

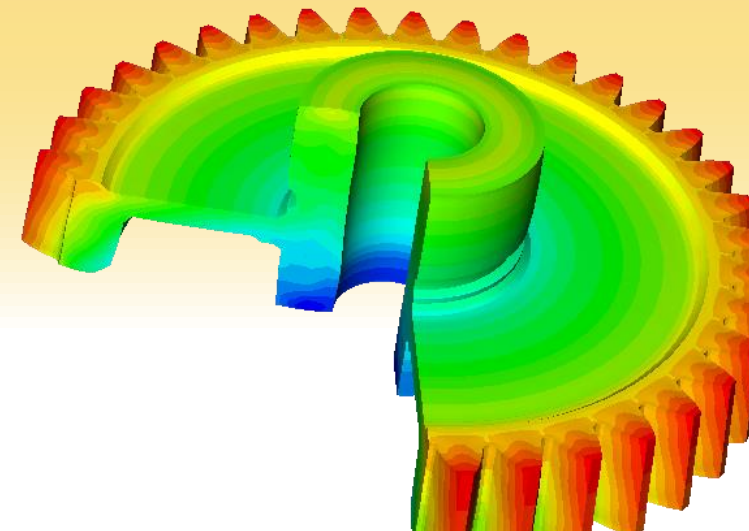
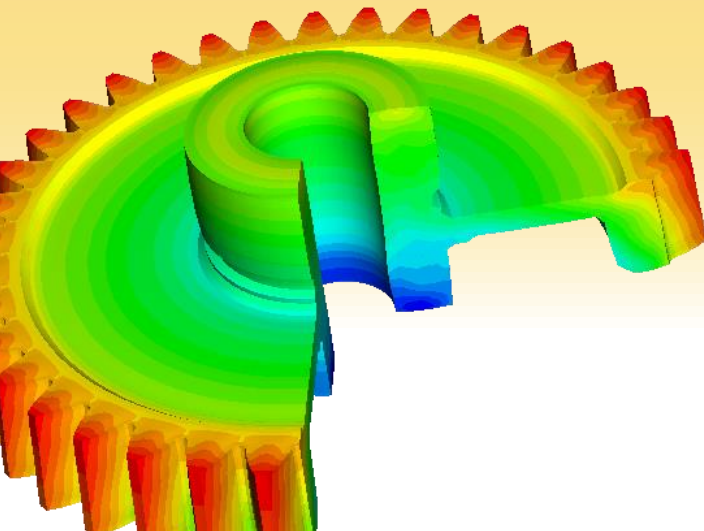
# Dante 5.0 Tutorial

## Coupled ANSYS DANTE Quench Hardening Model of Steel Ring Component

Prepared By

DANTE Solutions, Inc.

Cleveland Oh



## Background

- This workshop will demonstrate the quench hardening modeling coupled between ANSYS R16.1 (or later) and DANTE using a simple sliced ring model
- This workshop includes sequentially coupled carburization, thermal and stress models

## Objectives

- Setting up heat treatment models
- Using ANSYS ACT for DANTE models
- Post-processing modeling results

## Material: AISI 8620 (Carburized)

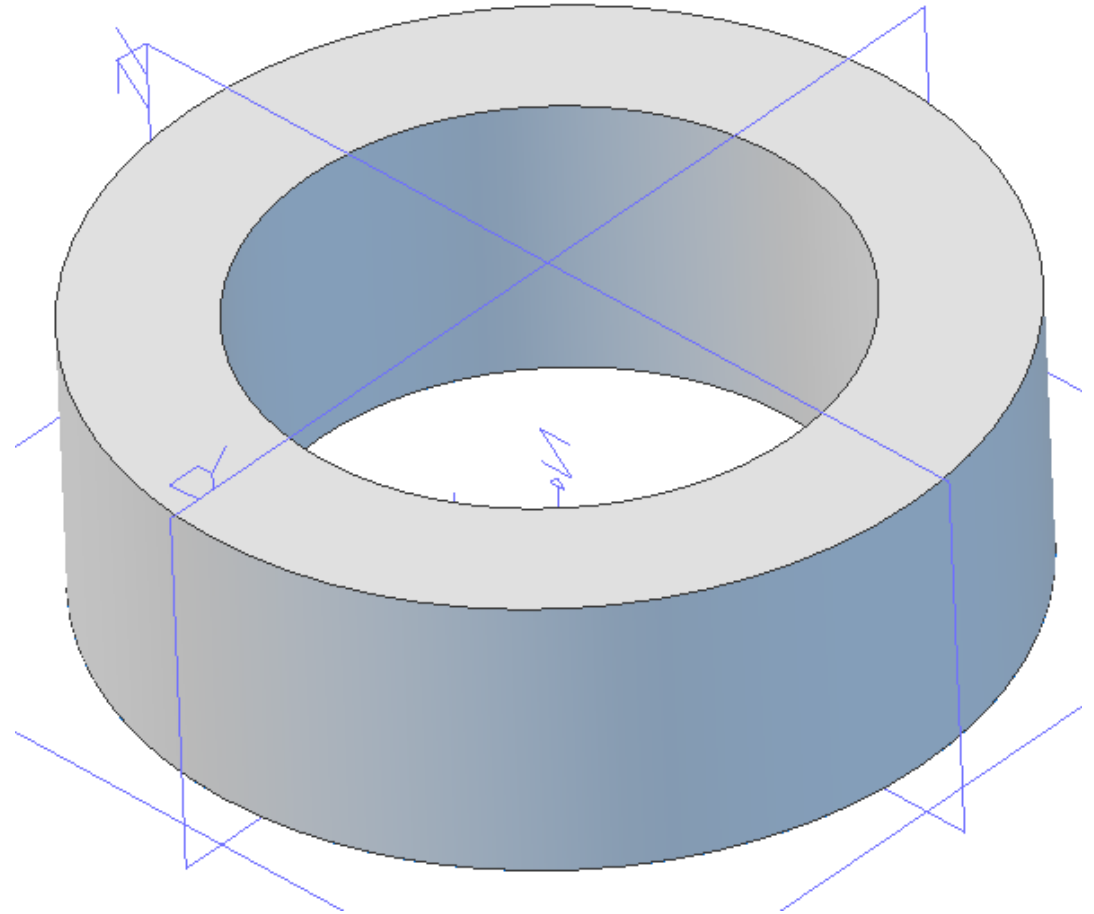
- DANTE database grade: S86XX

## Ring geometry for this workshop

- Inner diameter: 40 mm
- Outer diameter: 60 mm
- Height: 20 mm
- One slice of the ring ( $1.5^\circ$ ) will be modeled

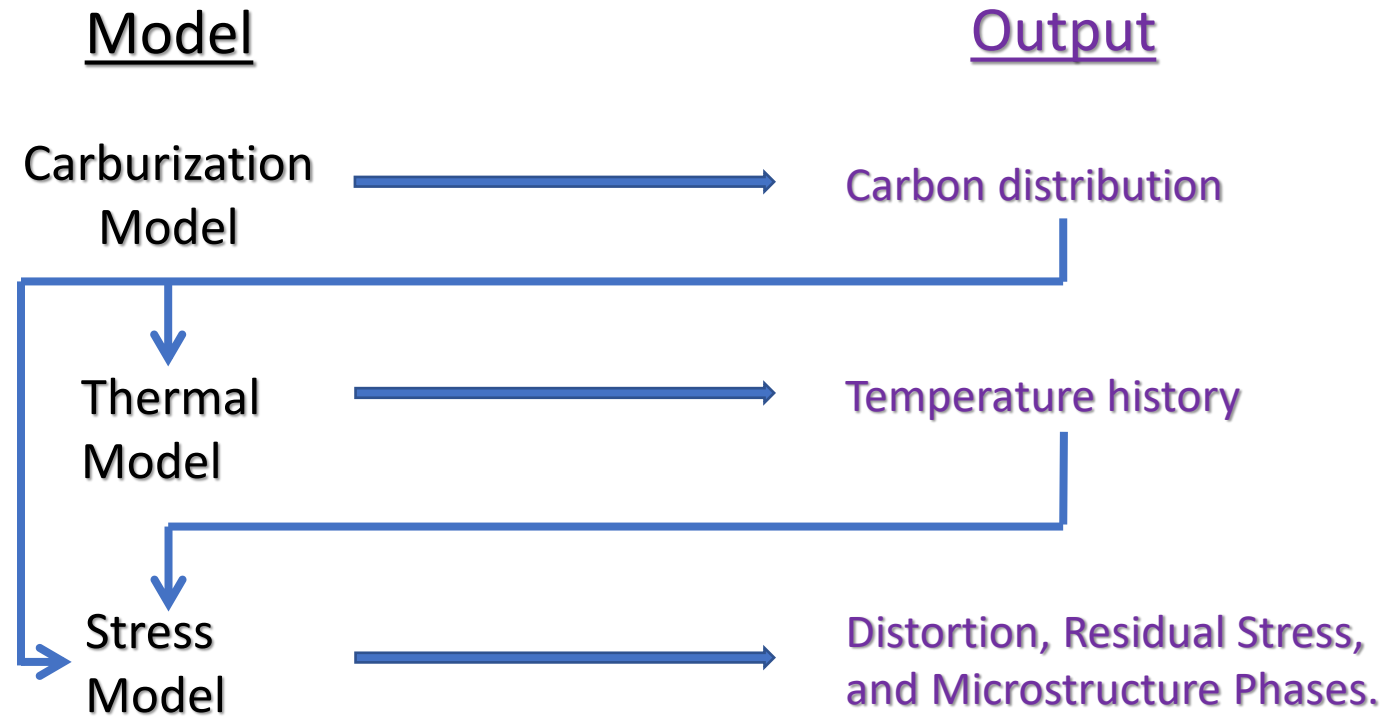
## Heat treatment process steps

- Heating (austenitizing / solution treating)
- Gas carburizing
- Transferring from furnace to quench tank
- Oil Quenching
- Taking the ring out of quench tank and cooling it to room temperature



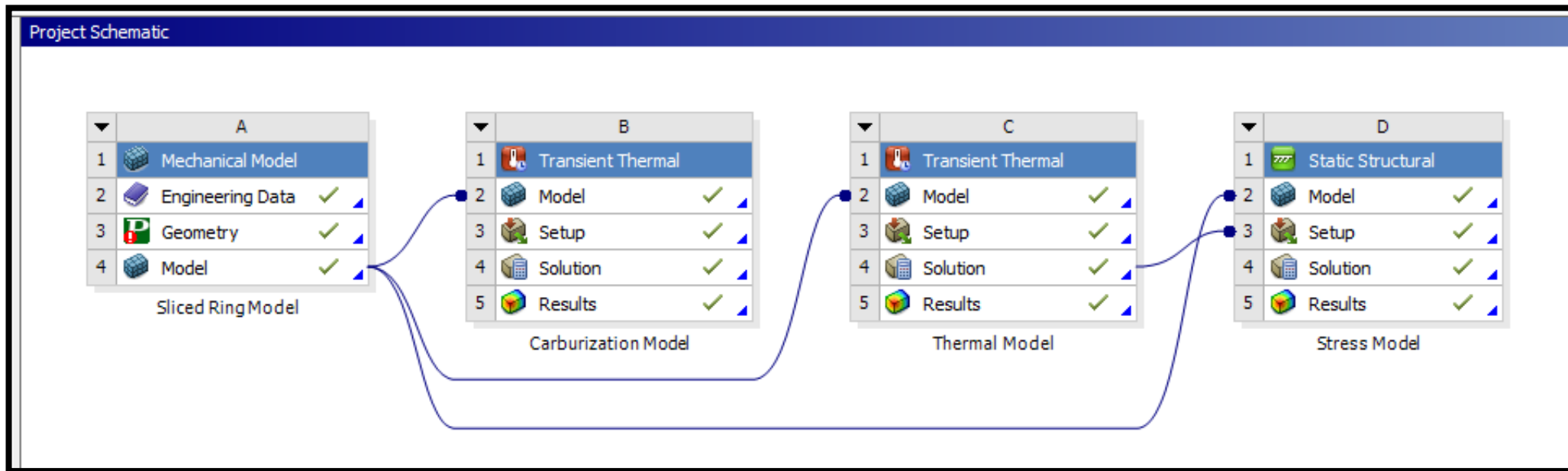
# Modeling Procedure

The carburization model, thermal model and stress model are sequentially coupled.



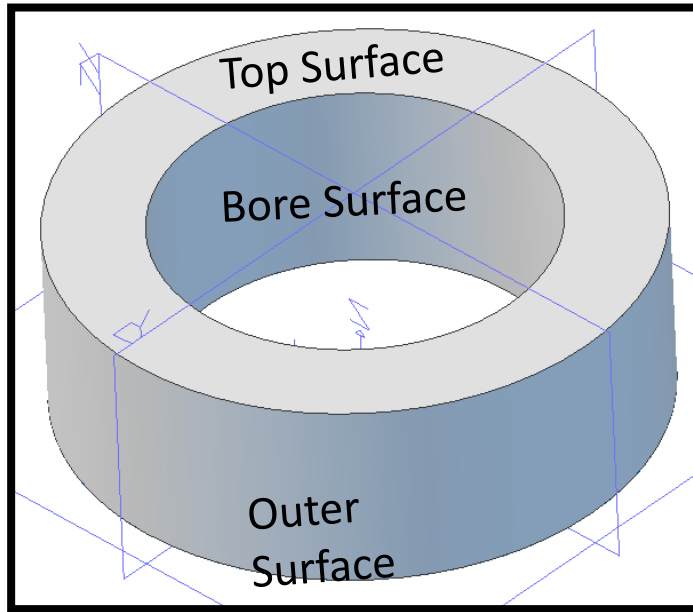
# DANTE Model – Project Schematic

The Project Schematic from Ansys Workbench will include a gas carburization model, followed by a thermal model of the oil quenching process and a static structural stress analysis.

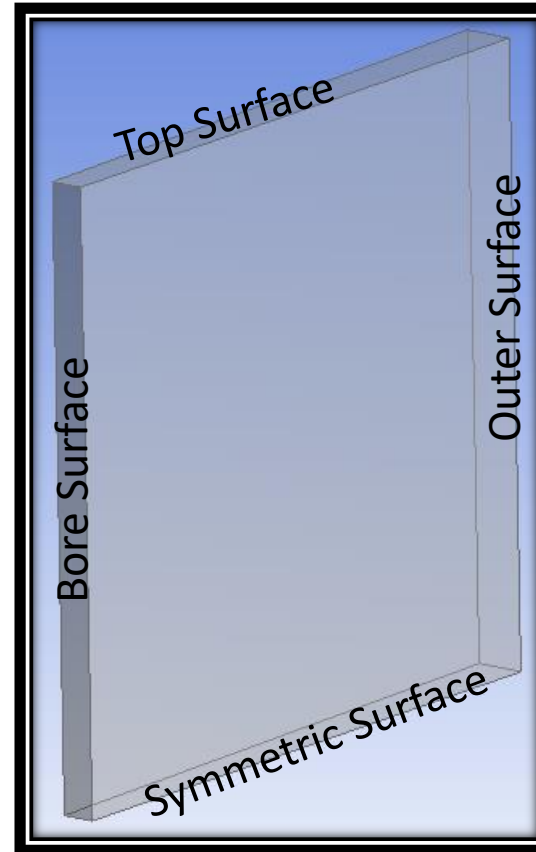


# Simplified Model and Meshing

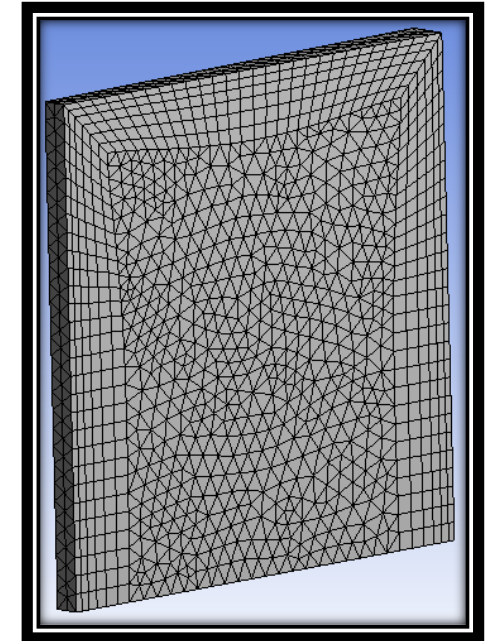
Simplified half sliced ring model used for this workshop:



Ring CAD Model



Sliced Half Ring  
(1.5° slice)



Finite Element  
Meshing

## Thin Sliced Ring

1. Model Geometry and Meshing
2. Carburization Model
3. Quench Hardening Thermal Model
4. Quench Hardening Stress Model

# Model Geometry and Meshing



# Step 1: Start Ansys Workbench

## Create the Project

### 1. Start Workbench:

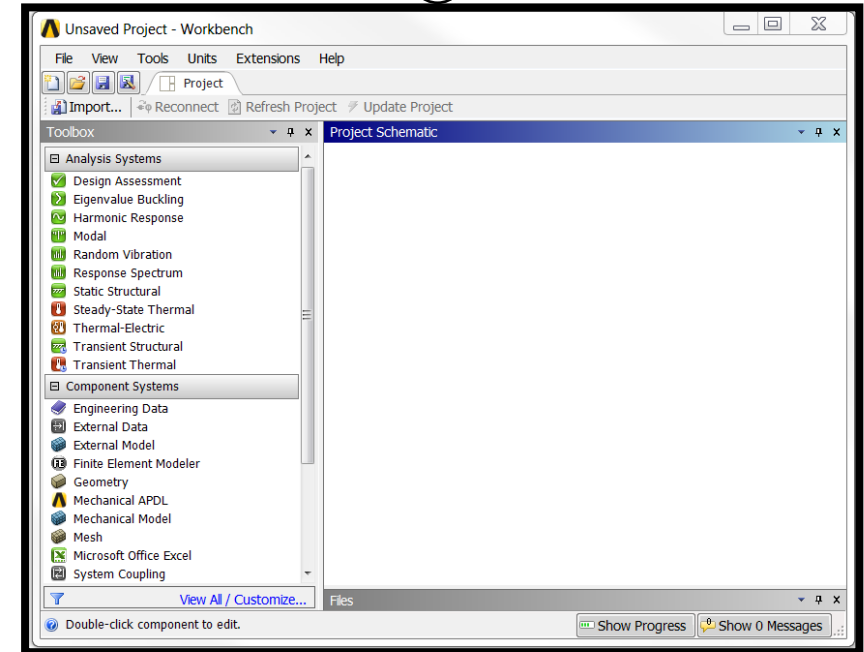
– Start → All Programs → ANSYS “XX.X” → Workbench “XX.X”

**NOTE: XX.X is the Ansys version installed on the computer; e.g., 16.1, 17.2, etc. This workshop is developed under 2019R3 (19.5) but will work for any version 16.1 and later**

Software required for this project:

- ANSYS Mechanical Enterprise R16 or later
- DANTE (Compiled user subroutines and materials database)
- DANTE ACT

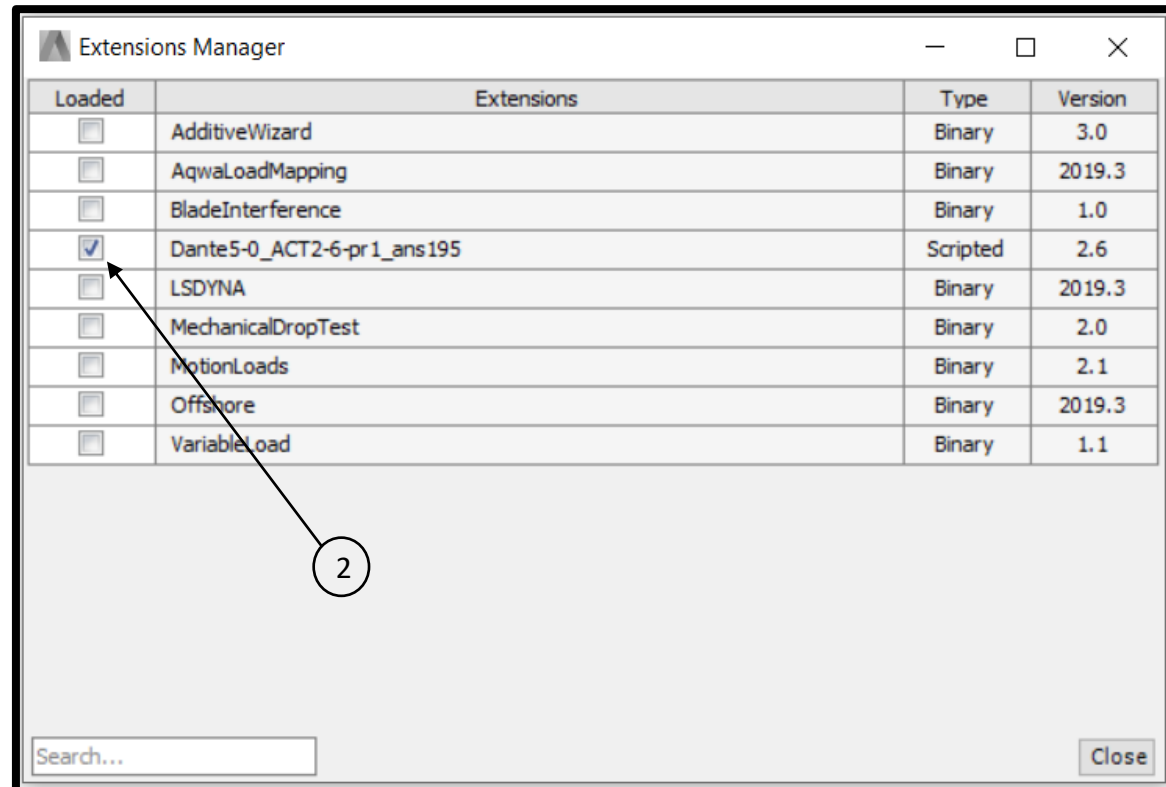
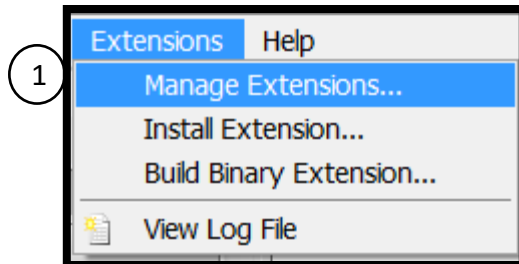
**NOTE: All packages should be installed prior to this workshop**



## Step 2: Activate the DANTE ACT

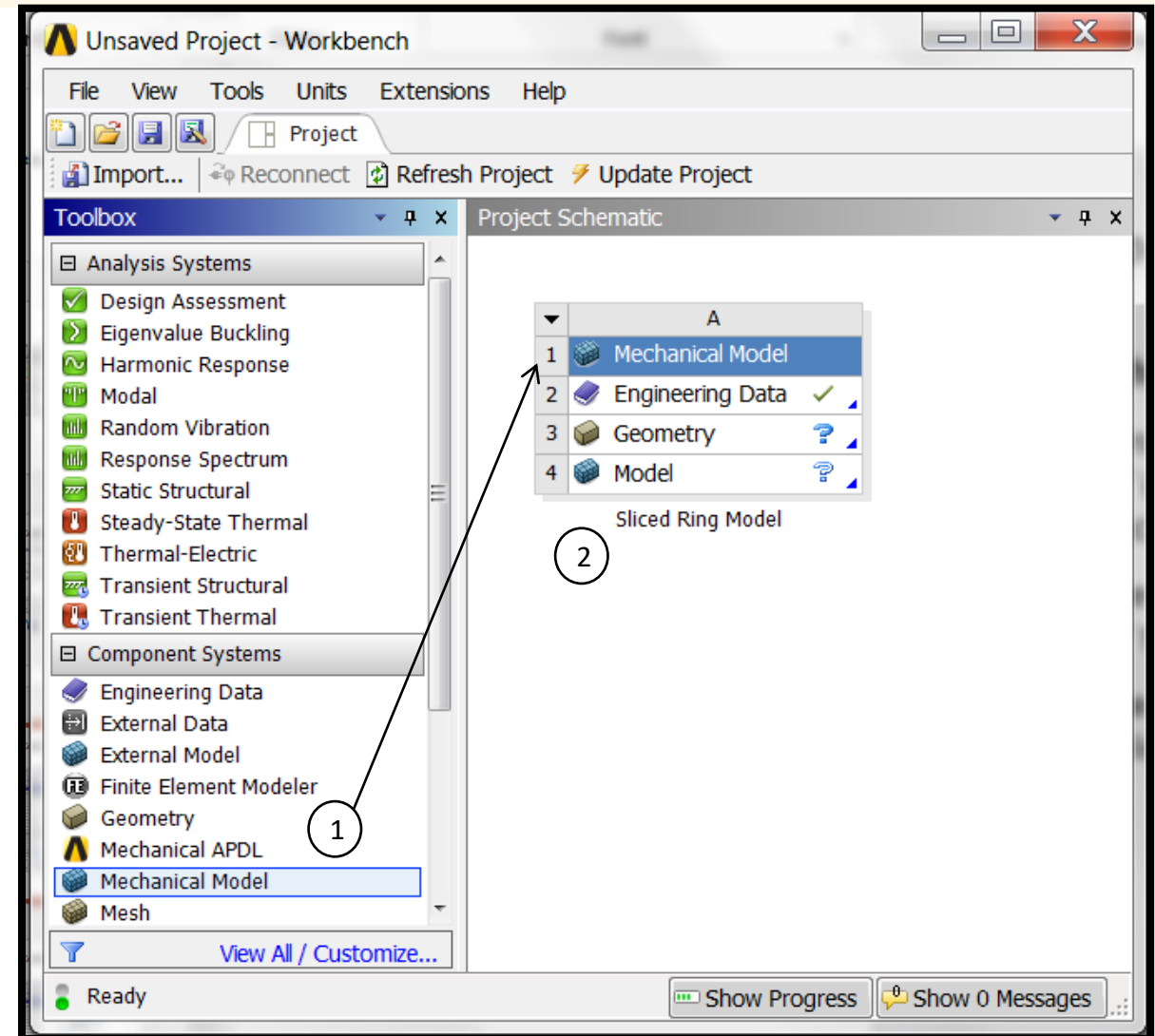
The DANTE ACT is required to setup and run the heat treatment models.

