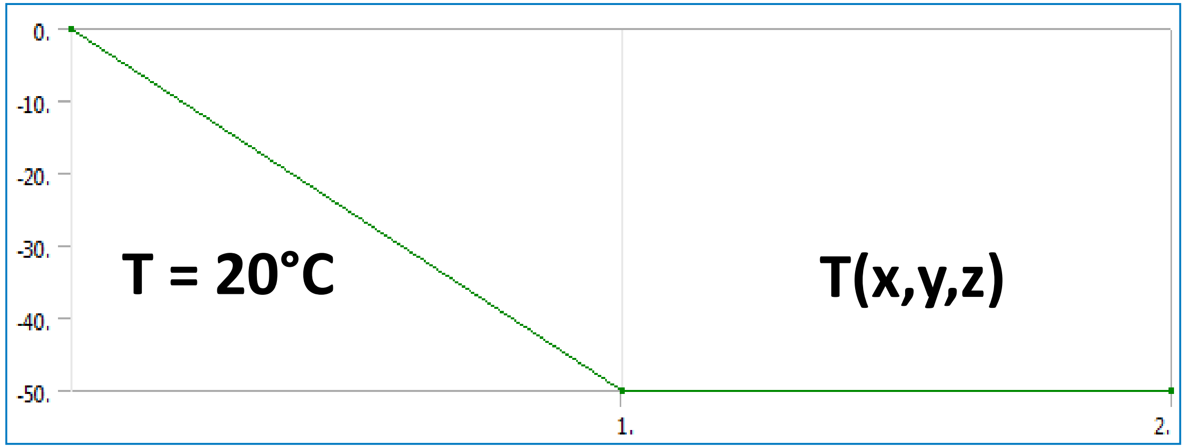
Tabular Data

	Steps	Time [s]	X [mm]	Y [mm]	Z [mm]	RX [°]	RY [°]
1	1	0.	0.	0.	0.	0.	0.
2	1	1.	0.	-50.	0.	0.	0.
3	2	2.	= 0.	-50.	= 0.	= 0.	= 0.



Details of "Remote Displacement"

Scoping Method	Geometry Selection
Geometry	1 Edge
Coordinate System	Global Coordinate System
X Coordinate	99.999 mm
Y Coordinate	9.2157e-005 mm
Z Coordinate	5.1659e-005 mm
Location	Click to Change
Definition	
Type	Remote Displacement
X Component	0. mm (ramped)
Y Component	0. mm (ramped)
Z Component	0. mm (ramped)
Rotation X	0. ° (ramped)
Rotation Y	0. ° (ramped)
Rotation Z	Free
Suppressed	No



- The structure is first loaded at 20°C with the temperature load applied in the second load step.

14. Workshop, Temperature Dependent Material

- Solve the model and observe reaction forces in Post Processing. When Temperature load is applied, the reaction force reduces according to the temperature dependent stiffness defined in material model

Details of "Force Reaction"

Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Remote Displacement 2
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	All
Display Time	5.3476e-002 s
Results	
Maximum Value Over Time	
X Axis	4.7999e-002 N
Y Axis	-466.57 N
Z Axis	18.625 N
Total	1415.5 N
Minimum Value Over Time	
X Axis	-0.28762 N
Y Axis	-1415.5 N
Z Axis	-20.402 N
Total	467.02 N