1. Under Workbench Project, click **Extensions**, and select **Manage Extensions** to open the Extensions Manager window
2. In the Extensions Manager window, check the DANTE ACT, **Dante5-0\_ACT2-6-prX\_ans19X**, and close the window
  - The numbers in the Dante version varying according to the ACT revision and Ansys Workbench version



## Step 3: Create Project

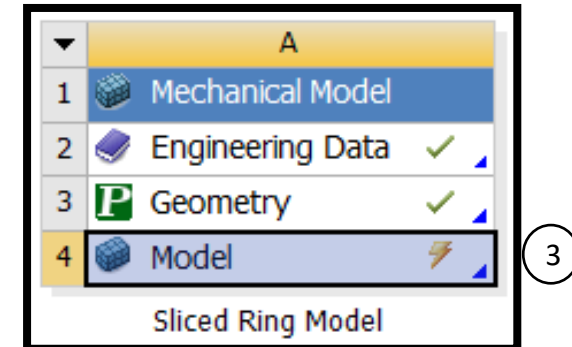
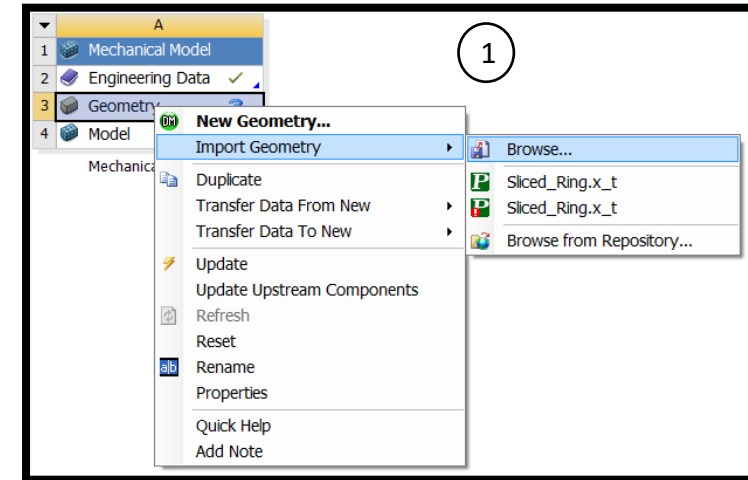
1. Drag and drop **Mechanical Model** from the **Component Systems** under Toolbox into the Project Schematic
2. Rename it as “Sliced Ring Model” (This is optional)
3. Save the project with the name: “sliced\_ring.wbpj”



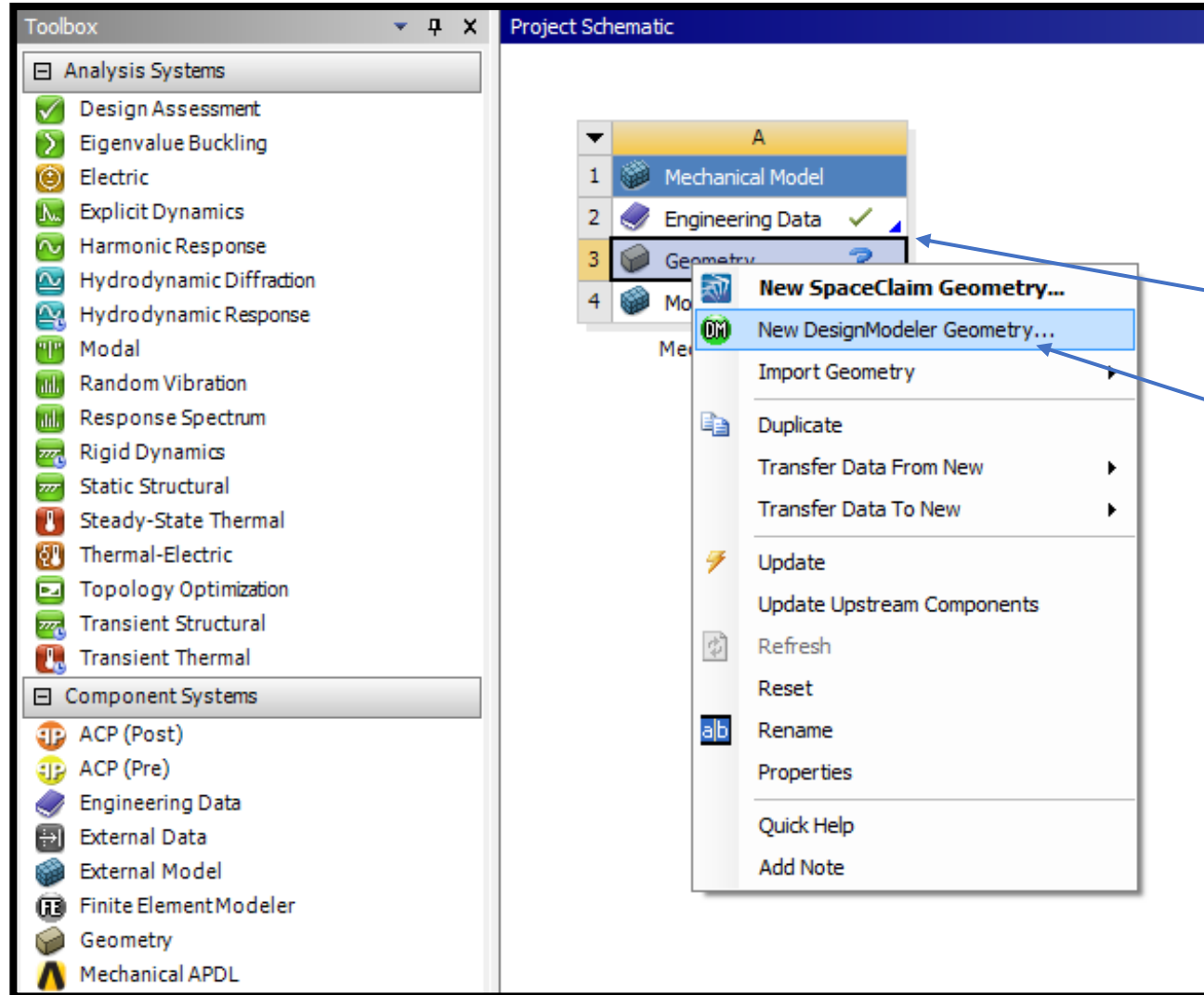
## Step 4: Import the Geometry

The sliced ring geometry for this workshop can be imported from a CAD model created by the instructor (see this page) or can be built using DesignModeler (skip this page)

1. Right click Geometry in the Mechanical Model, and select Import Geometry, then select Browse
2. Navigate to and select the CAD file “Sliced\_Ring.x\_t” to import (The file is provided in this tutorial directory)
3. Double click Model in the Mechanical Model to open ANSYS Mechanical to begin meshing the model (Jump to the “Create Mesh” section)



# Step 5: Opening DesignModeler to Build the Geometry

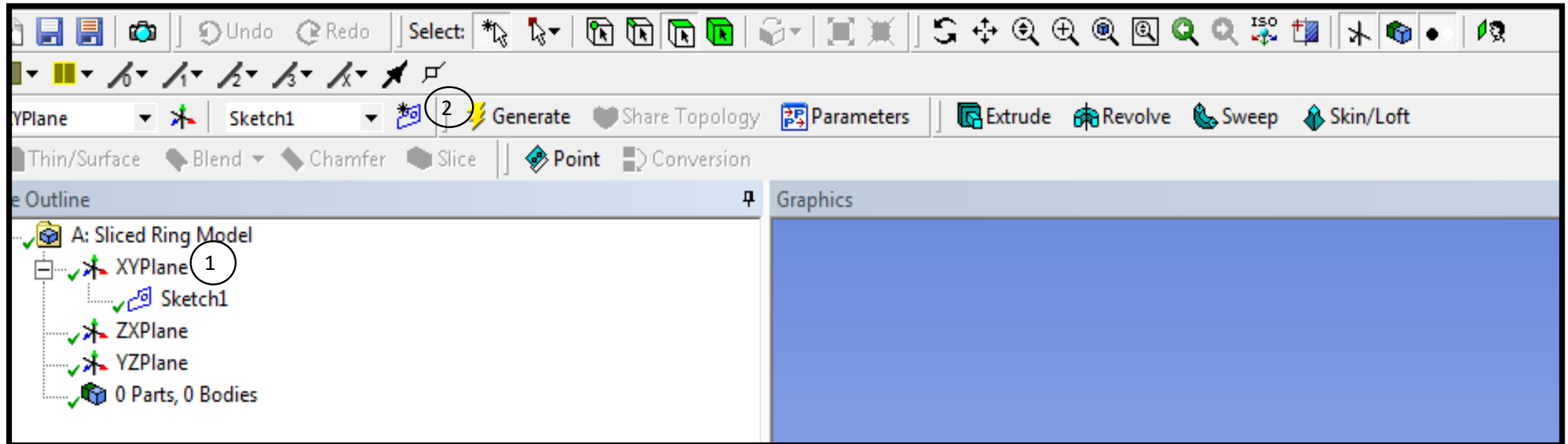


Right click on Geometry

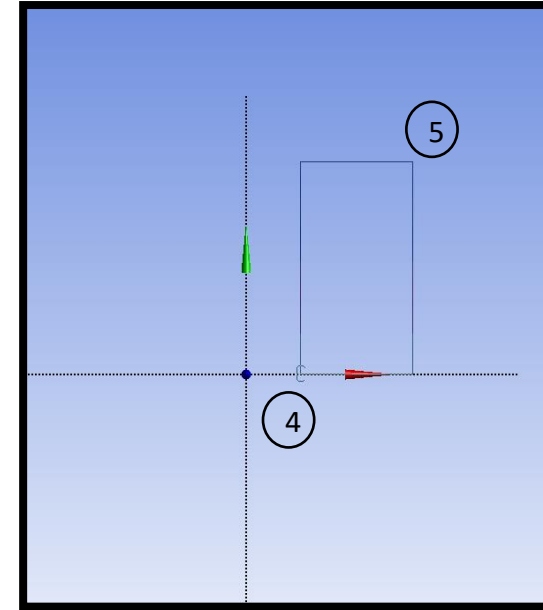
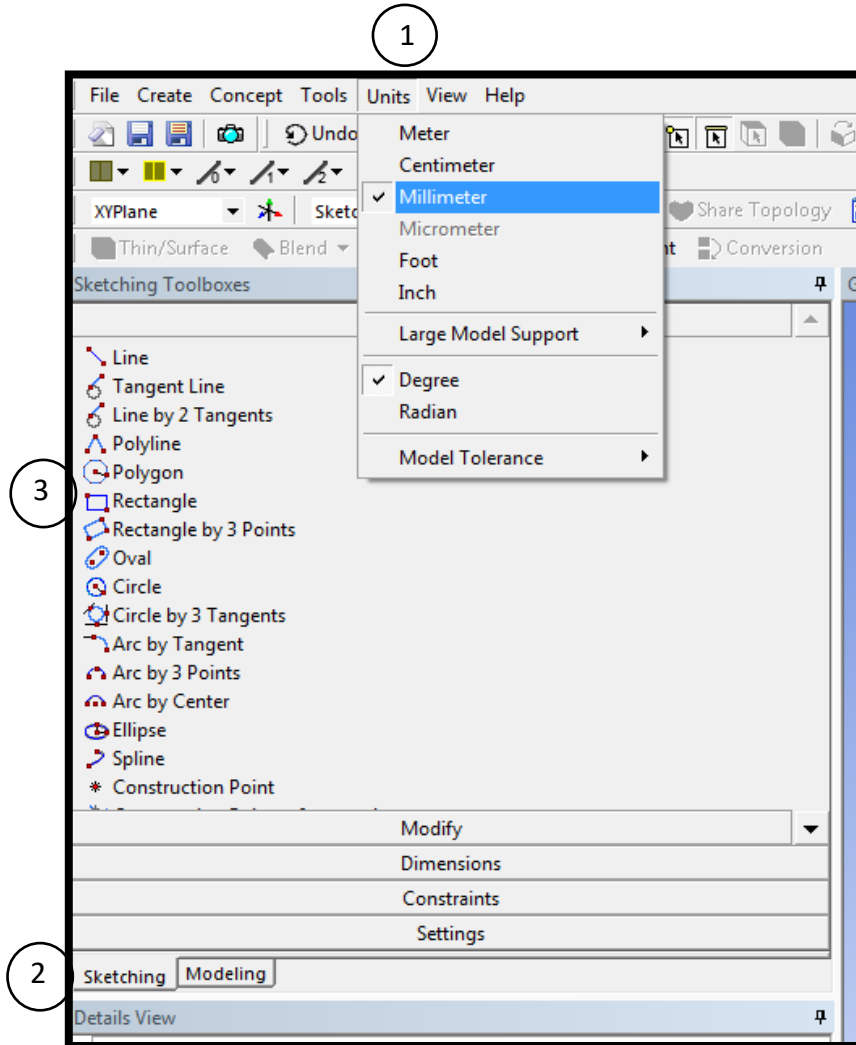
Select New DesignModeler Geometry to open DesignModeler

## Step 6: DesignModeler: Open a New Sketch

1. Select XY Plane to sketch on
2. Select the New Sketch icon to begin a new sketch
3. Select the Look at Face/Plane/Sketch icon to correctly orient the XY Plane

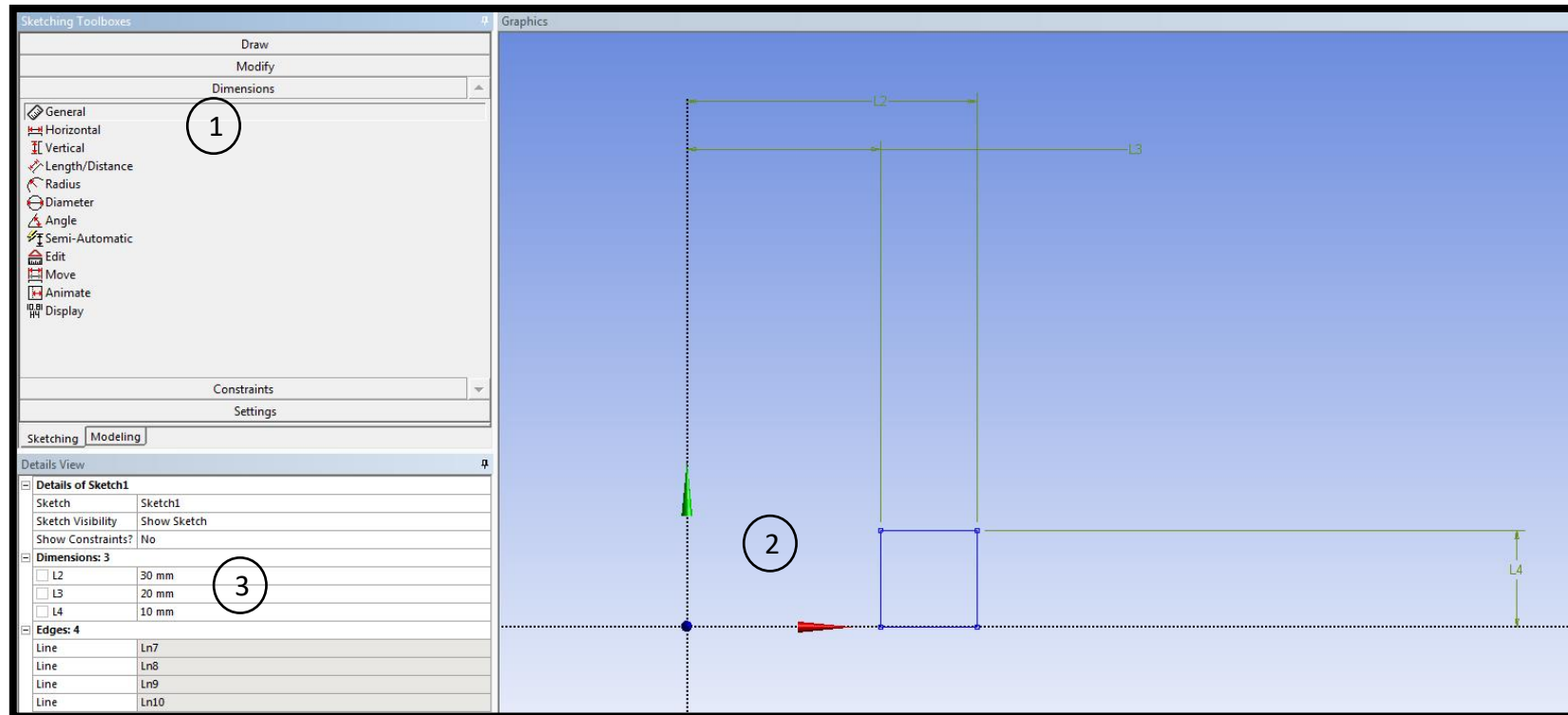


# Step 7: DesignModeler: Sketch Ring Cross-Section



1. Make sure the units are in mm (All DANTE models must use mm length unit)
2. Change from the Modeling tab to the Sketching tab
3. Select Rectangle to draw the ring cross-section
4. Place the first corner of the rectangle on the positive x-axis
5. Place the second corner of the rectangle in the positive x-direction and positive y-direction to the right of the first corner. This allows the use of symmetry about the XZ Plane

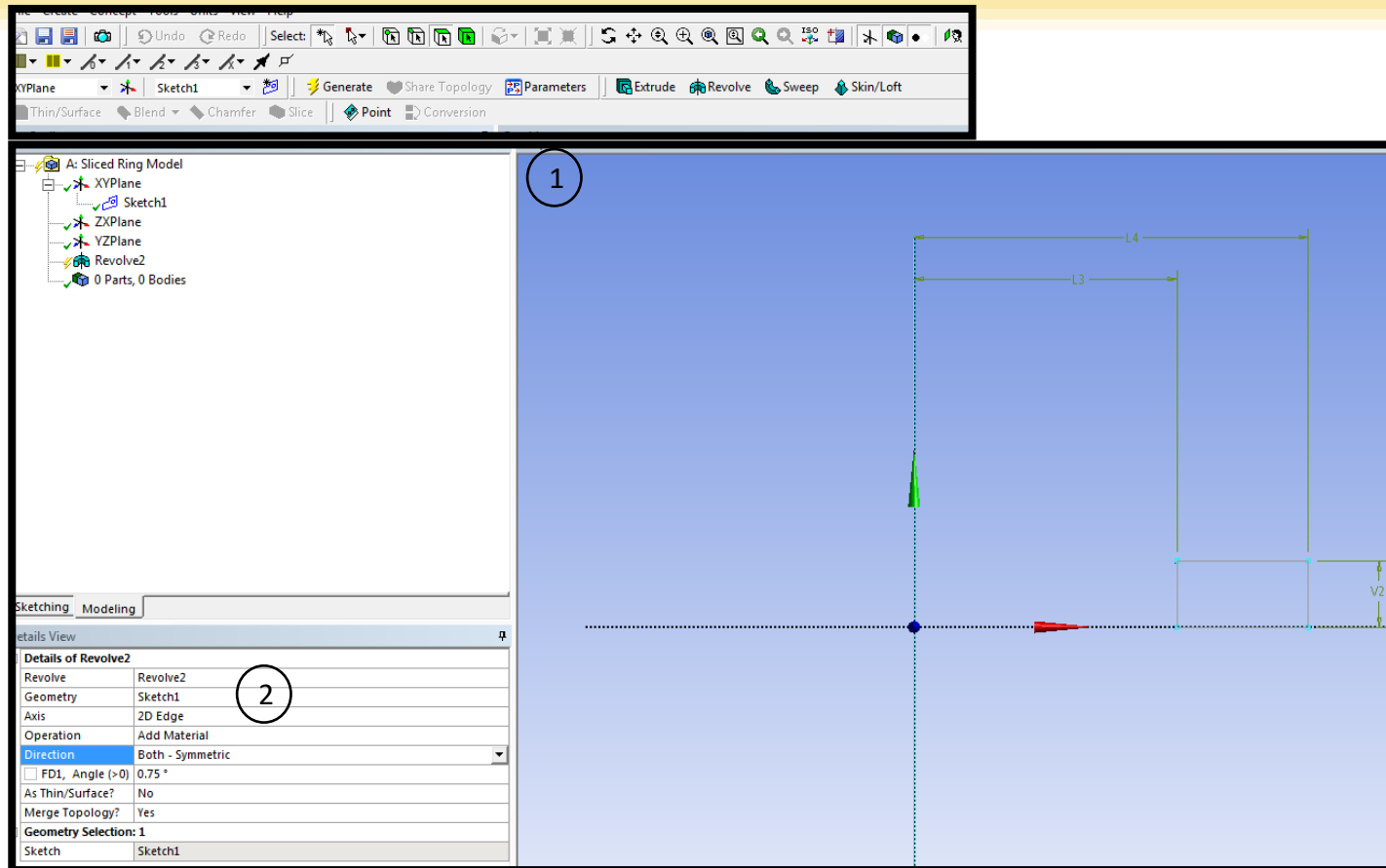
# Step 8: DesignModeler: Sketch Ring Cross-Section



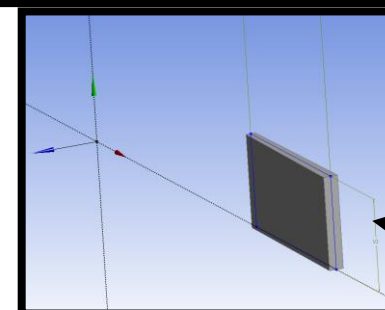
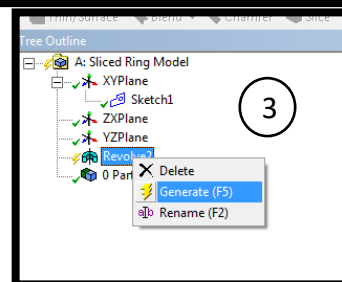
1. Select the Dimensions tab and the General option
2. The inner and outer radii are dimensioned from the y-axis and the height is dimensioned from the x-axis
3. The dimensions can be modified in the Details of Sketch area and are 20 mm, 30 mm, and 10 mm for the inner radius, outer radius, and height/2, respectively. The height is height/2 because of the symmetry about the XZ Plane



# Step 9: DesignModeler: Create Ring Slice



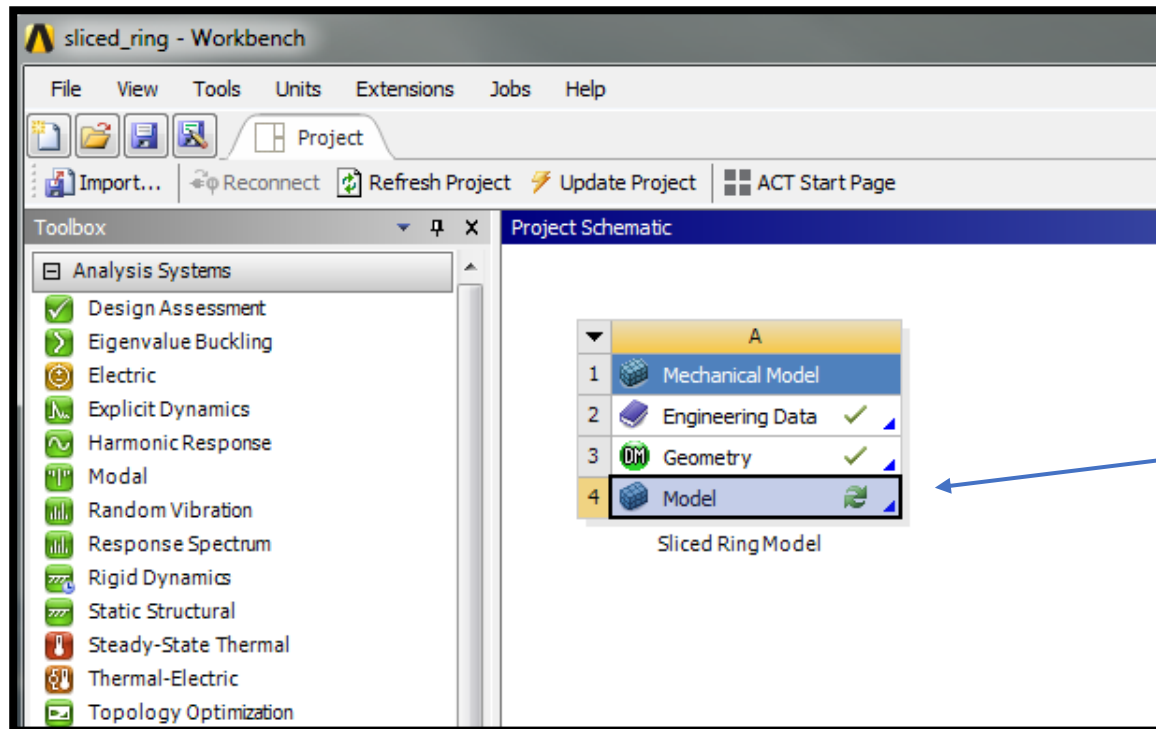
1. Select Revolve in the toolbar
2. In the Details View, select the newly created sketch for Geometry (should be able to just hit Apply), select the y-axis for Axis, select Both – Symmetric for Direction, and enter 0.75° (creating a 1.5° slice) for Angle
3. Right click on the newly created Revolve in the Tree Outline and click Generate to create the slice



1.5° Slice

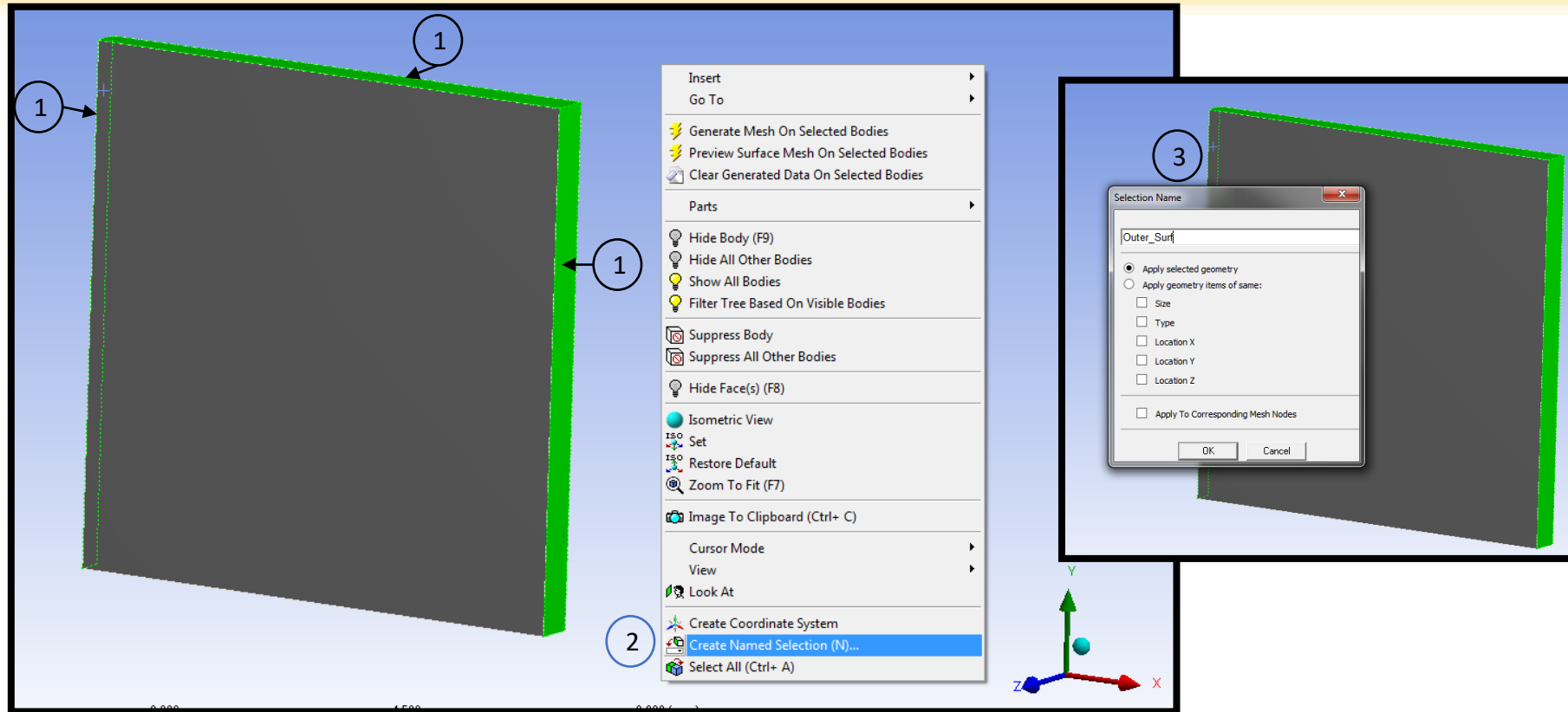
# Step 10: DesignModeler: Save Project

The ring slice geometry is complete.  
Close DesignModeler and save the project in Workbench.



Double click on Model to  
open Mechanical and begin  
meshing the model

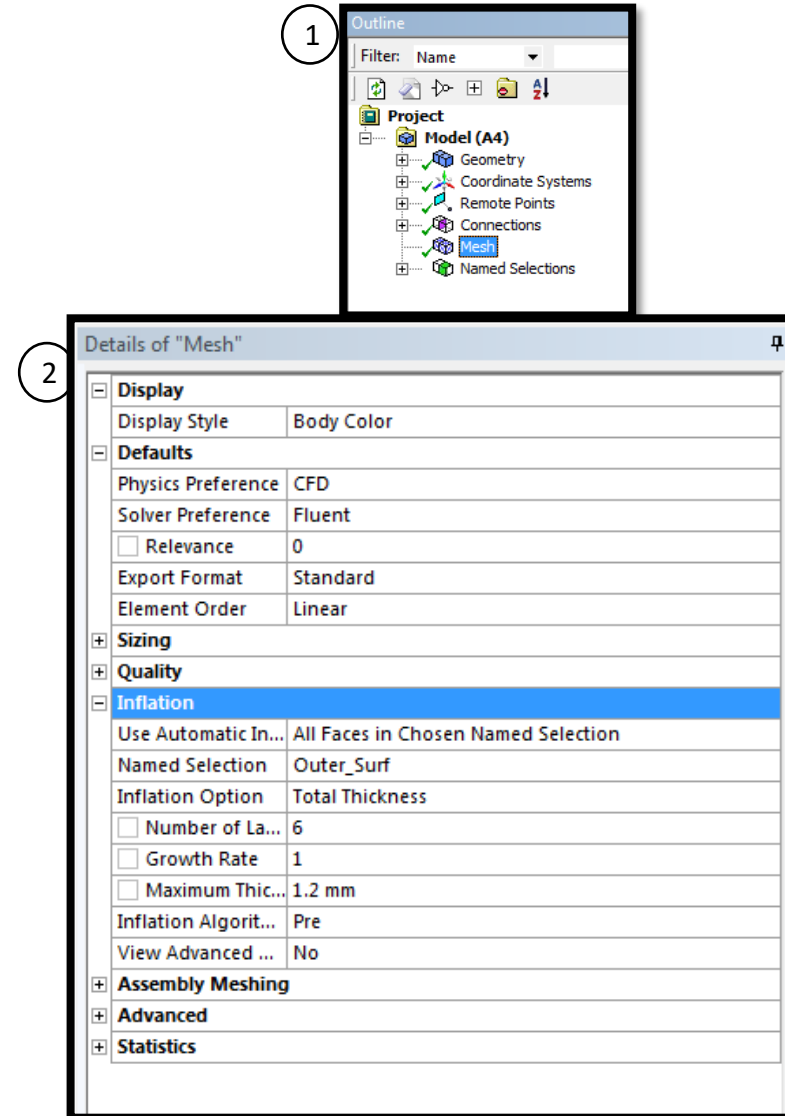
# Step 11: Define Surfaces for Meshing & Boundary Conditions



1. Use ctrl + f keyboard shortcut to activate the face selection, hold down ctrl on the keyboard to select multiple surfaces, and select the inner diameter, outer diameter, and top surfaces of the ring slice
2. Click the right mouse button in the work space and select **Create Named Section**
3. Name the selection **Outer\_Surf**. Click **OK** when complete.

# Step 12: Define Meshing Parameters

1. Select **Mesh** in the Project Tree
2. In the Details of "Mesh"
  - Change the Physics Preference to CFD
  - Choose Linear for Element order  
(REQUIRED FOR ALL DANTE MODELS)
  - Expand the Inflation tab in Details of "Mesh"
  - Change the Use Automatic Inflation to All Faces in Chosen Named Selection
  - Select Outer\_Surf as the Named Selection (This is the surface we just defined in the previous slide)
  - Choose Total Thickness for the Inflation Option
  - Make the Number of Layers equal to 6
  - Make the Growth Rate equal to 1
  - Make the Maximum Thickness equal to 1.2 mm

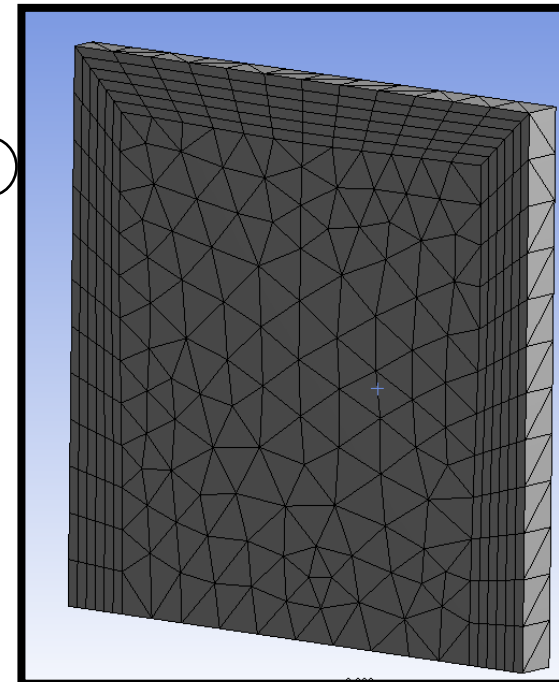
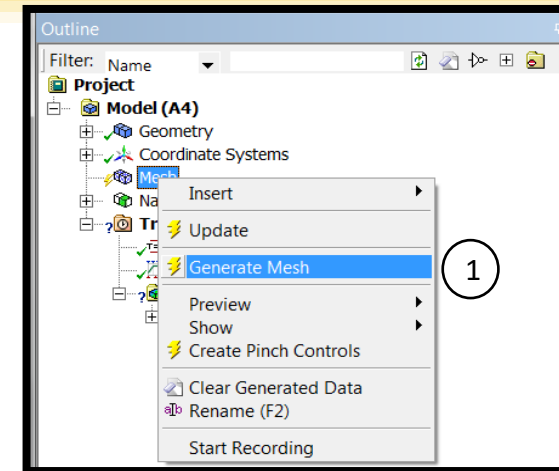


## Step 12b: Brief Explanation of Meshing Techniques

Using the Inflation meshing option allows the user to create a fine layer of elements on the surface which is needed to catch the steep carbon, thermal, phase transformation, and stress gradients present close to the surface during the quenching process. Coarser elements are used away from this fine surface layer. It is ideal to make the Maximum Thickness equal to the total expected case depth, not just the effective case depth. If no carburization is used, a minimum of 1 mm should be used for the Maximum Thickness. The Number of Layers can then be determined by the desired thickness of the element:  $\text{Number of Layers} = \text{Maximum Thickness} / \text{Element Thickness per Layer}$ . DANTE recommends using a minimum element thickness per layer of 0.2 mm for gas carburization or no carburization and a minimum element thickness per layer of 0.05 mm for low pressure carburization.

# Step 13: Mesh the Component

1. In the Project Tree, right click Mesh, and select **Generate Mesh** to mesh the part
2. The mesh is generated with six layers of wedge elements in the surface, and coarser tetrahedral elements in the core
3. Close ANSYS Mechanical and save the project in ANSYS Workbench before moving forward



**Note: Different methods can be used to create the mesh. However, finer wedge or hexagonal elements are preferred close to the surface to catch carbon, thermal, phase transformation and stress gradients during heat treatment processes**

# Carburization Model

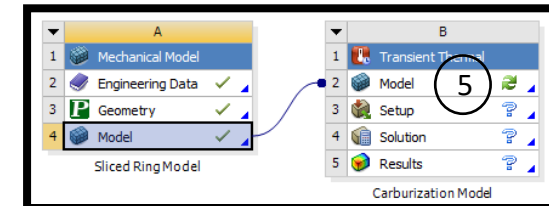
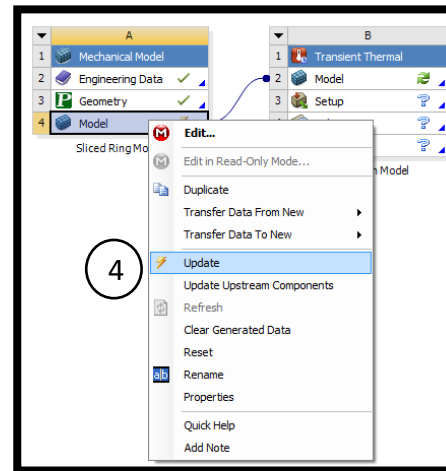
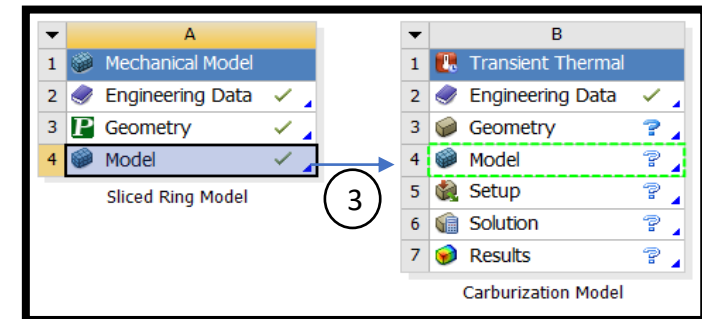
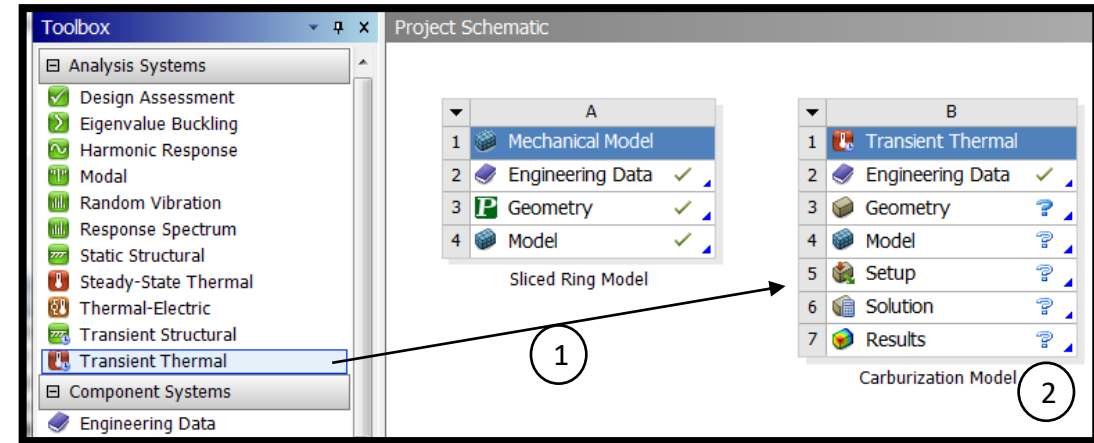
# Carburization Process Description

- After the ring reaches thermal equilibrium with the furnace temperature, the carburization process will start
- Carburization process:
  - Carburization Step 1: 6 hours (21,600s); Temperature: 900 °C; Carbon potential: 0.95%
  - Carburization Step 2: 2 hours (7,200s); Temperature: 875 °C; Carbon potential: 0.80%
- The carburization model is executed first. The carbon profile obtained is then imported to the thermal and stress models when they are executed
  - The carbon profile is mapped to the thermal and the stress models at the beginning of the air transfer step in this workshop. The carburization process does not need to be modeled as a separate step in the thermal and stress models.



# Step 1: Carburization Model Setup, Add Analysis System to Project

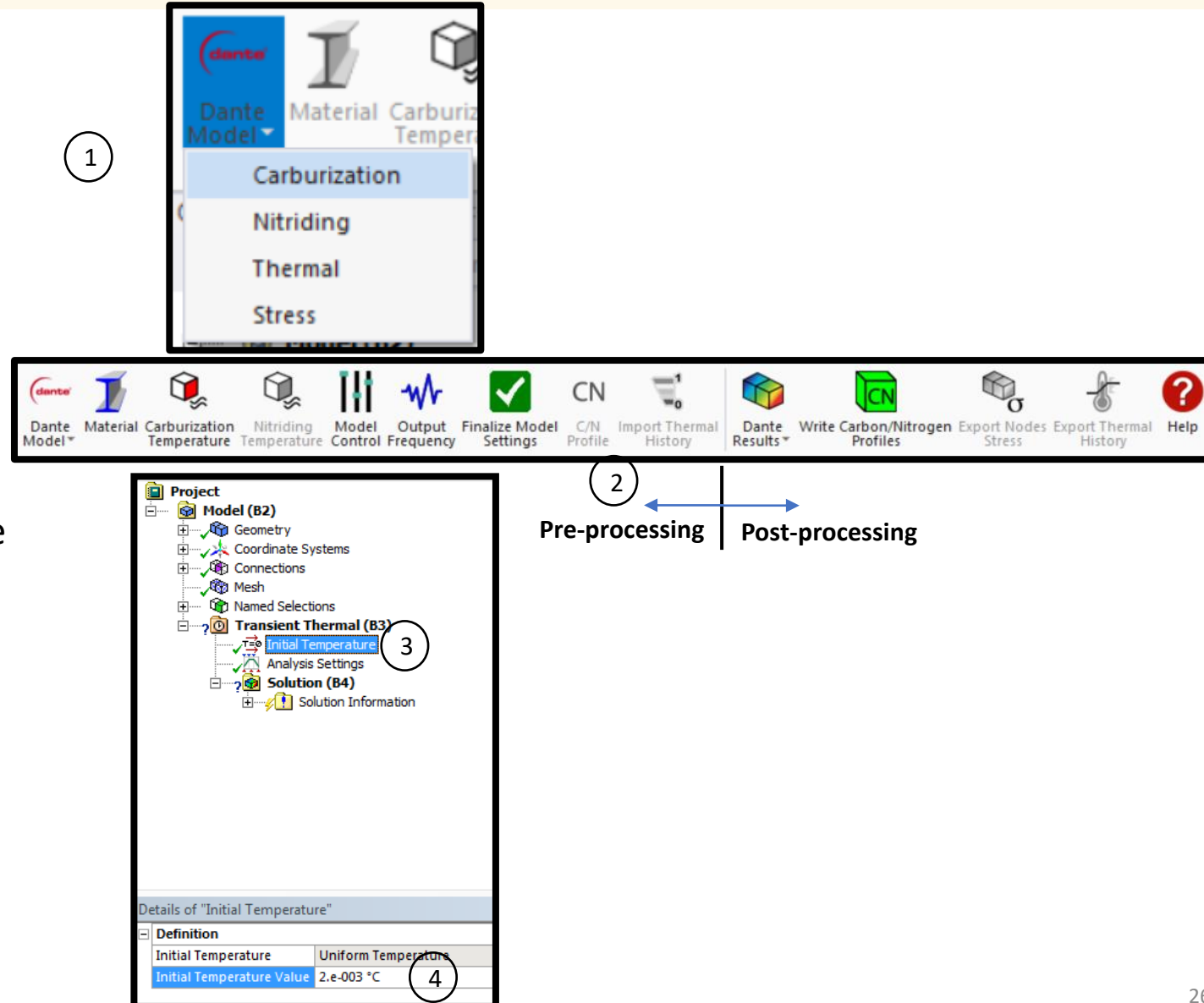
1. Drag and drop a Transient Thermal Analysis System into the Project Schematic
2. Rename it “Carburization Model”
3. Drag and drop the Model from the Sliced Ring Model to the Carburization Model
4. Right click on Model in the Sliced Ring Model (now has a lightening bolt symbol next to it) and select Update
5. Double click Model in the Carburization Model to open Ansys Mechanical after the Update is complete



## Step 2: DANTE Model & Initial Temperature

1. With Mechanical open, click on Dante Model in the Dante toolbar and select **Carburization**
2. Selecting Carburization opens the buttons needed to complete the Carburization Model. The buttons to the left of the single bar are for pre-processing (setting up the model) and the buttons to the right of the single bar are used for post-processing (viewing the results). The Question Mark (?) is a link to the Dante Help File
3. Click **Initial Temperature**
4. Under Details of "Initial Temperature", assign a value of 0.002. This is the base carbon in weight fraction.

**Note: ANSYS Transient Thermal model and user subroutines are used to model the carbon diffusion process. Ansys's Temperature variable is used to represent carbon in the carburization model. A Carburization Temperature will be defined to handle the furnace temperature.**



The screenshot illustrates the DANTE software interface during the Carburization setup process. It is divided into two main sections: Pre-processing and Post-processing, separated by a vertical line with a double-headed arrow. A horizontal toolbar at the top contains various icons for different model types and actions. A dropdown menu for 'Dante Model' is open, showing options: Carburization, Nitriding, Thermal, and Stress. The 'Carburization' option is highlighted. Below the toolbar, the Project tree on the left shows the hierarchy: Project > Model (B2) > Transient Thermal (B3) > Initial Temperature (B4). The 'Initial Temperature' item is highlighted with a blue box and a circled '3'. The bottom panel shows the 'Details of "Initial Temperature"' with a table for the 'Definition' section:

Definition	
Initial Temperature	Uniform Temperature
Initial Temperature Value	2.e-003 °C

The value '2.e-003 °C' is highlighted with a blue box and a circled '4'. A circled '1' points to the 'Dante Model' dropdown, and a circled '2' points to the Pre-processing/Post-processing separator.

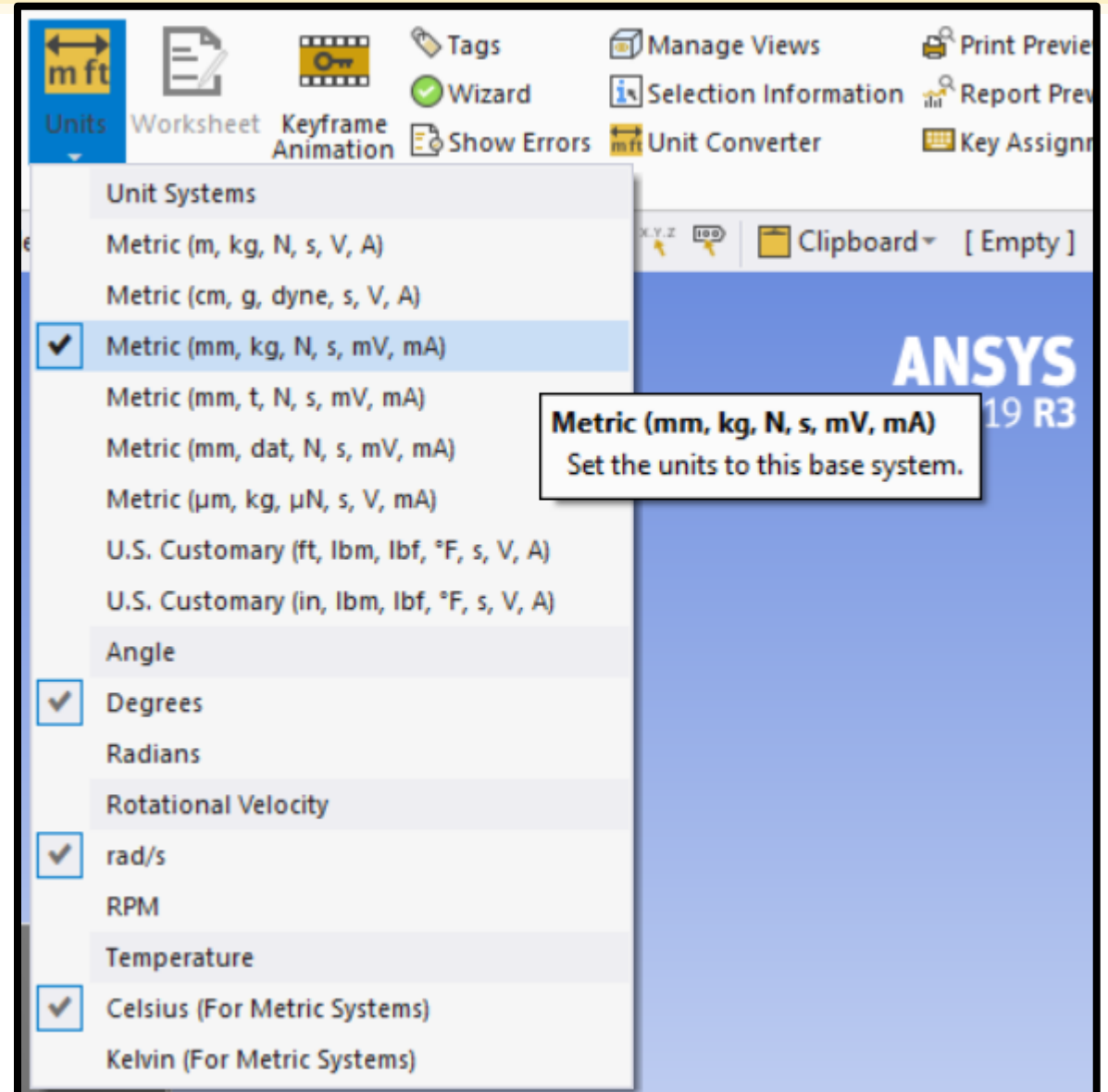
## Step 3: Define Units

**It is critical that the units be properly defined**

1. Click on **Units** in the **Home** toolbar
2. **Select Metric (mm, kg, N, s, mV, mA)**
3. Select Degrees
4. **Select rad/s (This isn't critical as there is no motion defined in the heat treatment models)**
5. Select Celsius (For Metric Systems)

**NOTE:** It is absolutely critical that these units are selected. If different units are selected, the model will run, but the results will be WRONG.

**For Example:** If grams are chosen, carbon diffusivity will be very high and thru carburizing will most likely occur. If tonne is chosen, the carbon diffusivity will be very low and the carbon will not penetrate the part.



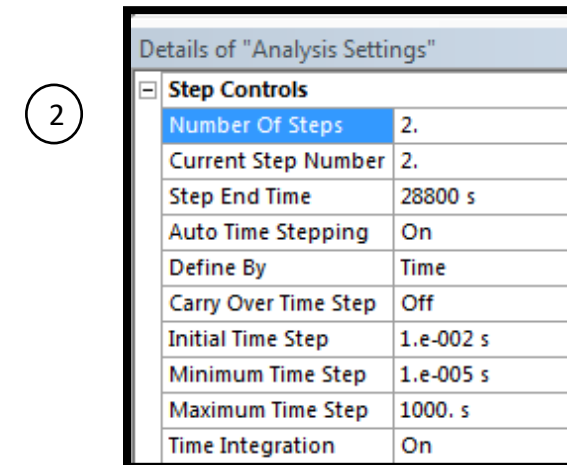
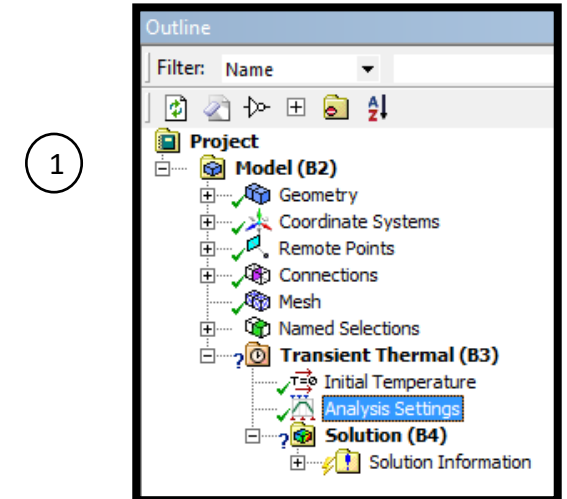
# Step 4: Define Processing Steps

1. Select Analysis Settings in the Project Tree to open the Details of "Analysis Settings"
2. Modify the options listed in the Details of "Analysis Settings" under Step Controls to the values listed in the table. The Number of Steps for this analysis is 2

**NOTE: The step numbers MUST be entered from highest to lowest; i.e., start with Current Step Number 2 for ANSYS to accept the time because the default for each step is 1 second. So, if 21,600 seconds is entered for the Step 1 End Time, ANSYS will reject it because the Step 2 End Time is still only 2 seconds.**

3. Repeat the same procedure for Current Step Number 1

Current Step Number	Step End Time	Auto Time Stepping	Define By	Carry Over Time Step	Initial Time Step	Minimum Time Step	Maximum Time Step	Time Integration
2	28800	On	Time	Off	1.00E-02	1.00E-05	1000	On
1	21600	On	Time	N/A	1.00E-02	1.00E-05	1000	On



# Step 5: Nonlinear Formulation

1. Change Nonlinear Formulation to Full under Nonlinear Controls in the Details of “Analysis Settings”

**Note: This is required to call the DANTE user subroutines. Be aware that the model may run if this step is omitted, but the results will be WRONG.**

1

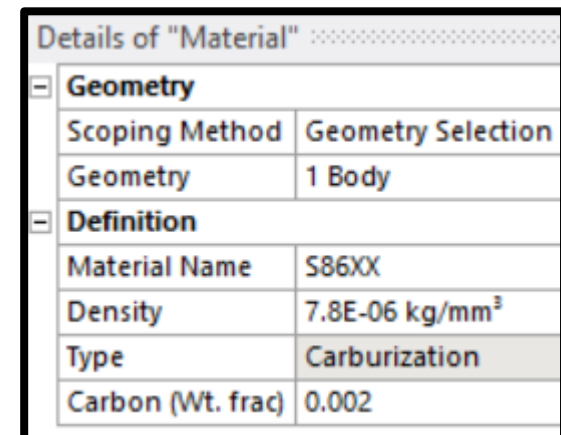
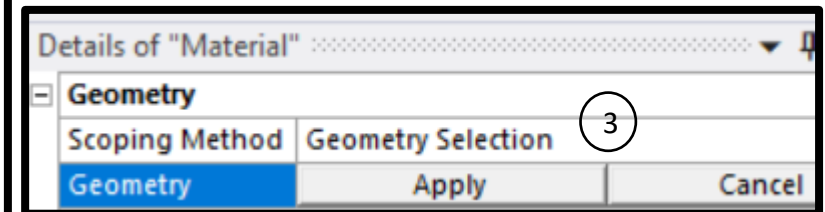
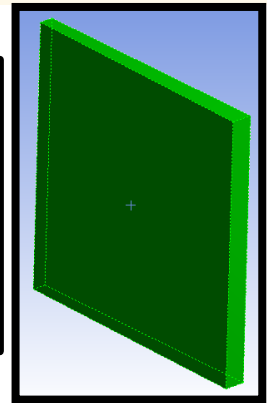
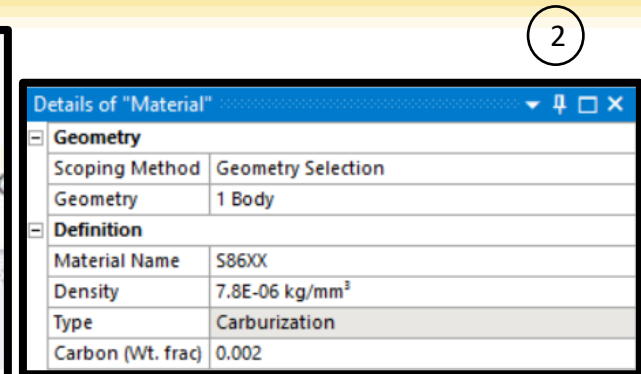
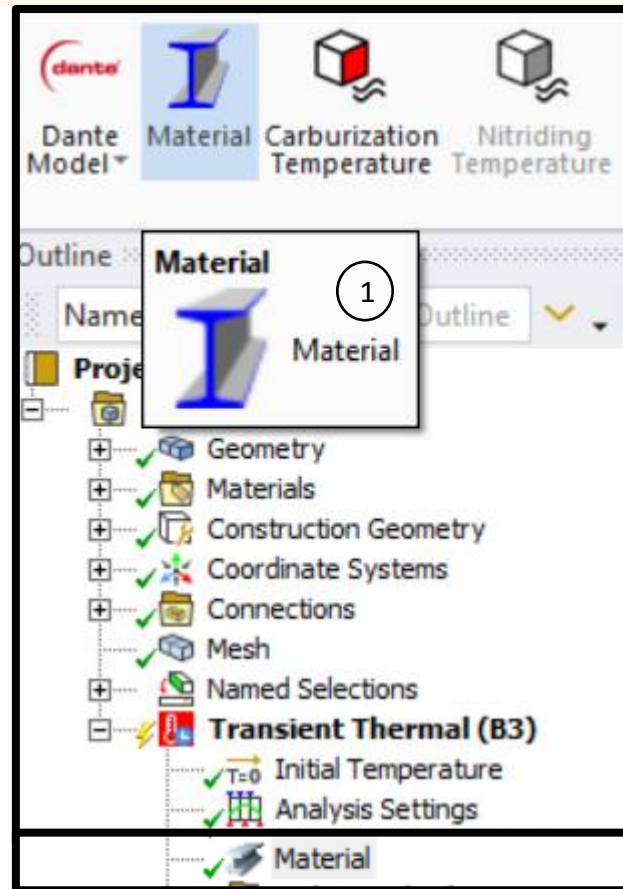
Details of "Analysis Settings"	
[-] <b>Step Controls</b>	
Number Of Steps	2.
Current Step Number	1.
Step End Time	21600 s
Auto Time Stepping	On
Define By	Time
Initial Time Step	1.e-002 s
Minimum Time Step	1.e-005 s
Maximum Time Step	1000. s
Time Integration	On
[-] <b>Solver Controls</b>	
Solver Type	Program Controlled
+ <b>Radiosity Controls</b>	
[-] <b>Nonlinear Controls</b>	
Heat Convergence	Program Controlled
Temperature Convergence	Program Controlled
Line Search	Program Controlled
Nonlinear Formulation	Program Controlled
+ <b>Output Controls</b>	
+ <b>Analysis Data Management</b>	
+ <b>Visibility</b>	

# Step 6: Assign Material

1. Select **Material** from the Dante toolbar to add it to Transient Thermal in the Project Tree

The following Steps apply to modifying values in the Details of "Material":

2. Click on the yellow box next to **Geometry**, select the entire part body. Simply click on the part to select the entire body
3. Select Apply for Geometry
4. Select **S86XX** for the AISI 8600 series steel from the Material Name dropdown menu
5. Change the Carbon (Wt. frac) to 0.002 to indicate AISI 8620
6. The Type should be set to **Carburization**

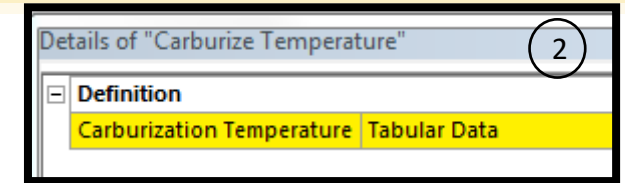
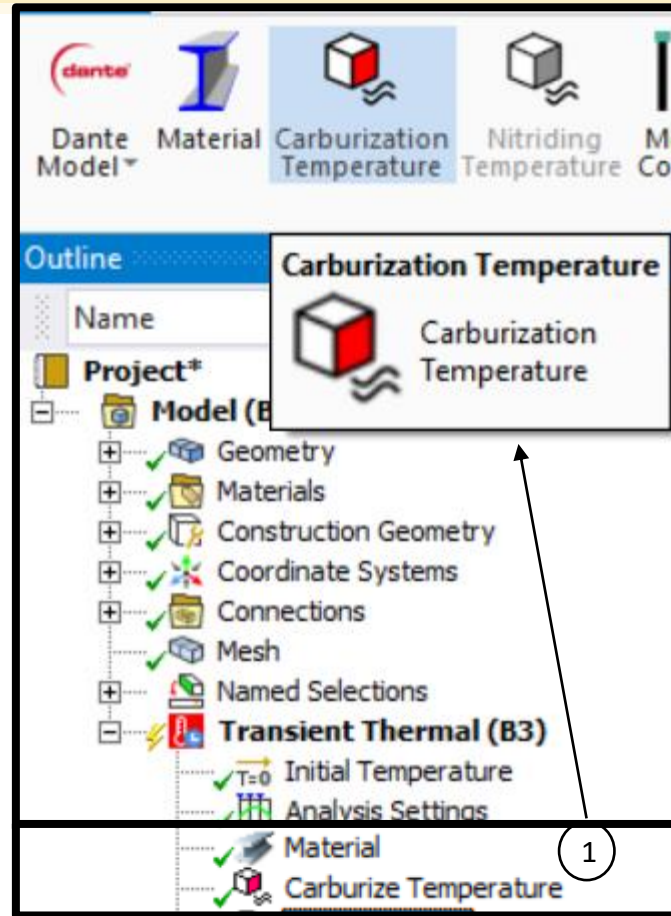
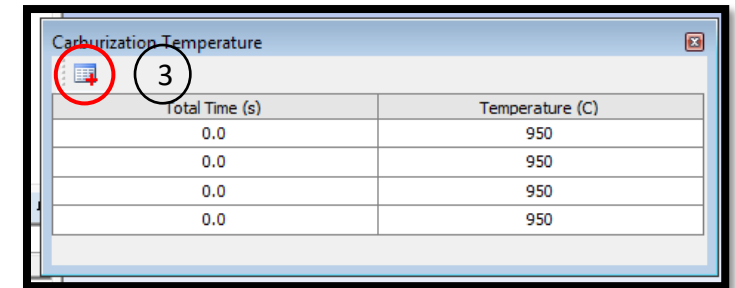




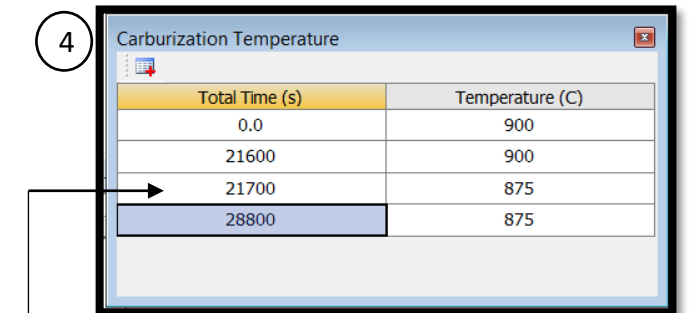
# Step 7: Define Carburization Temperature

1. Select **Carburization Temperature** from the Dante Toolbar, and **Carburize Temperature** will be added under Transient Thermal in the Project Tree
2. Click **Tabular Data** in the Details of “**Carburize Temperature**” to open the Carburization Temperature table
3. Add 4 rows to the table; time zero, the two step end times, and an intermittent time for temperature ramping
4. Input the time-temperature table for the carburization process shown and click Apply to close the table

**NOTE: ANSYS will ramp the temperature, so it is important to add an intermittent time to allow for the ramping.**

Total Time (s)	Temperature (C)
0.0	950
0.0	950
0.0	950
0.0	950

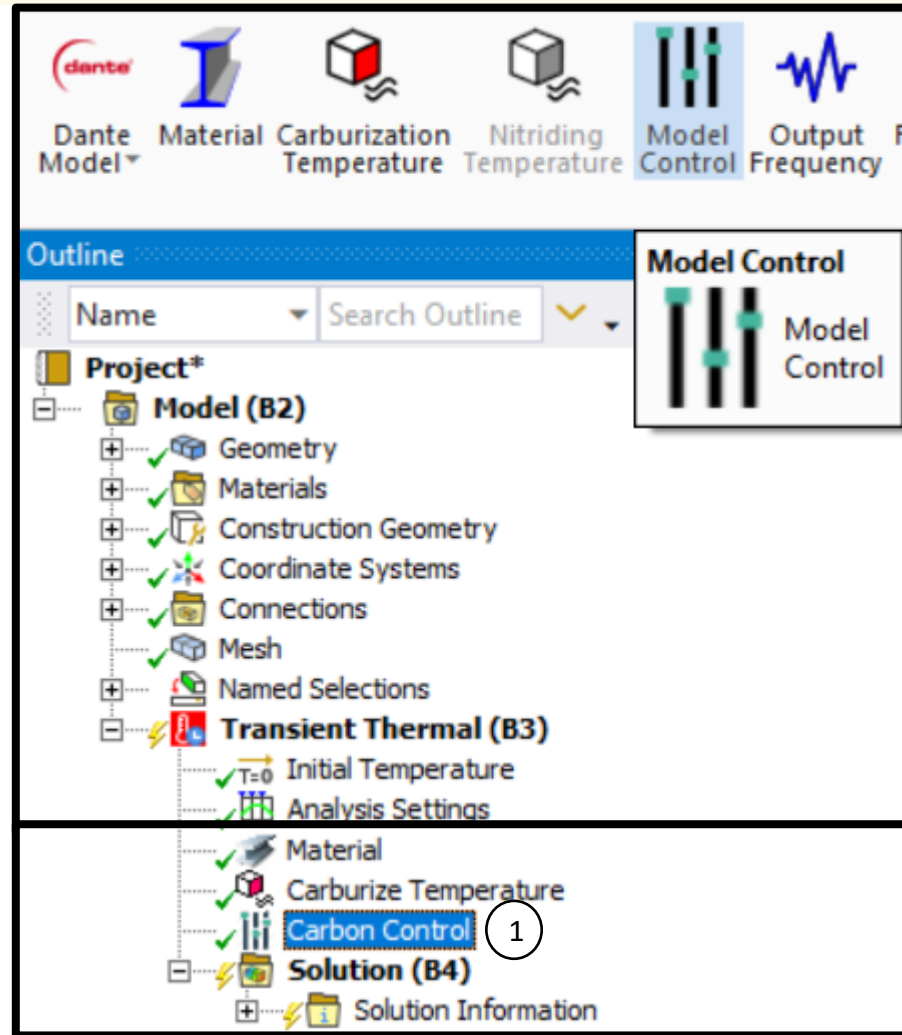


Total Time (s)	Temperature (C)
0.0	900
21600	900
21700	875
28800	875

Add intermittent time

# Step 8: Define Carbon Control

1. Select **Model Control** from the Dante toolbar to add it to Transient Thermal in the Project Tree
2. Select value for **Max. Carbon Change (Wt.frac) per substep** as 0.0005



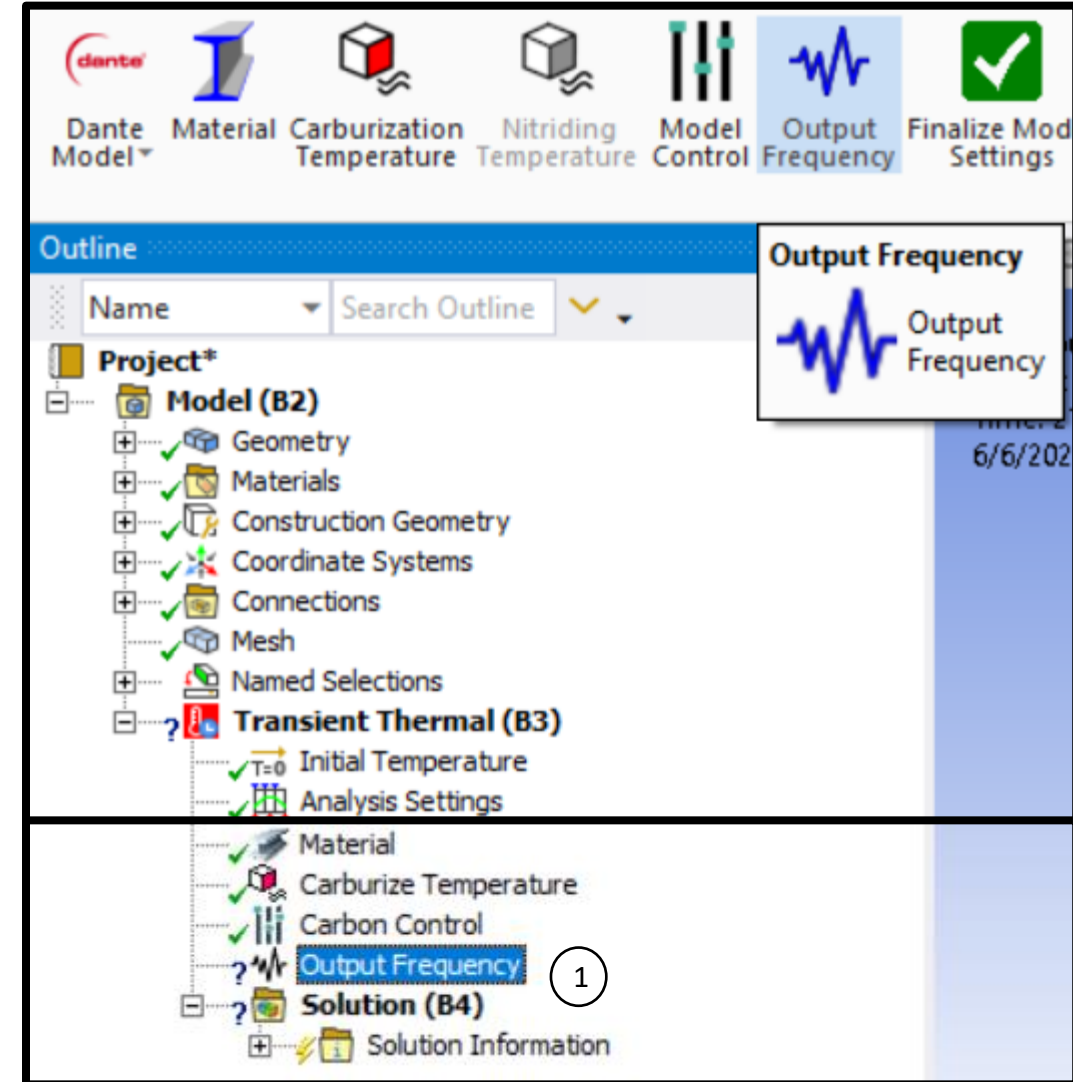
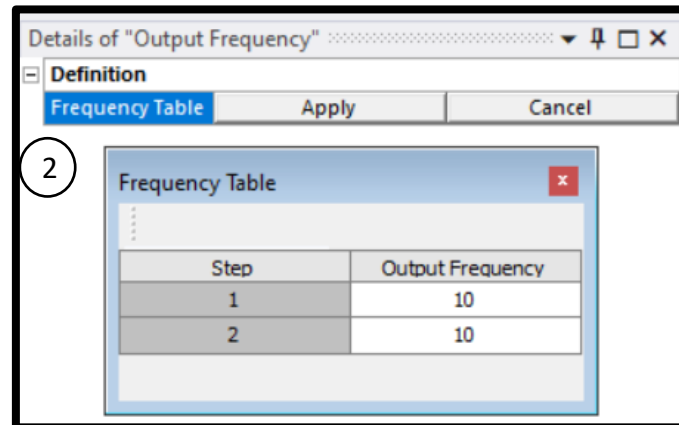
Details of "Carbon Control"	
<b>Definition</b>	
Max. carbon change (Wt. frac) per substep	0.0005
Material Directory	default

2



# Step 9: Define Output Frequency

1. Select **Output Frequency** from the Dante toolbar to add it to Transient Thermal in the Project Tree. The Output Frequency is used to define the frequency of writing simulation results to the results file
2. Click on Tabular Data to open the Frequency Table
3. The default values can be used to save values at every 10<sup>th</sup> substep, click **Apply** to finish



## Step 10: Define Carbon Boundary Condition

There are two ways to apply the carbon boundary condition, referred to as the carbon potential. The first way is by using a Temperature boundary condition. This method assumes that the surface to be carburized instantaneously reaches the carbon potential. Although this method works for most cases, this is not what happens in the actual process.

In the actual process, the heat of the furnace and the surface of the component cause dissociation of the carbon atoms from the carrier gas. The carbon atoms can then begin diffusing into the austenitic matrix once they have dissociated from the carrier gas. Although this process is relatively fast, it is not instantaneous.

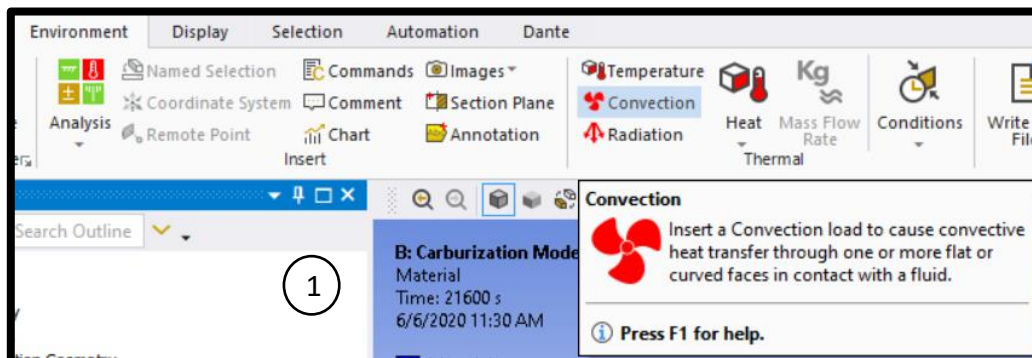
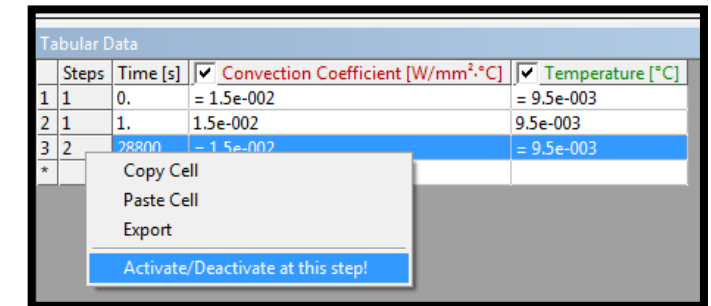
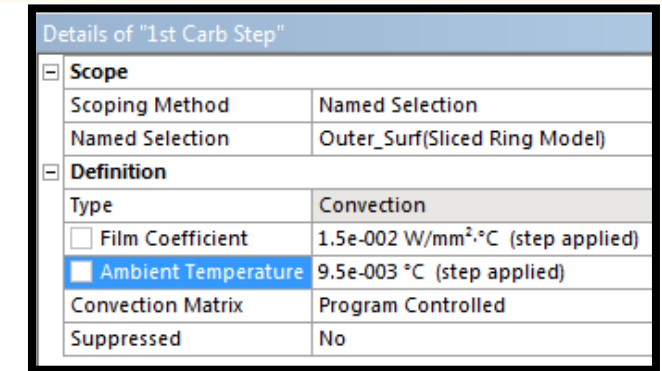
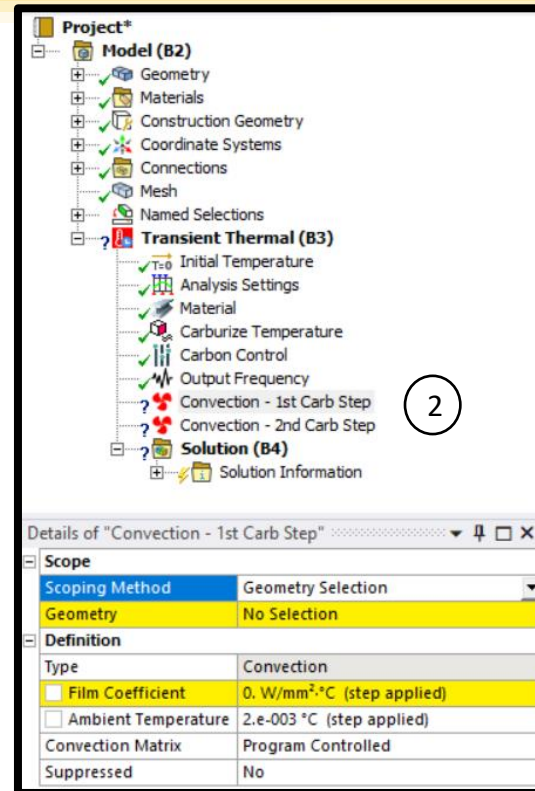
The other, and preferred, method is to use Convection. This method allows the carbon to ramp up to the carbon potential on the surface and represents the actual process much better. The ramping of carbon on the surface is still relatively fast and can be controlled by adjusting the convection coefficient. This is the method used here, with the steps explained on the next page.

# Step 10b: Define Carbon Boundary Conditions

1. From the **Environment** toolbar, add 2 **Convections** to Transient Thermal for the 2 carburizing steps
2. Rename the 2 Convections; 1<sup>st</sup> Carb Step & 2<sup>nd</sup> Carb Step

Proceed through the next 2 steps 1 BC at a time

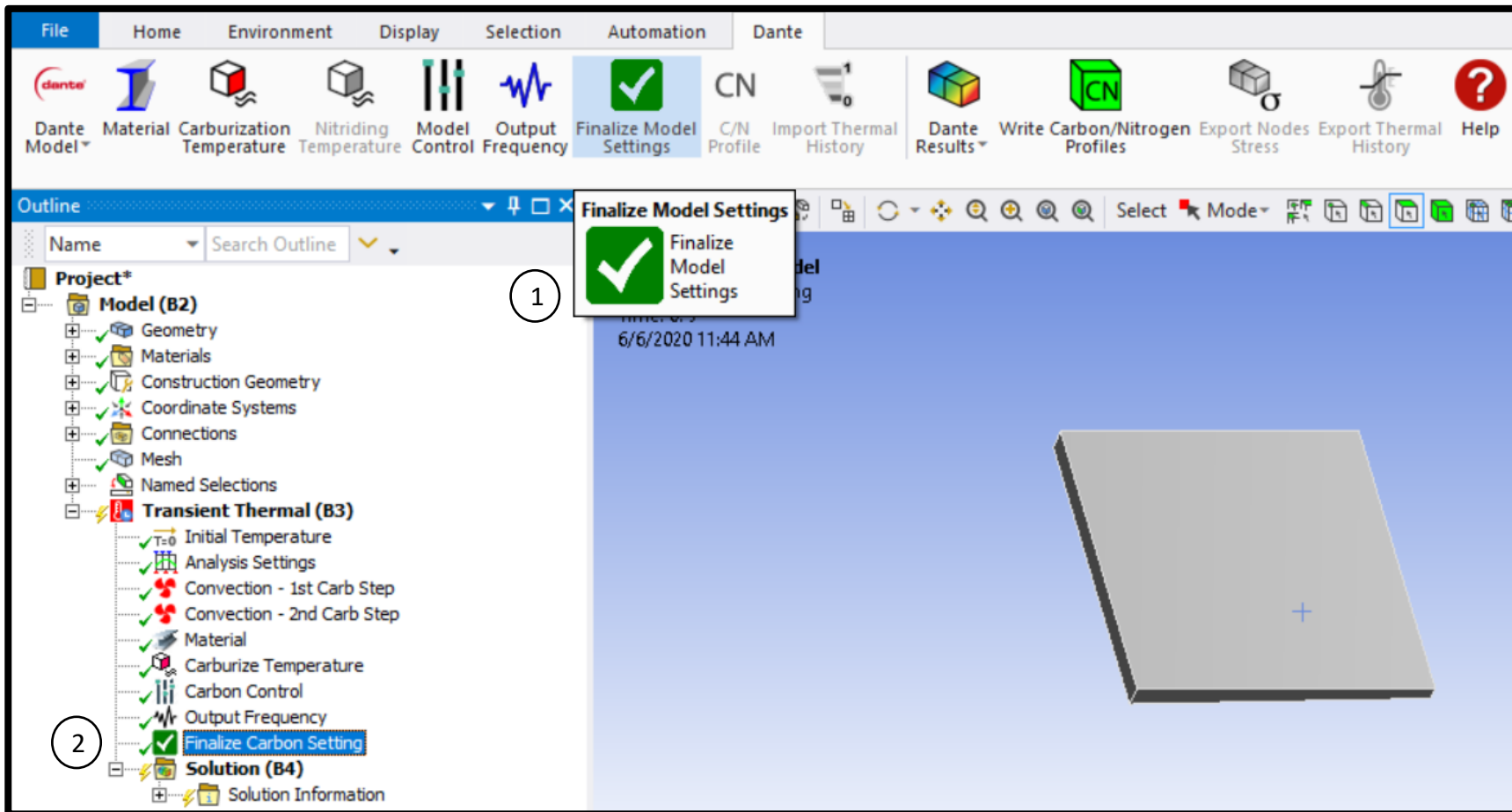
3. Fill out the Details section with the information shown in the table
4. To apply the BC to the desired step, the other step(s) must be deactivated. In Tabular Data, highlight the step(s) to be deactivated by clicking on the step number. Right click and choose Activate/Deactivate At This Step!



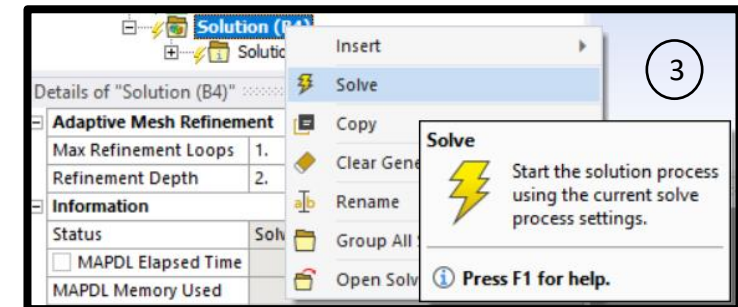
Step	Scoping Method	Geometry	Film Coefficient	Ambient Temperature (Carbon Potential Weight Fraction)	Convection Matrix
1st Carb Step	Named Selection	Outer_Surf	1.50E-02	9.5e-03	Program Controlled
2nd Carb Step	Named Selection	Outer_Surf	1.50E-02	8.0e-03	Program Controlled

# Step 11: Running the Carburization Model

1. Select **Finalize Model Settings** from the Dante toolbar to add it to Transient Thermal in the Project Tree
2. This will perform a check on the carburization transient thermal model tree. If the model is set up properly, a green check mark will be displayed next to the **Finalize Carbon Setting** in the model tree indicating that the model is ready to run



3. Be sure to save the model, right click **Solution** in the Project Tree and **Solve** to run the model

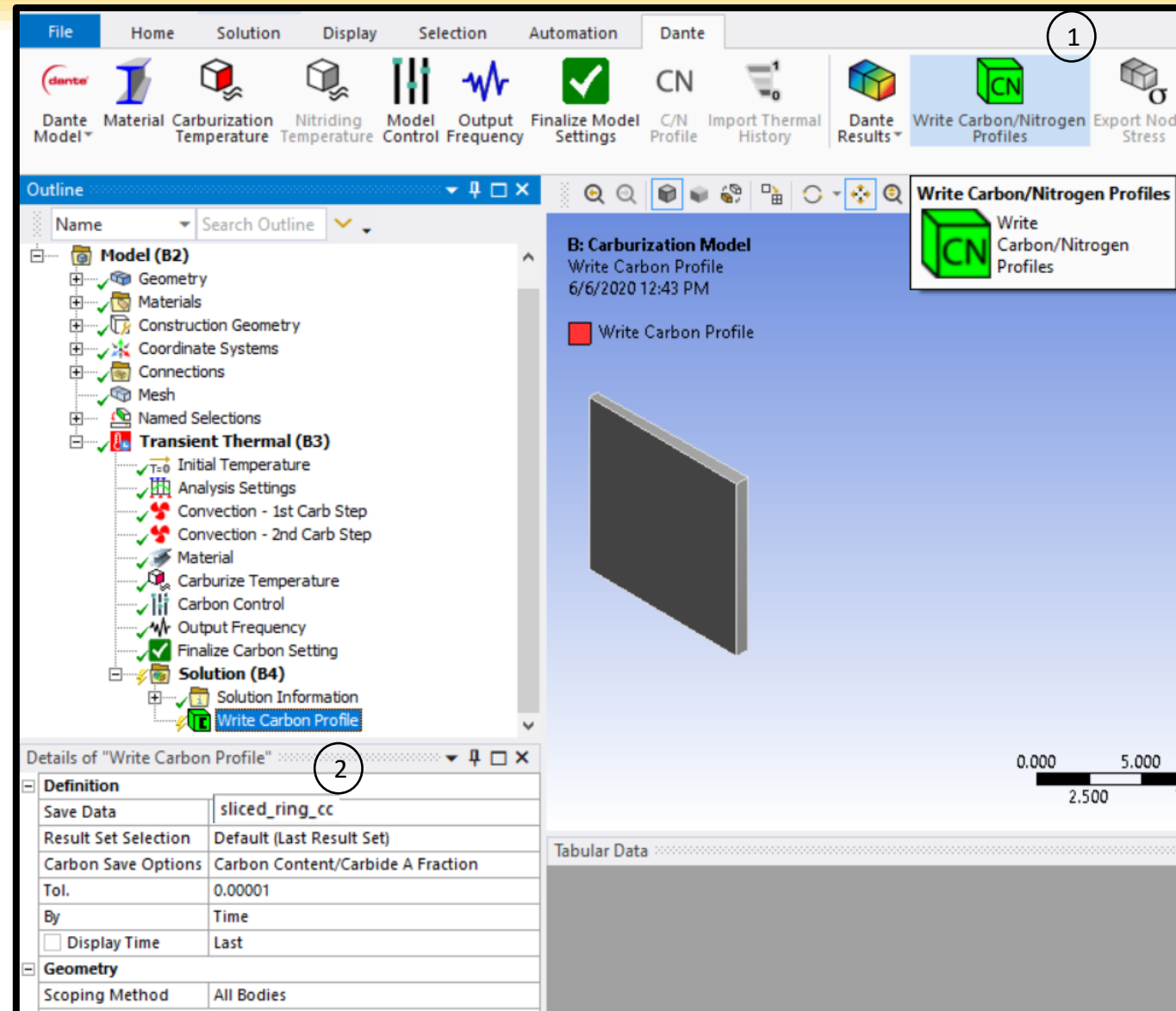




# Step 12: Carburization Model Post-Processing, Write Carbon Profile

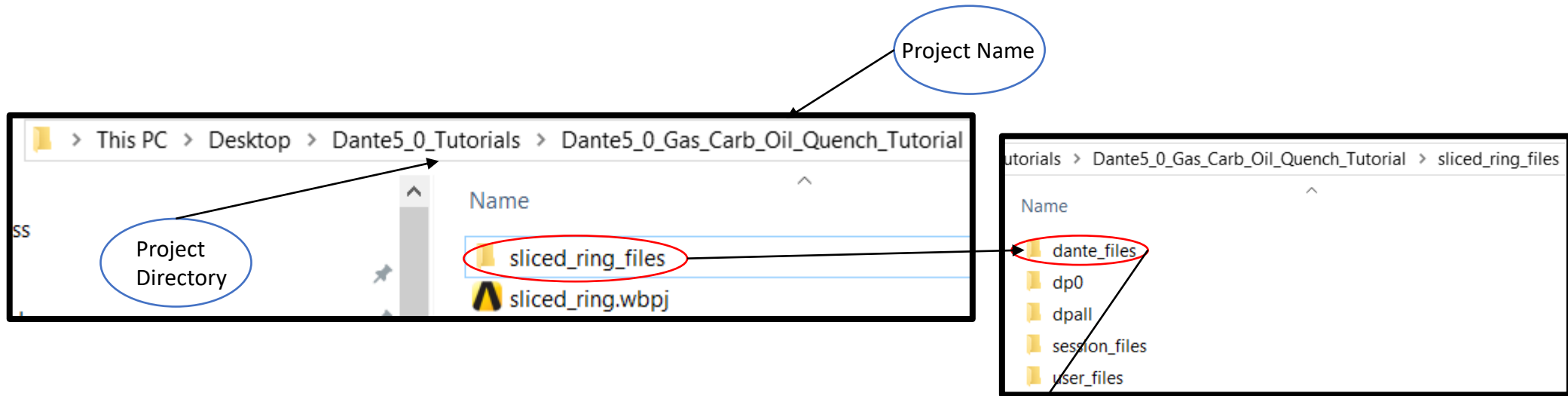
1. Select **Write Carbon Profile** from the Dante toolbar, and the Write Carbon Profile is added to the Solution
2. Under the Details of “Write Carbon Profile”, change the file name to “**sliced\_ring\_cc**” for the Save Data. The default result set is from the final frame of the solution, but the user can select the carbon data from any saved frame
3. Right click **Write Carbon Profile** under Solution in the Project Tree, and select **Evaluate All Results** to write the carbon profile file. The file is then saved in the Project directory (the directory in which you saved your current Project.) with a folder name as “dante\_files” as a .cbn file (see the next slide)

**NOTE: We will need to find this file later, so knowing what directory it is in is important.**

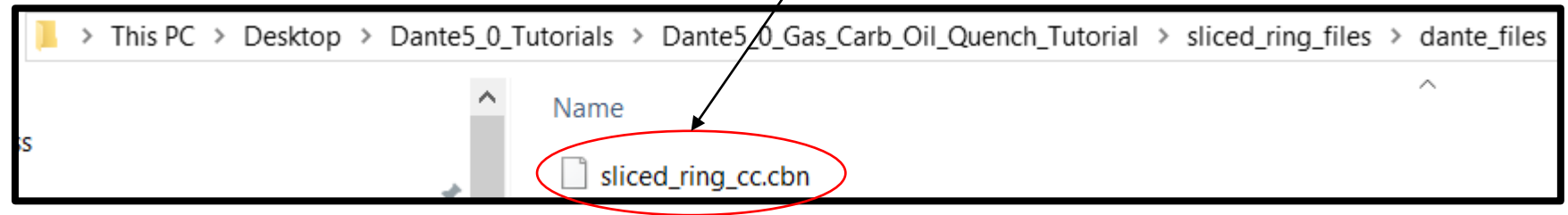




## Step 12b: DANTE Carbon Profile Directory

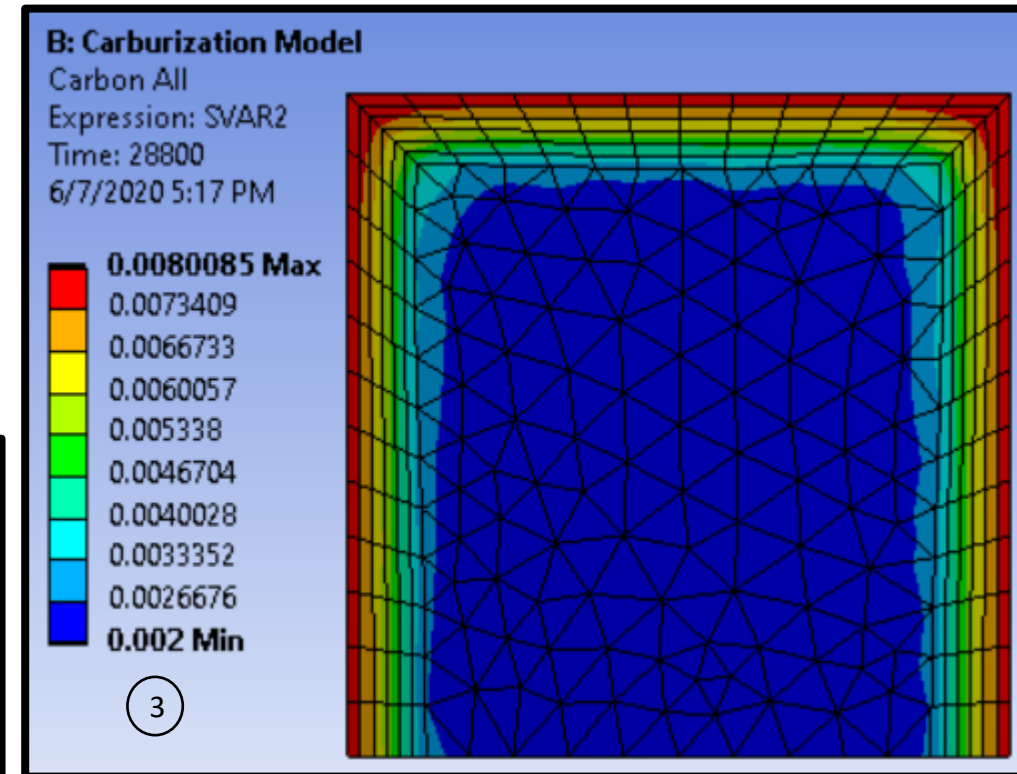
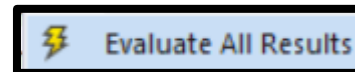
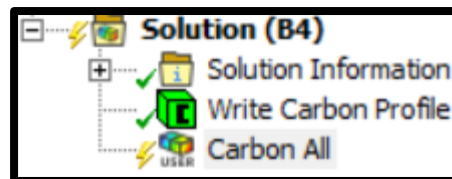
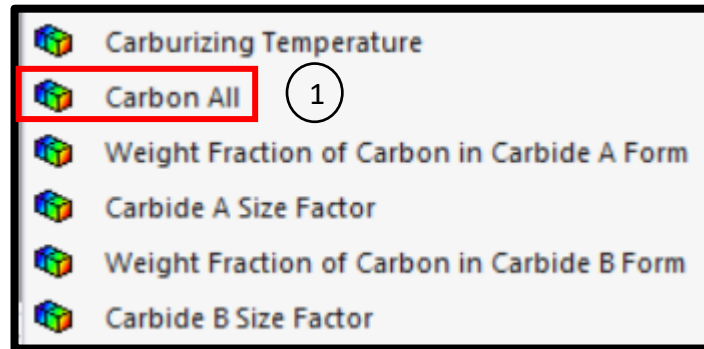


Saved file in Project directory (the directory in which you saved your current Project) with a folder name as “dante\_files” as a .cbn file



# Step 13: Carburization Model Post-Processing, Carbon Distribution Visualization

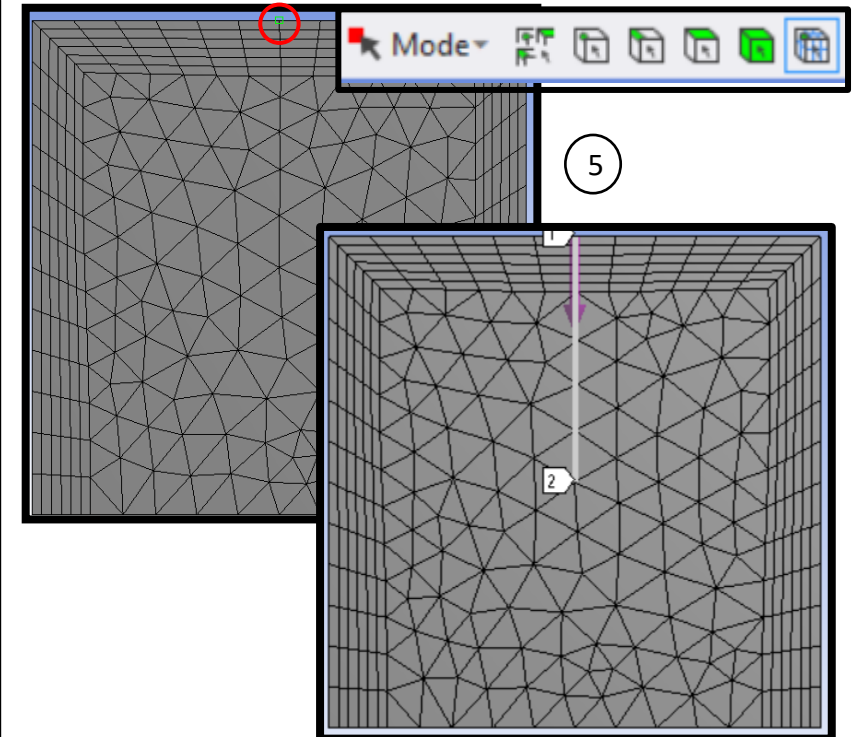
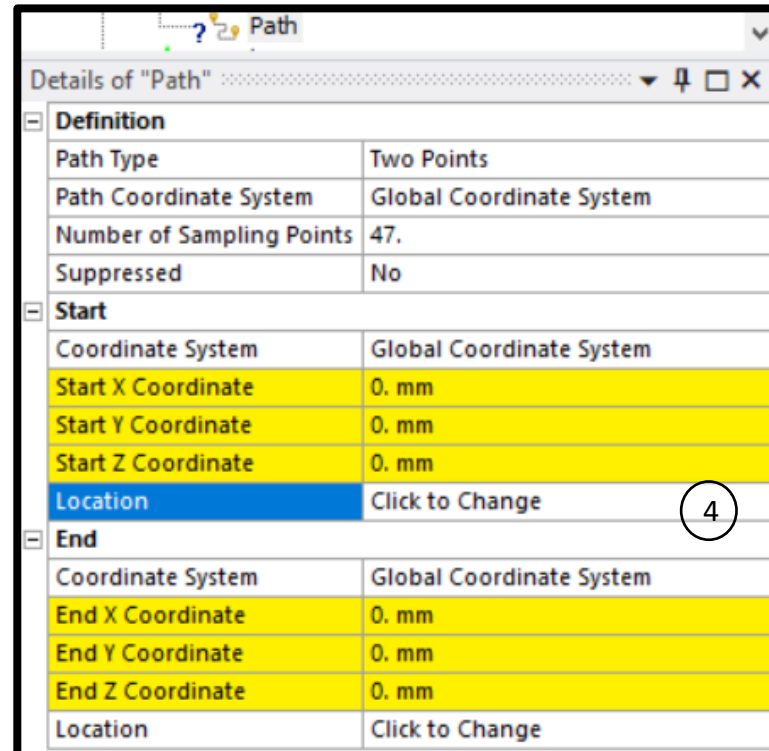
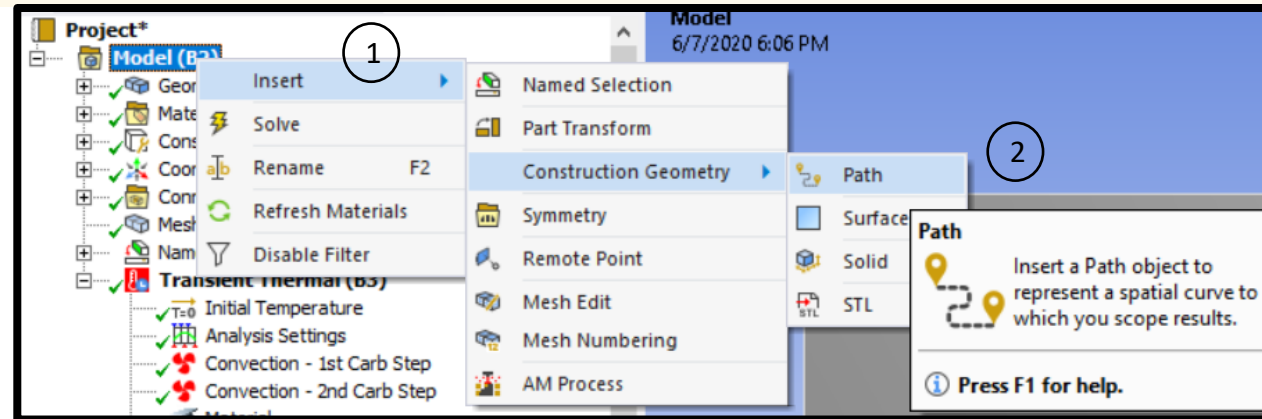
1. Under **Dante Results**, the six solution variables can be viewed in the Carburization model, select **Carbon All** to visualize the carbon distribution in the part
2. A **Carbon All** solution object will appear under Solution, right click and select **Evaluate All Results**
3. Checking the result, the Carbon content contours will appear. Here the max Carbon content on the surface is around 0.8% at the end of the simulation as was specified in the carburization schedule



**NOTE:** These DANTE results can be added before running the model, when solving the model, they will be evaluated along with it

# Step 14: Carburization Model Post-Processing, Carbon Case Depth

1. Right click on **Model** in the Project Tree and Insert a **Construction Geometry**
2. Add a **Path** to the Construction Geometry
3. Choose the **Node Selection** method and select Click to Change for the Location entry under Start in Details of "Path"
4. Pick the first node of the Path; this will define the zero position on the x-axis of the plot. Click Apply in the Details of "Path" under Start
5. For the End entry, a point in the core is selected and this path will be used to plot solution variables such as carbon from case to core. This path can be renamed to Case-Core Path

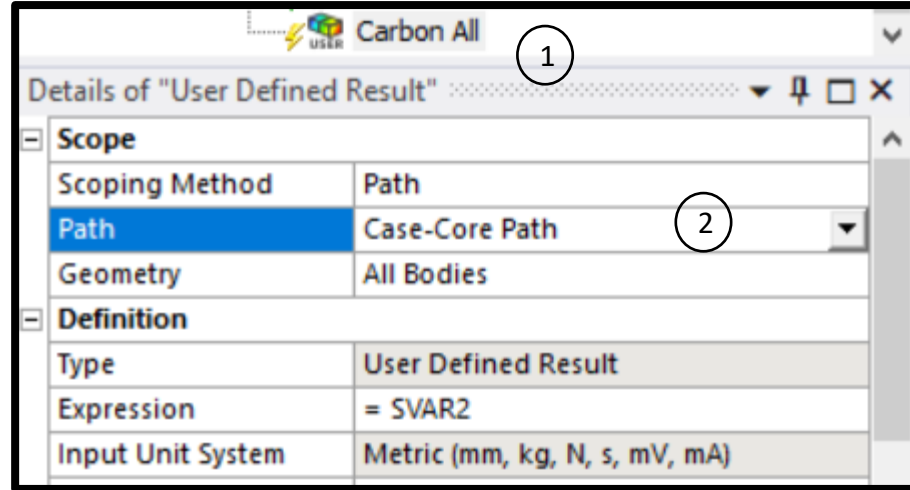




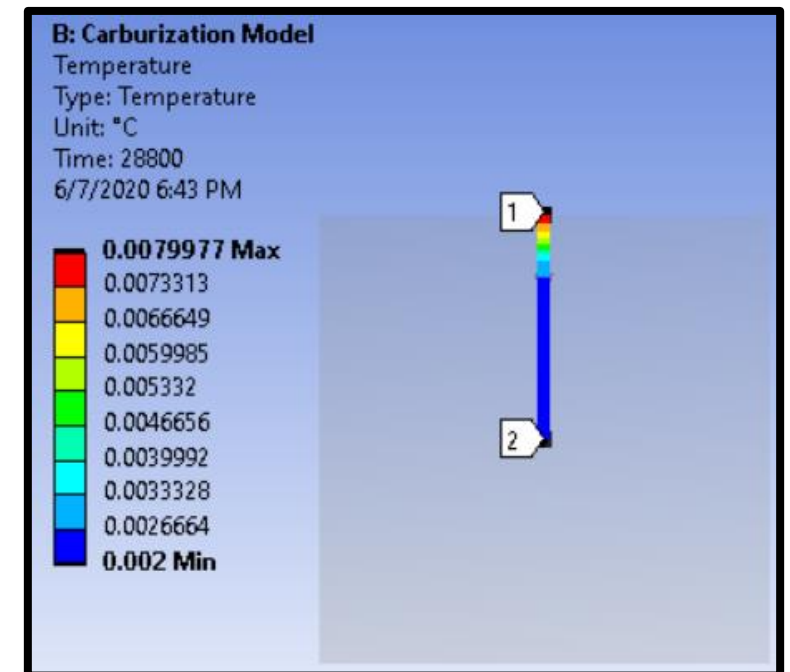
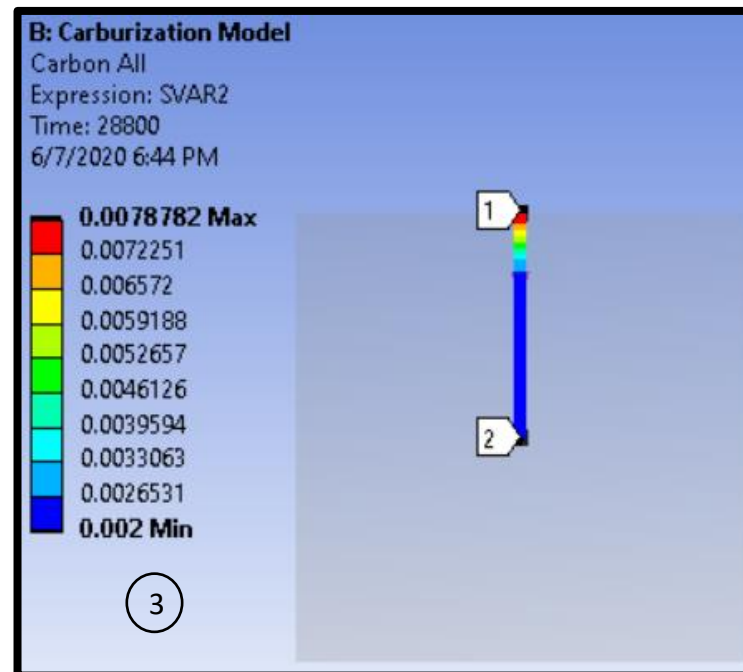
# Step 14b: Plotting the Carbon Profile

- Since the carbon is represented by temperature, we can plot the temperature or use the Dante Solution **"Carbon All"** to get the carbon profile

1. Insert a **"Carbon All"** from the Dante Solutions dropdown
2. In the Details of "Carbon All", change the **Scoping Method to Path**. Change the Path to the path to be plotted; in this case the path name is "Case-Core Path"
3. Right click on the Carbon All solution object under Solution in the Project Tree and select **Evaluate All Results** to plot the Carbon profile

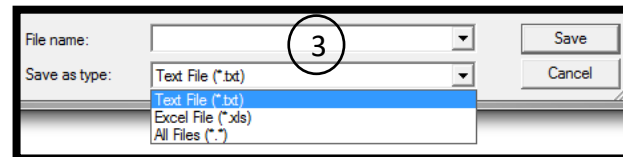
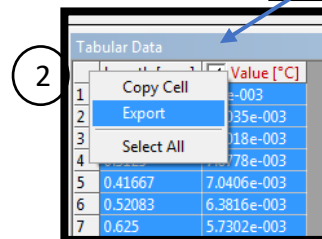
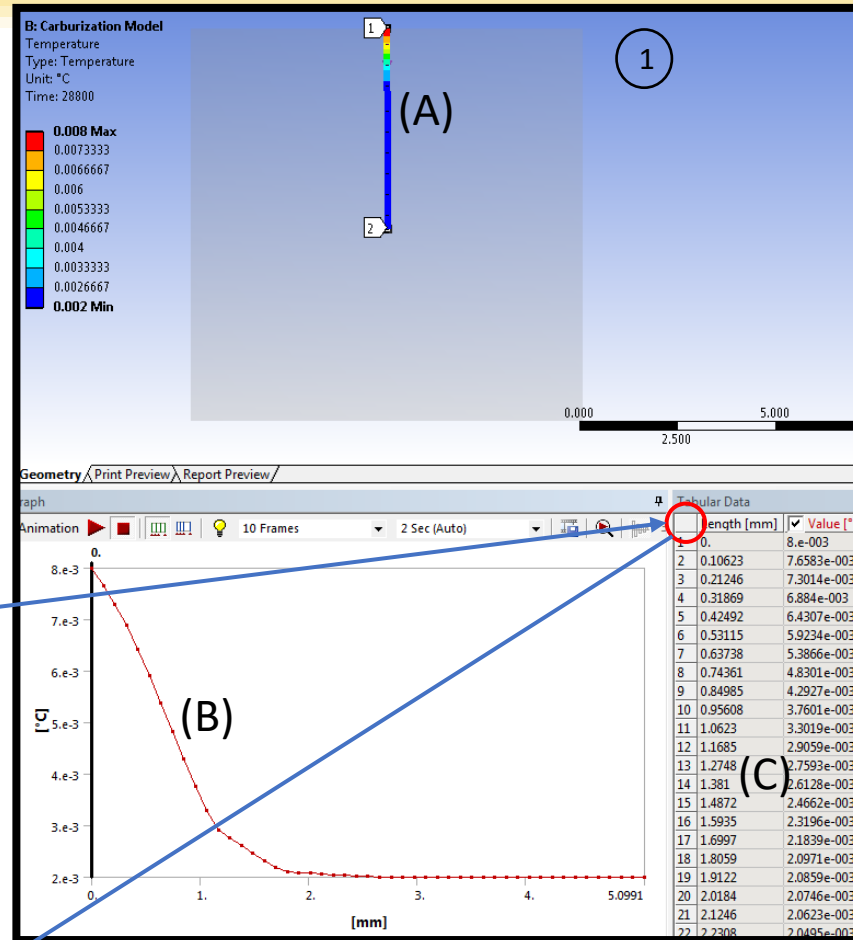


**NOTE: The Temperature and Carbon All variables should have the same values, the differences are due to numerical anomalies**



# Step 14c: Plotting the Carbon Profile

1. Shown is the results of the path plot. A) Elements used in the path plot are shown in the contour plot. B) The value at each point in the path are plotted in a line graph. C) The points used in the line graph, distance from point 1 in the path and the corresponding Carbon value, is given in tabular form
2. The values in Tabular Data, the data used to generate the line graph, can be exported to an external file. Right click in the empty box, above "1" and to the left of "Length (mm)". Select Export
3. The file can be saved in any directory as a .txt (text) or .xls (Excel) file



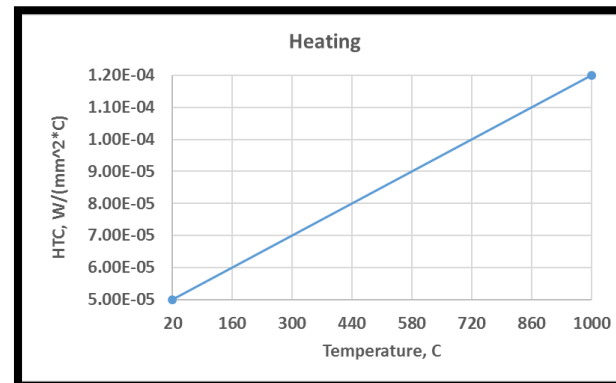
**That concludes the Carburization Model**  
**Save the Project and close Ansys Mechanical**  
**The Thermal Model can now be built**

# Quench Hardening Thermal Model

# Heating & Transfer Process Descriptions

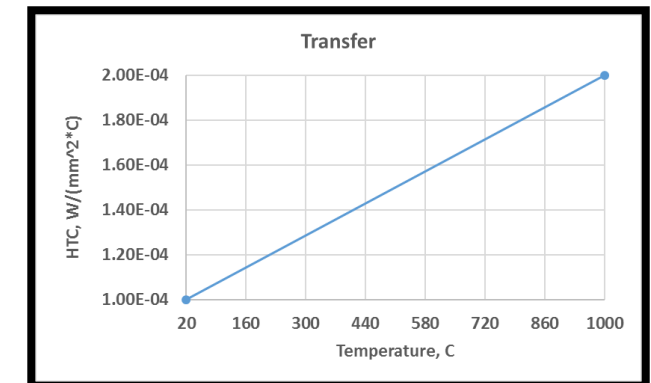
## Heating

- The ring is heated in a furnace:
  - Time in furnace: 1800 s
  - Furnace temperature: 900 °C
- Thermal boundary conditions:
  - Heat transfer coefficient (HTC) as a function of temperature and ambient temperature are used to define the thermal boundary conditions
  - The heat transfer coefficient is part surface temperature dependent, and it is defined in the figure below



## Transfer

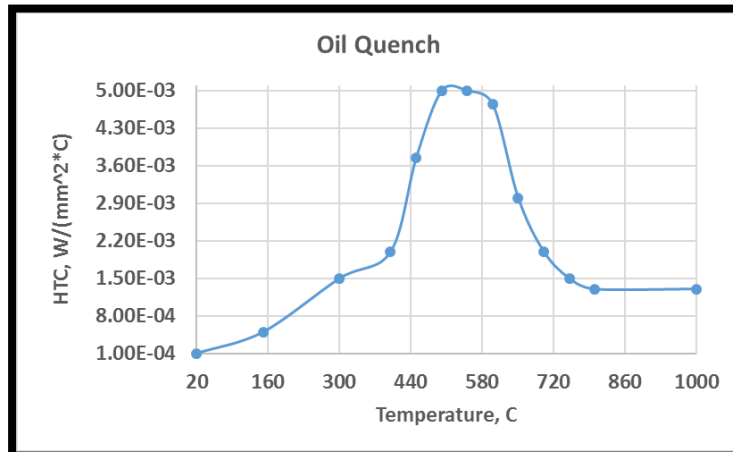
- After carburization, the furnace is opened, and the ring is transferred from the furnace to the oil quench tank
- Air transfer process:
  - Transfer duration: 10 s
  - Ambient temperature: 400 °C
  - HTC is a function of temperature and is part surface temperature dependent. The HTC is plotted below



# Oil Quench & Air Cool Process Descriptions

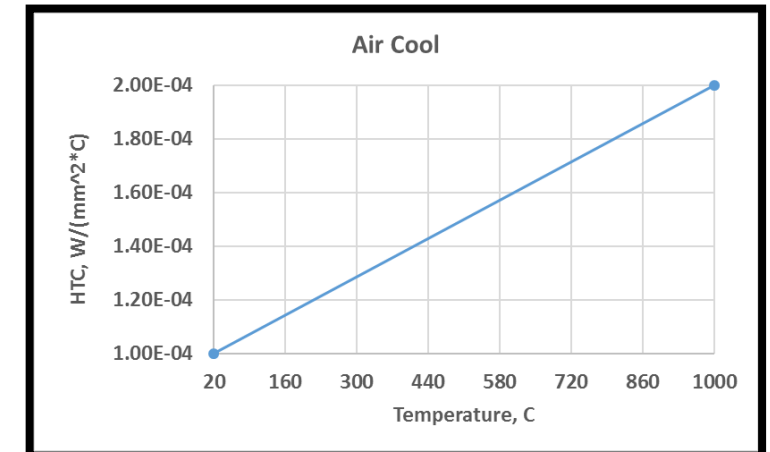
## Oil Quench

- During oil quench, three typical phases exist between the liquid and solid interface: vapor blanket, nucleate boiling, and convective cooling. The three phases are shown in the heat transfer coefficient plot in terms of part surface temperature.
- Oil quench process:
  - Time in oil tank: 300 s
  - Ambient temperature: 65 °C
  - Heat transfer coefficient is in terms of the part surface temperature, as shown below



## Air Cool

- After oil quench, the part is taken out of the oil tank, and cooled in open air to room temperature
- Air cool process:
  - Time duration: 3600 s
  - Ambient temperature: 20  $^\circ C$
  - Heat transfer coefficient shown below



# Deep Freeze & Temper Process Descriptions

## Deep Freeze

- Deep Freezing is done to make sure there is no retained Austenite during quenching
- Deep Freeze process:
  - Time duration: 7200 s
  - Ambient temperature: -30 °C
  - Heat transfer coefficient is 0.00005 W/mm<sup>2</sup> °C

## Temper

- Tempering is usually a post-quenching or post hardening treatment. It is done to relieve internal stresses, decrease brittleness, improve ductility and toughness
- Temper process:
  - Time duration: 7200 s
  - Ambient temperature: 150 °C
  - Heat transfer coefficient is 0.00005 W/mm<sup>2</sup> °C

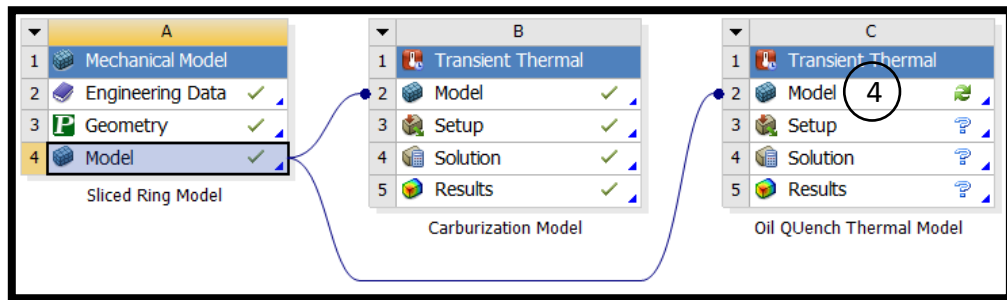
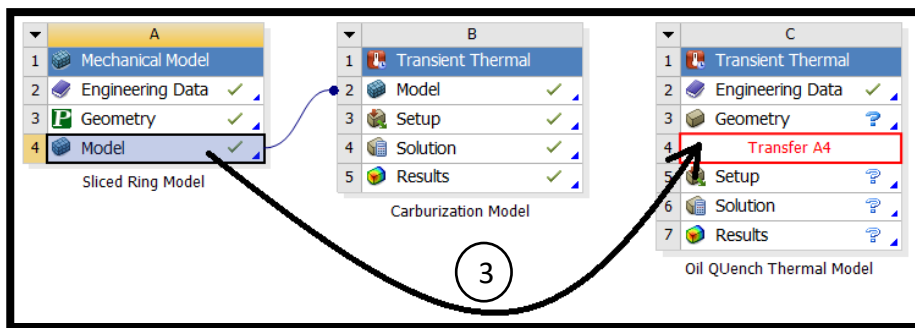
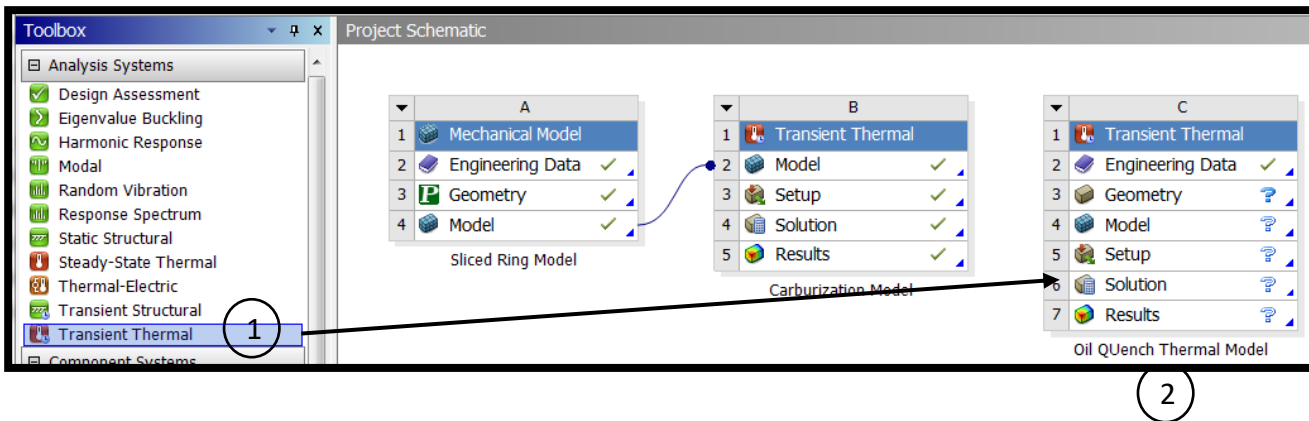
# Step 1: Thermal Model Setup, Add Analysis System to Project Schematic

1. Drag and drop a Transient Thermal Analysis System into the Project Schematic

2. Rename it as “Oil Quench Thermal Model”

3. Drag and drop Model from Sliced Ring Model to the Oil Quench Thermal Model

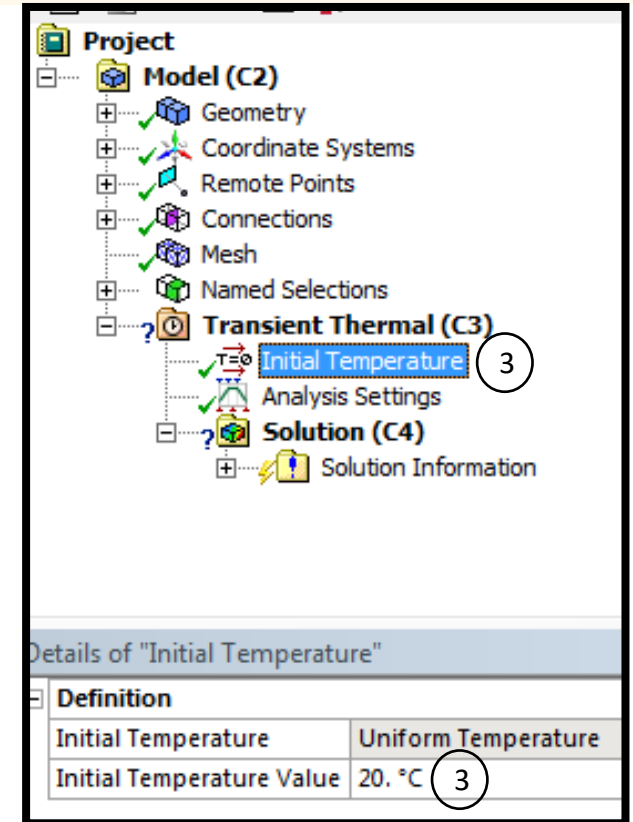
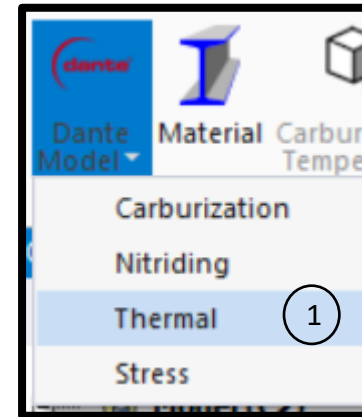
4. Double click Model in the Oil Quench Thermal Model to open Ansys Mechanical





## Step 2: Define DANTE Model & Initial Temperature

1. With Mechanical open, click on Dante Model in the Dante toolbar and select **Thermal**
2. Selecting Thermal opens the buttons needed to complete the Thermal Model. The buttons to the left of the single bar are for pre-processing (setting up the model) and the buttons to the right of the single bar are used for post-processing (viewing the results). The Question Mark (?) is a link to the Dante Help File
3. Select **Initial Temperature** under Transient Thermal in the Project Tree. In the Details of "Initial Temperature", change the Initial Temperature Value to 20.0 °C



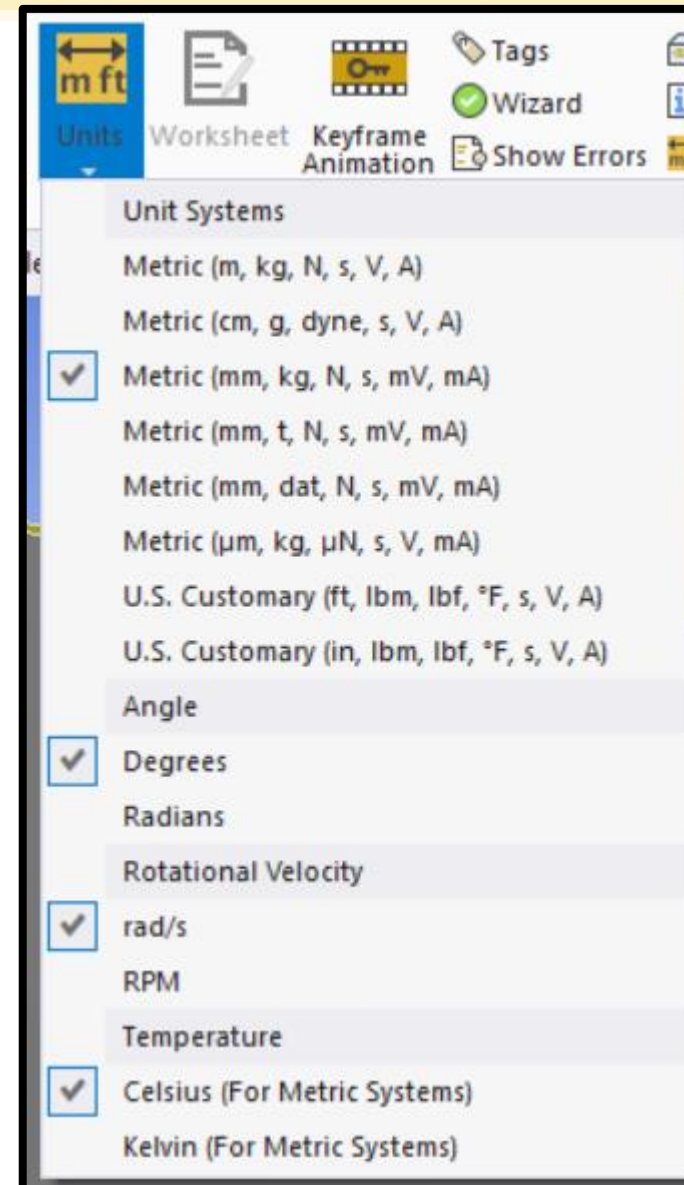
## Step 3: Define Units

It is critical that the units be properly defined

1. Click on Units in the Home toolbar
2. Select Metric (mm, kg, N, s, mV, mA)
3. Select Degrees
4. Select rad/s (This isn't critical as there is no motion defined in the heat treatment models)
5. Select Celsius (For Metric Systems)

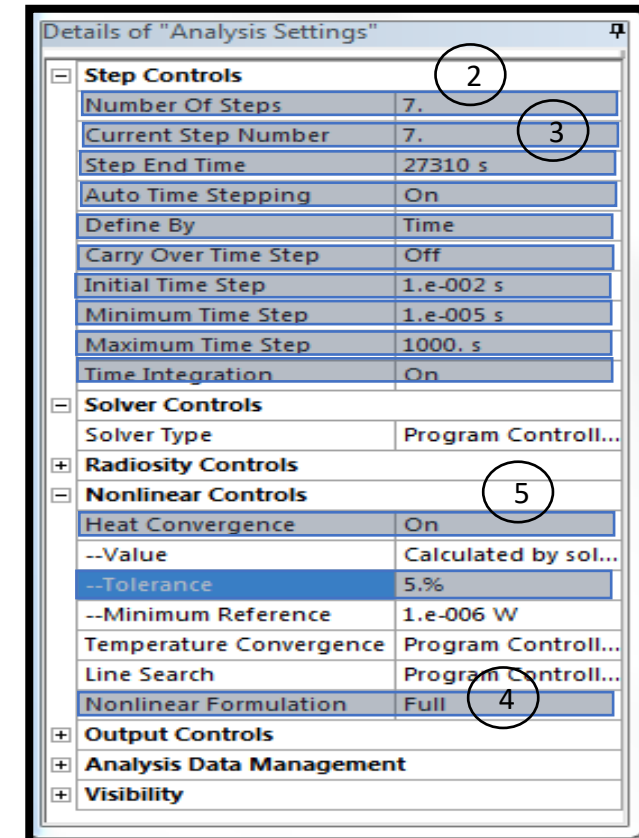
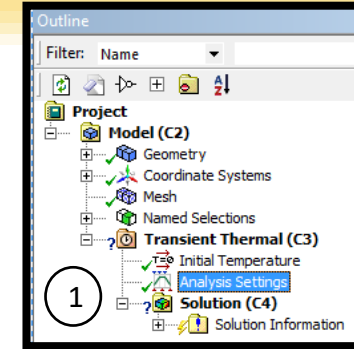
**NOTE:** It is absolutely critical that these units are selected. If different units are selected, the model may run, but the results will be WRONG.

For Example: If grams are chosen, the thermal diffusivity will be very high, and heating/cooling will occur far too quickly. If tonne is chosen, the thermal diffusivity will be very low, and heating/cooling will occur far too slowly.



## Step 4: Defined Processing Steps

1. Click on **Analysis Settings** in the Project Tree
2. In the Details of "Analysis Settings", change the Number of Steps to 7
3. Beginning with Current Step Number 7, enter the information from the table below into the Step Controls in the Details of "Analysis Steps". Also enter the rest of the steps from below table. It is necessary to work backwards with the step numbers because Ansys populates the Step End Time in 1 second increments for each step. Ansys will then reject the entry for the Step End Time if it does not progress chronologically
4. Change the **Nonlinear Formulation to Full** under Nonlinear Controls in the Details of "Analysis Settings". This is required to call the DANTE subroutines. If this step is skipped, the results will be incorrect, or the model may fail to run
5. If the thermal model fails to converge during solving, the problem can usually be traced back to the latent heat released during phase transformations. If this occurs, change the Heat Convergence under Nonlinear Controls in Details of "Analysis Settings" to On and change the Tolerance to 5%



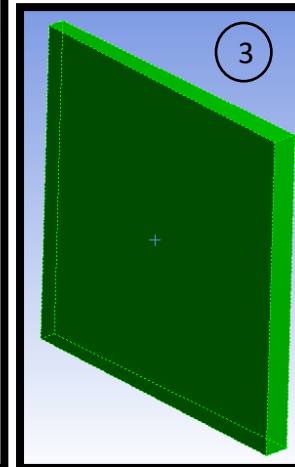
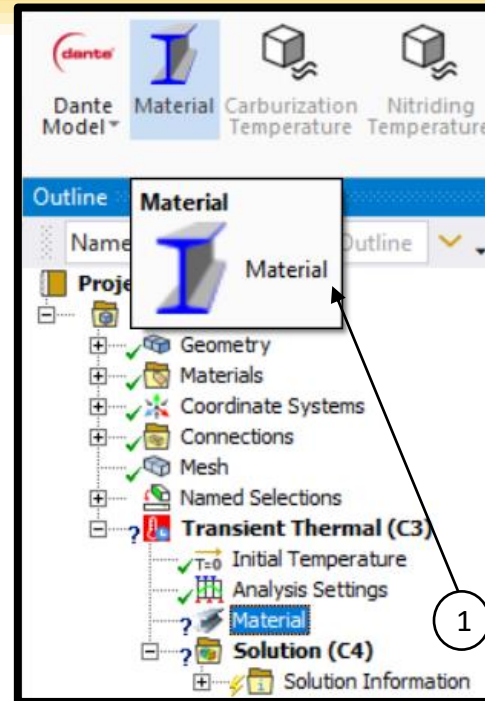
Current Step Number	Step End Time	Auto Time Stepping	Define By	Carry Over Time Step	Initial Time Step	Minimum Time Step	Maximum Time Step	Time Integration
7	27310	On	Time	Off	1.00E-02	1.00E-05	1000	On
6	20110	On	Time	Off	1.00E-02	1.00E-05	300	On
5	12910	On	Time	Off	1.00E-02	1.00E-05	300	On
4	5710	On	Time	Off	1.00E-02	1.00E-05	1000	On
3	2110	On	Time	Off	1.00E-02	1.00E-05	100	On
2	1810	On	Time	Off	1.00E-02	1.00E-05	10	On
1	1800	On	Time	N/A	1.00E-02	1.00E-05	1000	On

# Step 5: Assign Material

1. Select **Material** from the Dante toolbar to add it to Transient Thermal in the Project Tree

The following Steps apply to modifying values in the Details of "Material"

2. Click on the yellow box next to Geometry
3. Select the entire part body; performing Step 2 above activates the body selection. Therefore, just simply click on the part to select the entire body
4. Select **Apply** for Geometry
5. Change **Material Name** to S86XX for the AISI 8600 series steel
6. Change the **Carbon and Carbon in Carbide (wt )** to 0.002 to indicate AISI 8620



Geometry	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Material Name	S10XX
Density	7.8E-06 kg/mm <sup>3</sup>
Type	Thermal
Carbon and Carbon in Carbide (Wt. frac)	0.002
Carbon in Carbide (Wt. frac)	0.0
Carbide Size Factor	0.5
Nitrogen (Wt. frac)	0.0
Mat. Initial Phase	Tabular Data
Initial TMart. Temper Condition (C)	250.0
Chemical Composition	Tabular Data

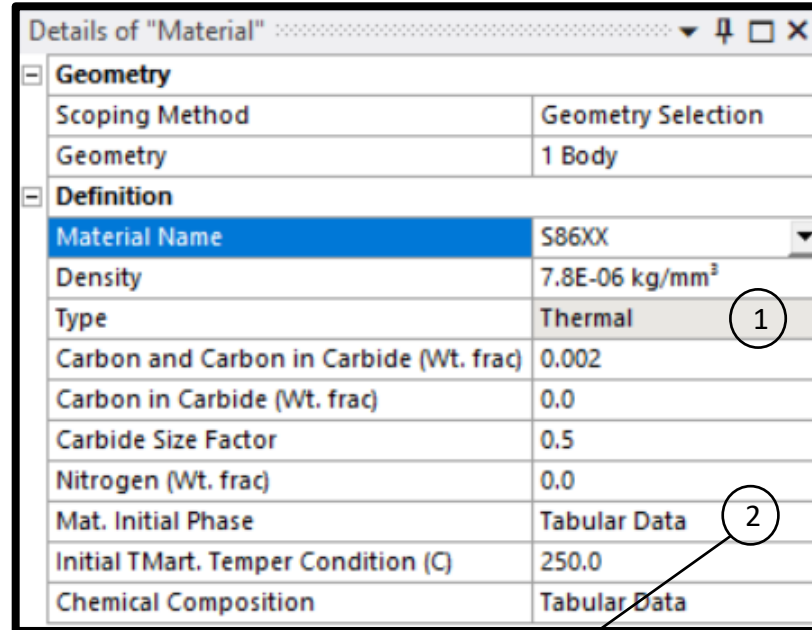
Geometry	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Material Name	S86XX
Density	7.8E-06 kg/mm <sup>3</sup>
Type	Thermal
Carbon and Carbon in Carbide (Wt. frac)	0.002
Carbon in Carbide (Wt. frac)	0.0
Carbide Size Factor	0.5
Nitrogen (Wt. frac)	0.0
Mat. Initial Phase	Tabular Data
Initial TMart. Temper Condition (C)	250.0
Chemical Composition	Tabular Data

**NOTE:** Initial microstructure and chemical composition adjustments can be made to the selected material along with the other parameters, which are all left to their default values here

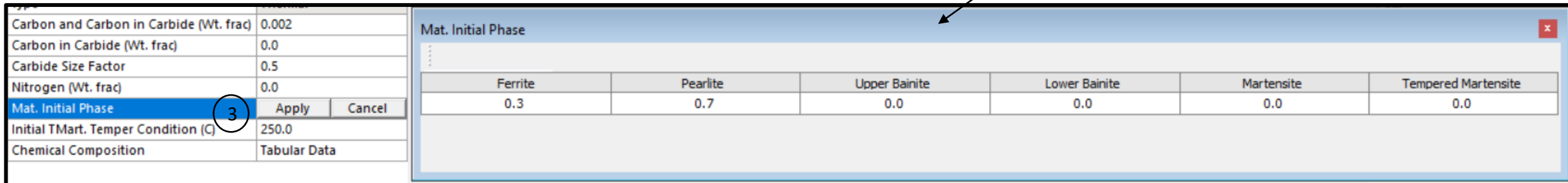
# Step 6: Define Initial Phase

The following Steps apply to modifying values in the Details of "Material"

1. The Type should be set to **Thermal**.  
This tells the user subroutine which mathematical models to use and reveals the Mat. Initial Phase option
2. Click Tabular Data for **Mat. Initial Phase** to enter the initial phases of the material
3. Click **Apply** when finished (Default values of 30% Ferrite & 70% Pearlite are fine for this exercise)



Details of "Material"	
<b>Geometry</b>	
Scoping Method	Geometry Selection
Geometry	1 Body
<b>Definition</b>	
Material Name	S86XX
Density	7.8E-06 kg/mm³
Type	Thermal (1)
Carbon and Carbon in Carbide (Wt. frac)	0.002
Carbon in Carbide (Wt. frac)	0.0
Carbide Size Factor	0.5
Nitrogen (Wt. frac)	0.0
Mat. Initial Phase	Tabular Data (2)
Initial TMart. Temper Condition (C)	250.0
Chemical Composition	Tabular Data



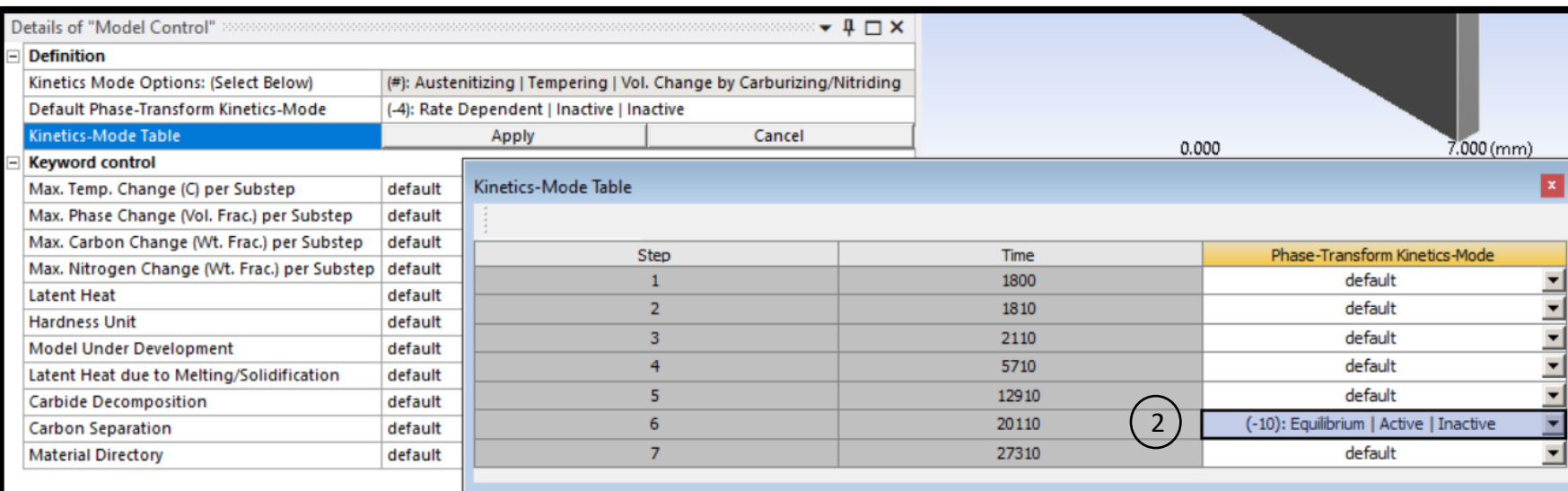
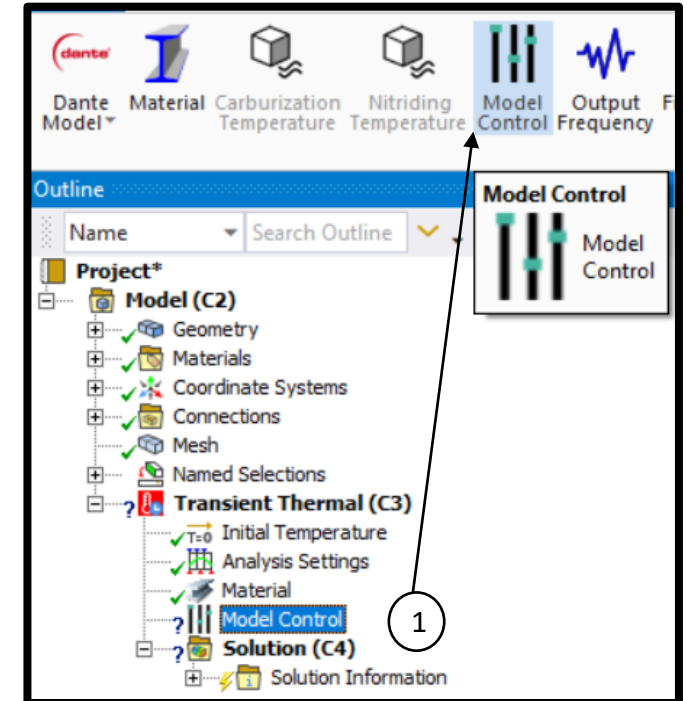
Mat. Initial Phase					
Ferrite	Pearlite	Upper Bainite	Lower Bainite	Martensite	Tempered Martensite
0.3	0.7	0.0	0.0	0.0	0.0



# Step 7: Define Control File

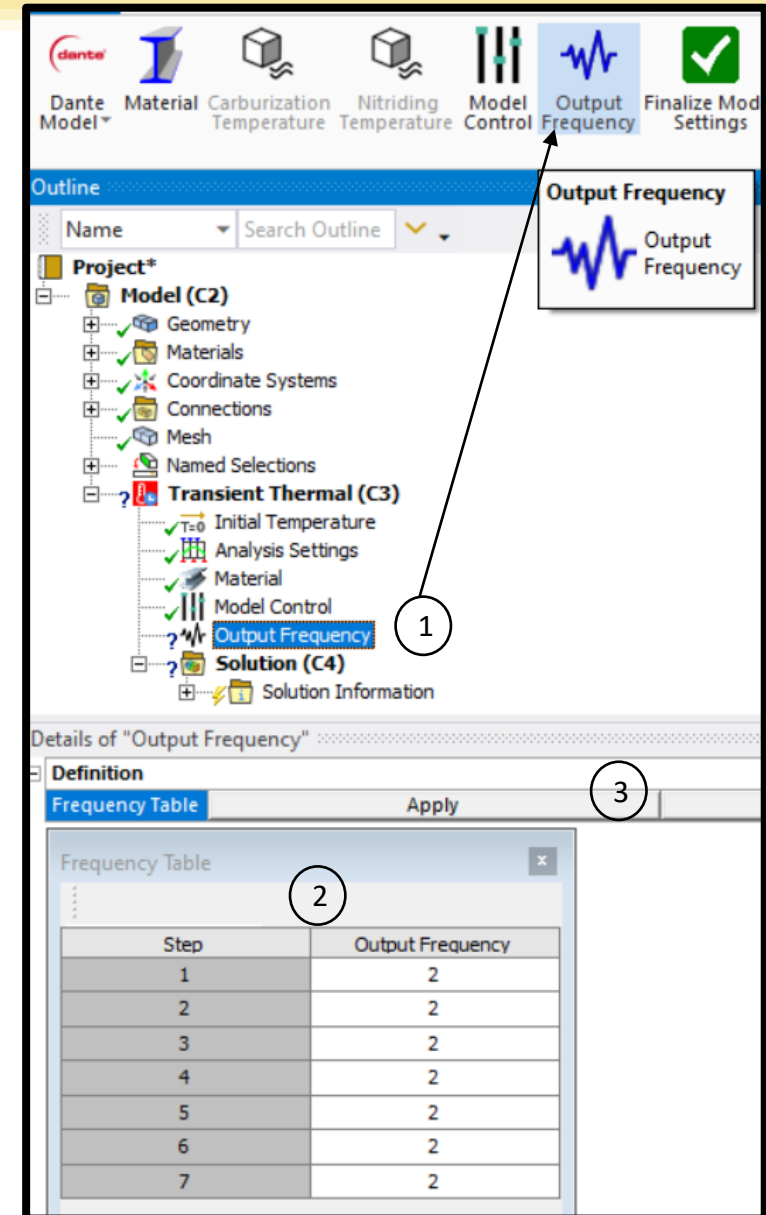
The Control File allows the user to set DANTE control keywords for the project such as the phase transformation kinetics mode, max temperature and max phase transformation changes per increment, etc.

1. Select **Model Control** in the Dante toolbar to add it to Transient Thermal in the Project Tree
2. In the Details of "Control File", review the **Kinetics-Mode Table** by clicking on Tabular Data, leave the Phase Transformation Kinetics Mode at **-4.0** (this indicates rate-based austenitization kinetics; see the help file for an explanation of this value). Step 6 should be set to **(-10)** to activate the tempering kinetics, leave all other control keyword parameters to default



# Step 8: Define Output Frequency

1. Select **Output Frequency** from the Dante toolbar to add it to Transient Thermal in the Project Tree. The Output Frequency is used to define the frequency of writing simulation results to the results file
2. In Details of “Output Frequency”, click on Tabular Data to review the contents of the **Frequency Table** entries. For a thermal model, the default frequency values should be set to 2
3. Click **Apply** when done



Details of "Output Frequency"

Definition

Frequency Table Apply

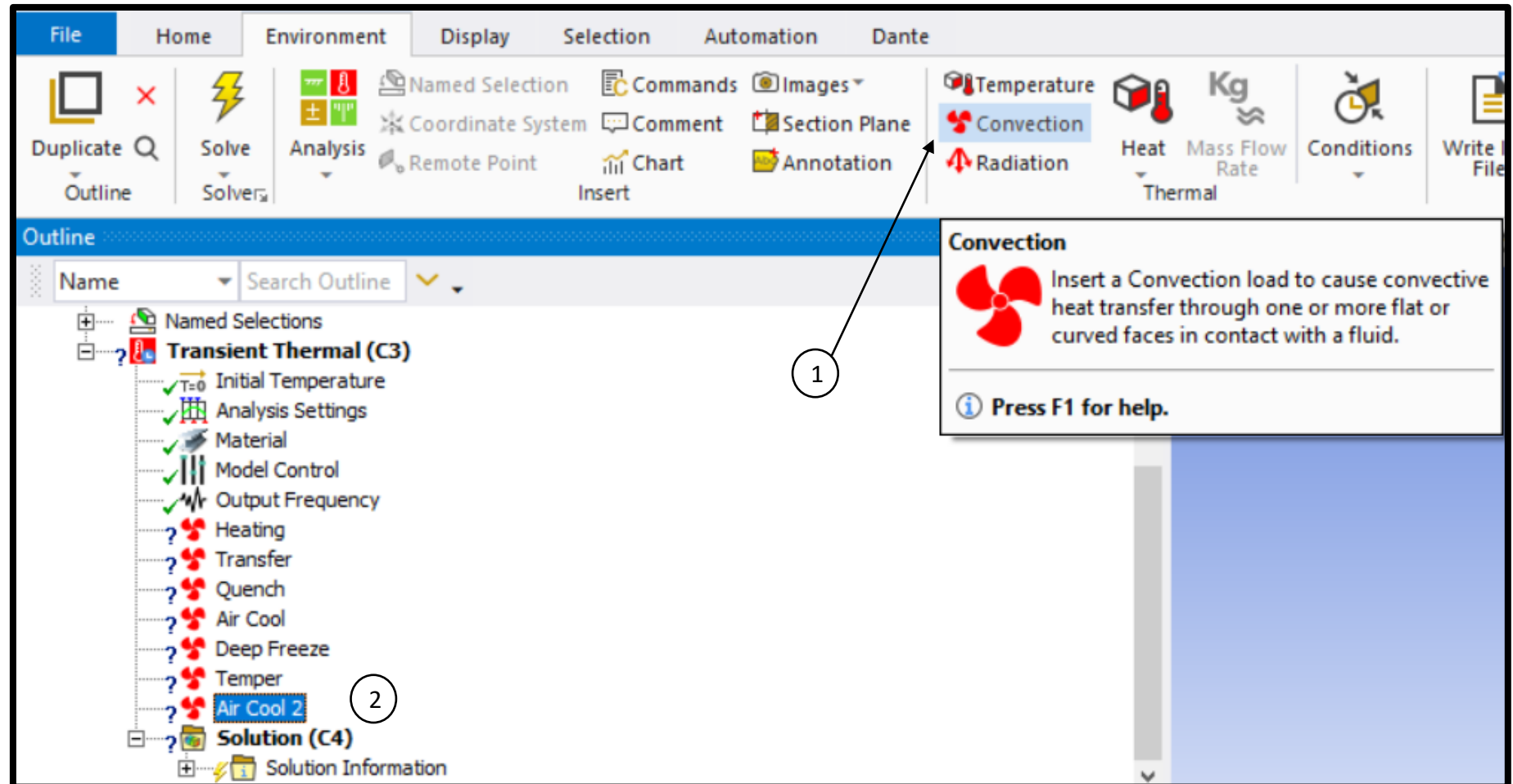
Frequency Table

Step	Output Frequency
1	2
2	2
3	2
4	2
5	2
6	2
7	2

# Step 9: Defined Thermal Boundary Conditions

Convection coefficients are used to define the thermal boundary conditions in the Thermal Model.

1. Add 7 **Convections** from the Environment toolbar to Transient Thermal in the Project Tree
2. Rename the Convections as Heating, Transfer, Quench, Air Cool, Deep Freeze, Temper and Air Cool 2 as shown. Having a name associated with them helps when defining their properties in the next steps

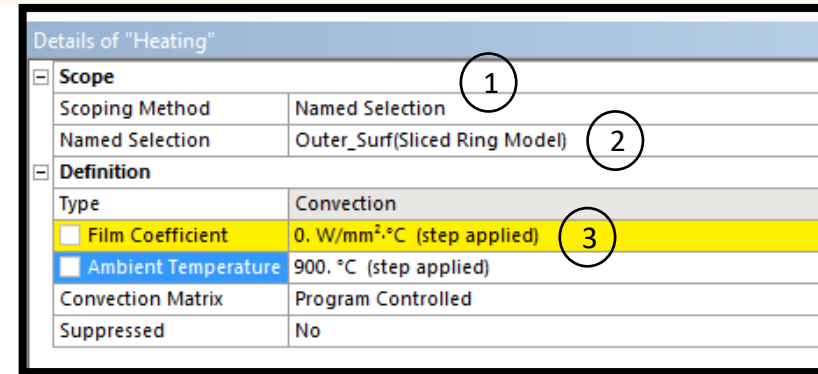




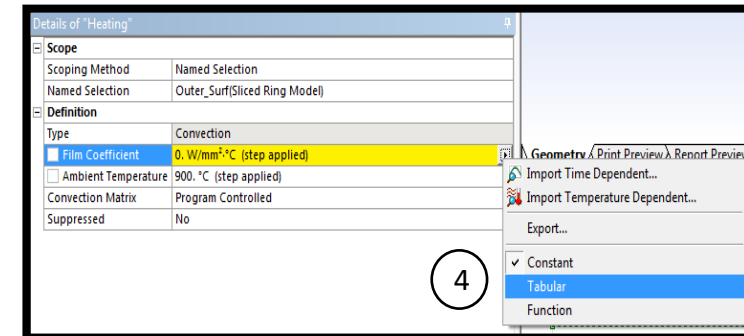
# Step 10: Define Thermal Boundary Conditions

It is helpful to perform the following steps in the same order for each Convection definition. Steps 1-4 are performed within options located in the Details section. Start with Heating and perform the following steps on this page and the next page for each Convection definition.

1. Change the **Scoping Method** to Named Selection
2. Choose the **Named Selection** for which the Convection is to be applied; in this case the name is Outer\_Surf for all Convection definitions. The surfaces for the Named Selection are then highlighted in yellow.
3. Change the **Ambient Temperature** to the desired value. For Heating, (see the table on slide no.60) the temperature is 900 °C
4. Expand the **Film Coefficient** option and choose **Tabular**
5. To apply the BC only to the active step, Step #1 in this case, the other steps must be deactivated. In **Tabular Data**, highlight the steps to be deactivated by clicking on the step number. Right click and choose **Activate/Deactivate At This Step!**

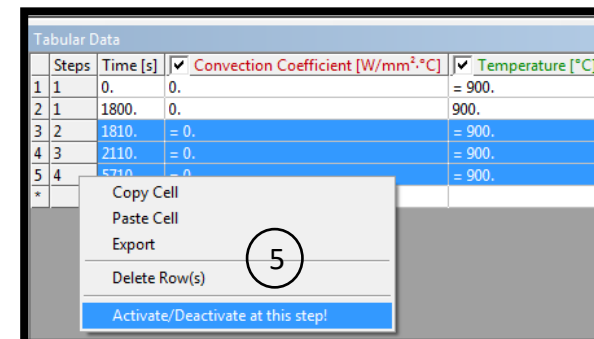


Details of "Heating"	
<b>Scope</b>	
Scoping Method	Named Selection (1)
Named Selection	Outer_Surf(Sliced Ring Model) (2)
<b>Definition</b>	
Type	Convection
<input checked="" type="checkbox"/> Film Coefficient	0. W/mm²·°C (step applied) (3)
<input type="checkbox"/> Ambient Temperature	900. °C (step applied)
Convection Matrix	Program Controlled
Suppressed	No



Details of "Heating"	
<b>Scope</b>	
Scoping Method	Named Selection
Named Selection	Outer_Surf(Sliced Ring Model)
<b>Definition</b>	
Type	Convection
<input checked="" type="checkbox"/> Film Coefficient	0. W/mm²·°C (step applied)
<input type="checkbox"/> Ambient Temperature	900. °C (step applied)
Convection Matrix	Program Controlled
Suppressed	No

4



Steps	Time [s]	Convection Coefficient [W/mm²·°C]	Temperature [°C]
1	0.	0.	= 900.
2	1800.	0.	900.
3	1810.	= 0.	= 900.
4	2110.	= 0.	= 900.
5	5710.	= 0.	= 900.

5

# Step 10b: Defined Thermal Boundary Conditions

Continuing with the Heating definition ...

1. Expand **Independent Variable** in Details for “Heating” and select **Temperature**. This allows for the heat transfer coefficient to be defined as a function of temperature
  2. Expand **Coefficient Type** in Details for “Heating” and select **Surface Temperature**. This means that the temperature the HTC is a function of is the surface temperature of the part
  3. The HTC vs. Temperature values can be entered into the table in **Tabular Data** from table given in the following slide
- The following page contains tables for all the necessary values for each Convection definition

Details of "Heating"

<b>Scope</b>	
Scoping Method	Named Selection
Named Selection	Outer_Surf(Sliced Ring Model)
<b>Definition</b>	
Type	Convection
Film Coefficient	Tabular Data
<input type="checkbox"/> Ambient Temperature	900. °C (step applied)
Convection Matrix	Program Controlled
Suppressed	No
Edit Data For	Film Coefficient
<b>Tabular Data</b>	
Independent Variable	Time
	Time
	X
	Y
	Z
	Temperature

Details of "Heating"

<b>Scope</b>	
Scoping Method	Named Selection
Named Selection	Outer_Surf(Sliced Ring Model)
<b>Definition</b>	
Type	Convection
Film Coefficient	Tabular Data
Coefficient Type	Average Film Temperature
<input type="checkbox"/> Ambient Temperature	Bulk Temperature
	Surface Temperature
Convection Matrix	Average Film Temperature
Suppressed	Difference of Surface and Bulk Temp
Edit Data For	Film Coefficient
<b>Tabular Data</b>	
Independent Variable	Temperature
<b>Graph Controls</b>	
X-Axis	Temperature

Tabular Data

	Temperature [°C]	<input checked="" type="checkbox"/> Convection Coefficient [W/mm <sup>2</sup> ·°C]
1	20.	5.e-005
2	1000.	1.2e-004
*		



# Step 10c: Define Thermal Boundary Conditions

Convection Name	Active Step #	Ambient Temp
Heating	1	900
Transfer	2	400
Quench	3	65
Air Cool	4	20
Deep Freeze	5	-30
Temper	6	150
Cool Down	7	20

Heating HTC vs. Temp	
Temperature	HTC
20	5.00E-05
1000	1.20E-04

Transfer HTC vs. Temp	
Temperature	HTC
20	1.00E-04
1000	2.00E-04

Air Cool and Air Cool 2 HTC vs. Temp	
Temperature	HTC
20	1.00E-04
1000	2.00E-04

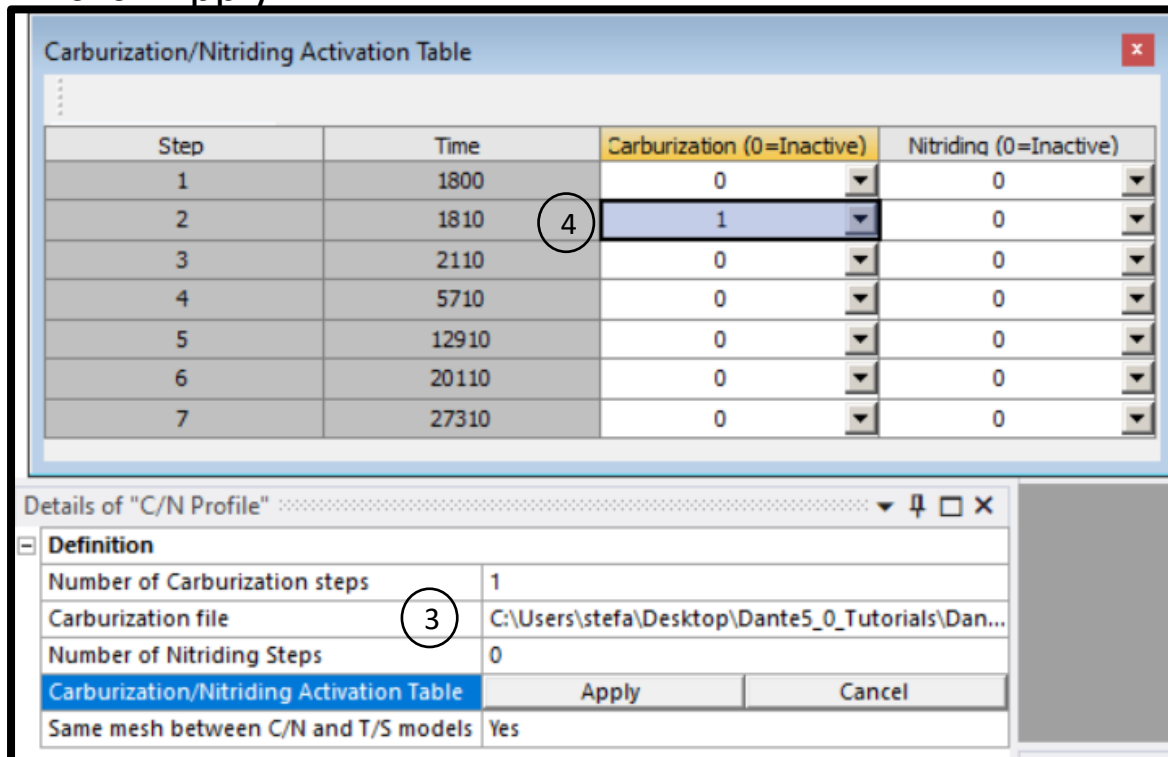
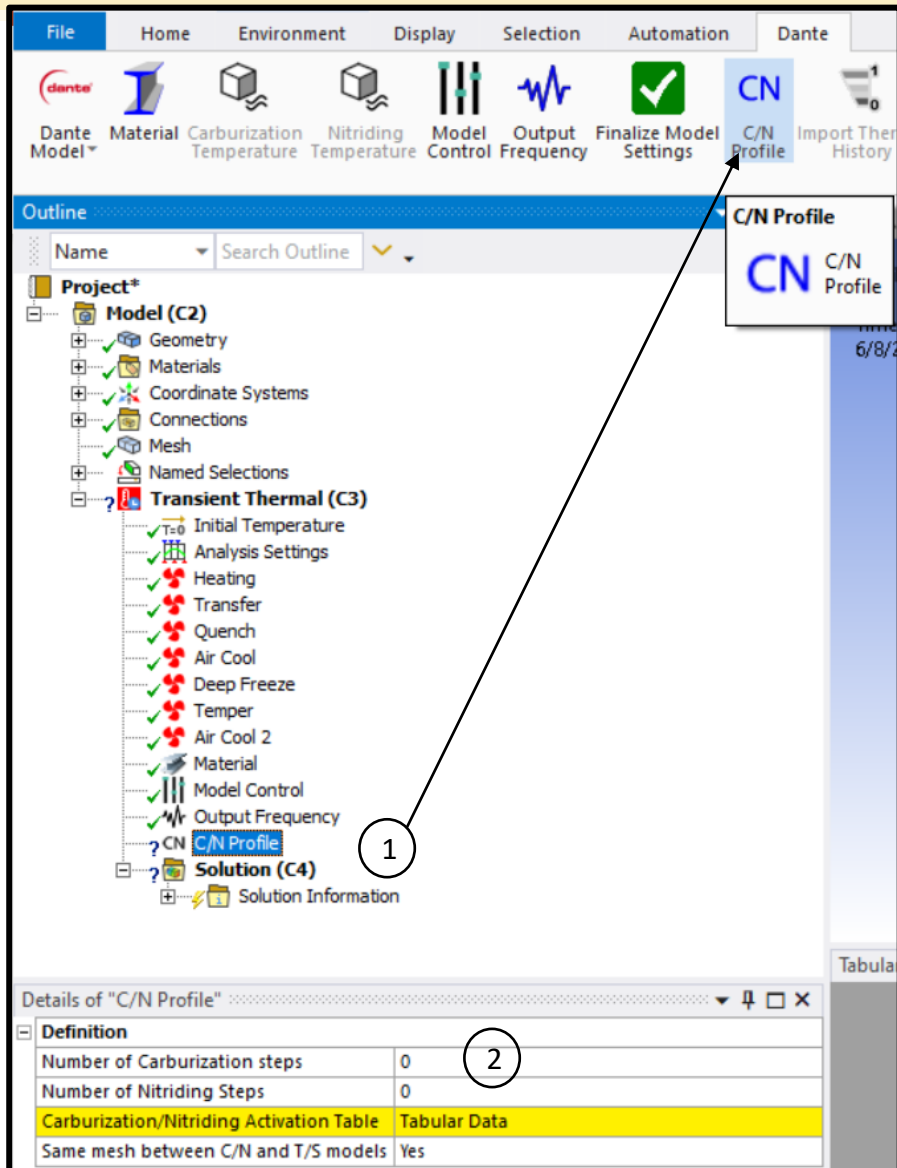
For Deep Freeze and Temper, the Film Coefficient will be a constant 5.00E-05

Quench HTC vs. Temp	
Temperature	HTC
20	1.00E-04
150	5.00E-04
300	1.50E-03
400	2.00E-03
450	3.75E-03
500	5.00E-03
550	5.00E-03
600	4.75E-03
650	3.00E-03
700	2.00E-03
750	1.50E-03
800	1.30E-03
1000	1.30E-03

# Step 11: Adding the Carbon Profile

1. Select **C/N Profile** to add the carbon profile that was saved in the carburization model
2. Up to 2 carbon profiles may be specified, set **Number of Carburization Steps** to 1
3. In the Carburization file, navigate to the project directory's dante files directory and select the carbon profile, here saved as sliced\_ring\_cc.cbn, click Apply

4. Set the carbon profile to be ramped into the model at step 2 by selecting '1' under the Carburization column in the **Carburization/Nitriding Activation Table**

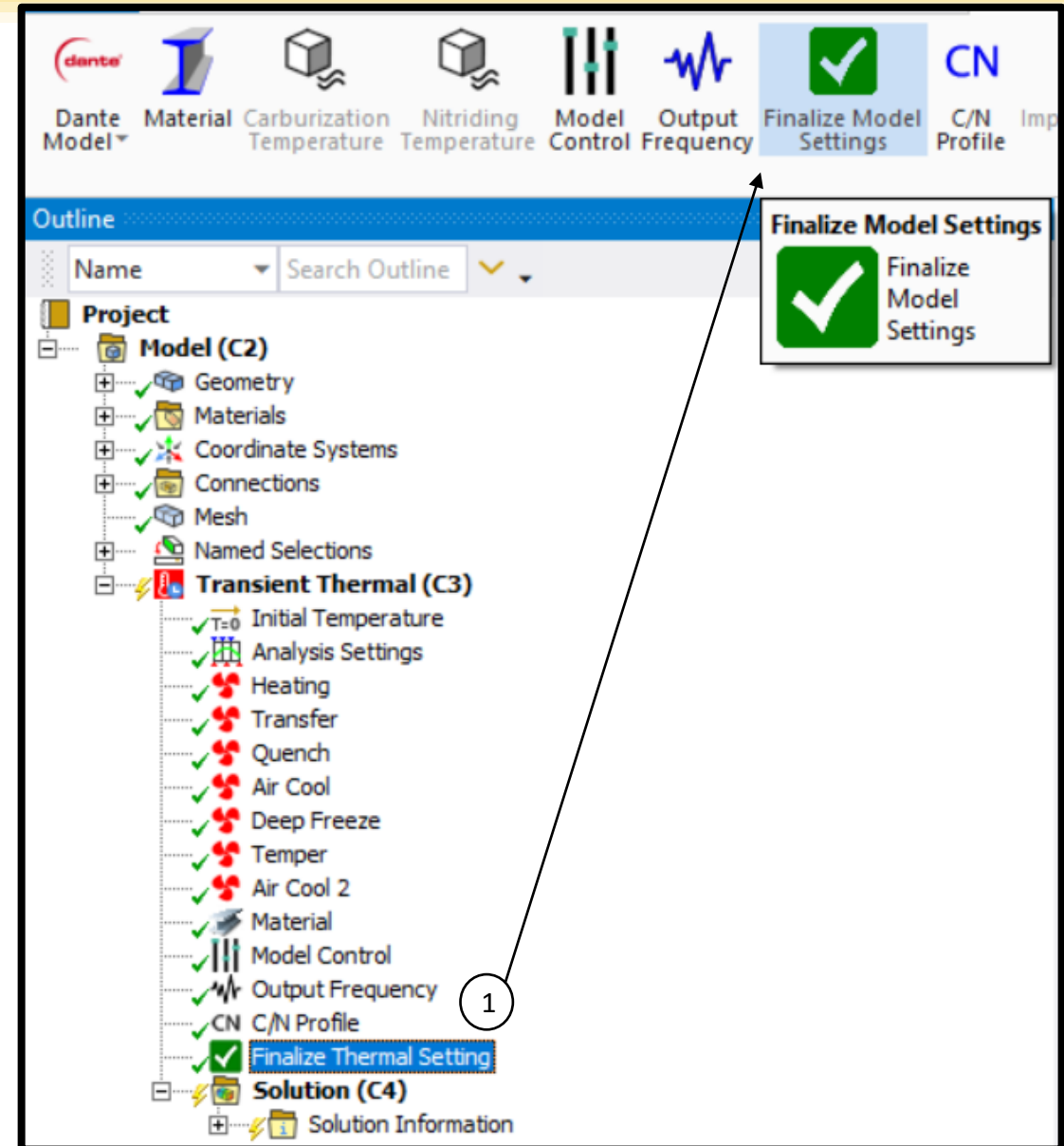
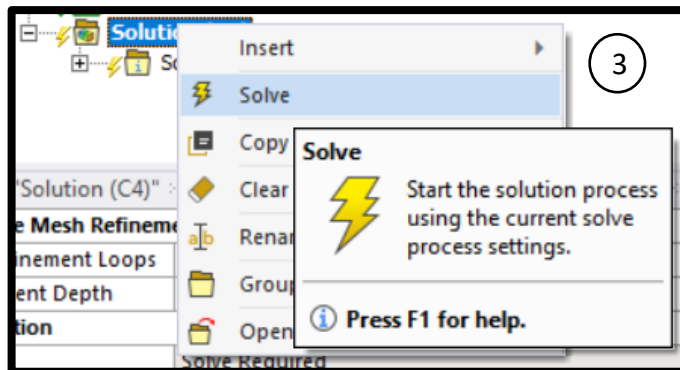


# Step 12: Running the Thermal Model

1. Select **Finalize Model Settings** from the Dante toolbar to add it to Transient Thermal in the Project Tree
2. It should be **green check** on Transient Thermal in the Project tree to make sure that what we did for Post-Processing is correct

**NOTE: Be sure to save the model at this point before Solving**

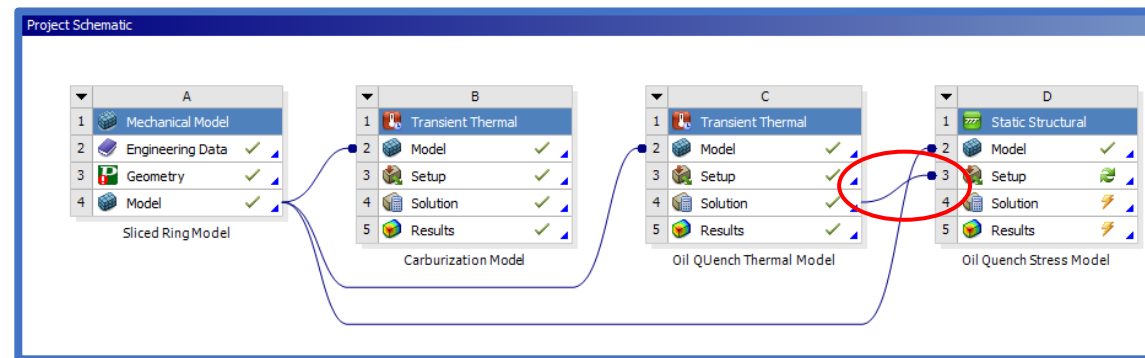
3. Right-click the Solution object and select **Solve** to run the model, when complete save the project and begin post-processing



# Step 13: Post-Processing Thermal History

There are 2 methods for supplying the thermal history of the component to the Stress Model:

1. Link the Thermal Model Solution to the Stress Model Setup in ANSYS Workbench (Shown below). This is part of the Stress Model Pre-Processing and is explained in the appropriate section. This is the preferred method
2. Use the DANTE ACT to generate a temperature history file that is then imported to the Stress Model (See next slide for this method)



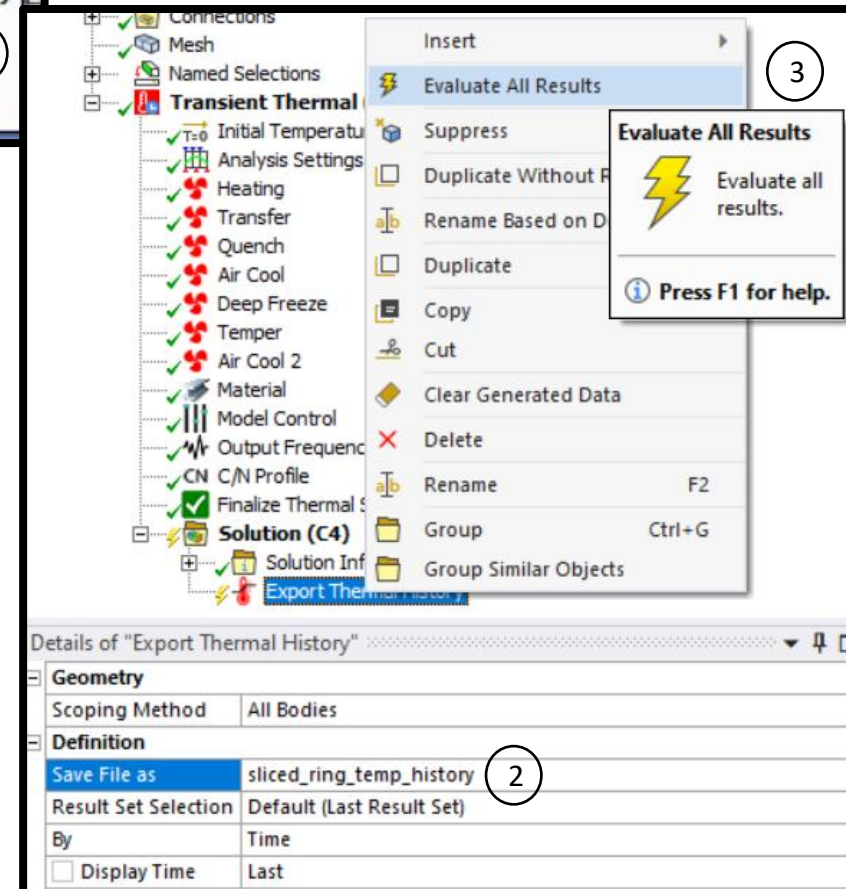
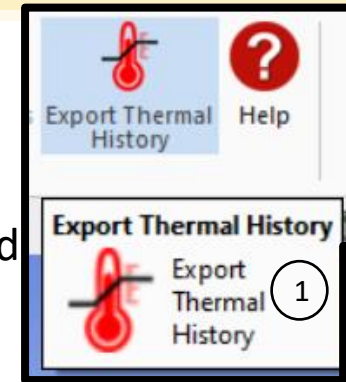


# Step 13b: Thermal Model Post-Processing, Thermal History File

**NOTE:** If the Thermal Model Solution is to be linked to the Stress Model Setup in Workbench during the Stress Model Pre-processing steps, this step should be skipped

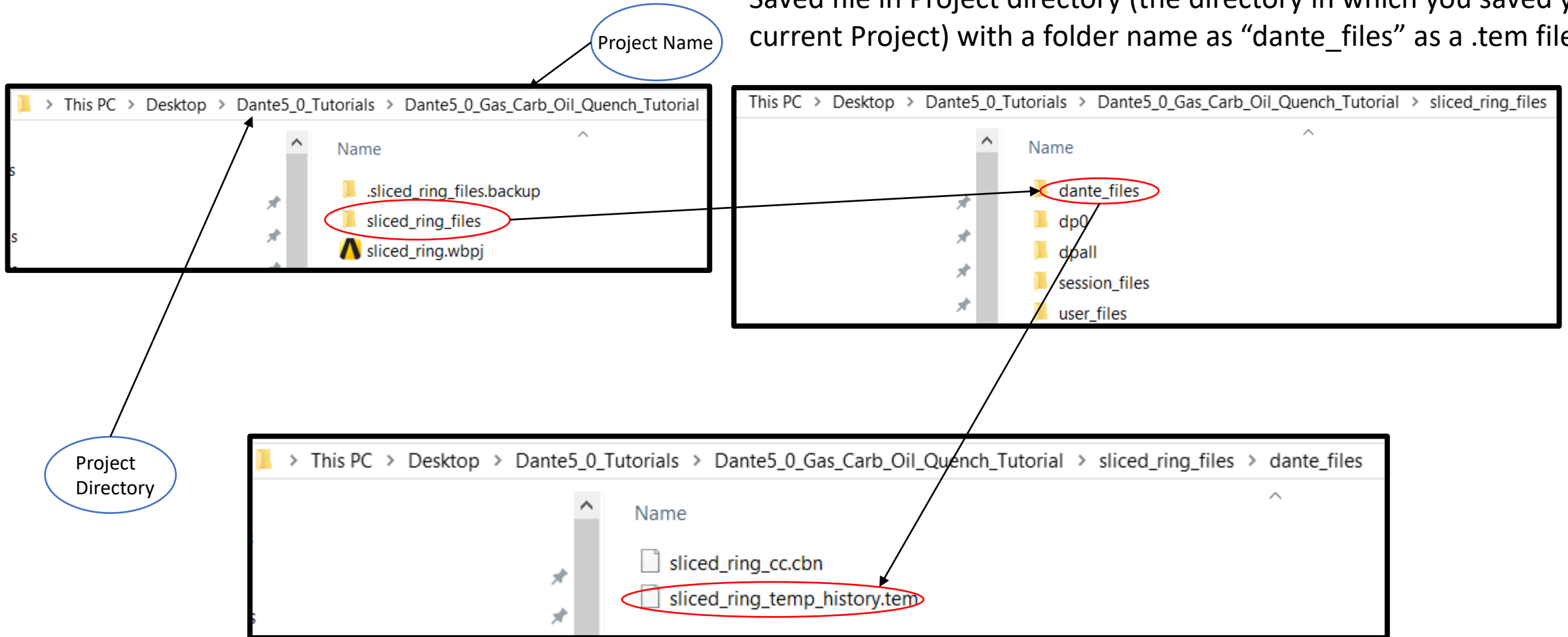
1. Click the **Export Thermal History** button in the Dante toolbar, and the Export Thermal History is added to the Solution
2. Under the Details of "Export Thermal History", change the file name to "sliced\_ring\_temp\_history" for the Save File As
3. Right click **Export Thermal History**, and select "**Evaluate All Results**" to write the thermal history results to the file. This writes out the temperature at each node, at each saved substep. The file is then saved in Project directory (the directory in which you saved your current Project.) with a folder name as "dante\_files" as a .tem file.(See next Slide)

**NOTE:** We will need to find this file later, so knowing what directory it is in is important. For larger models, writing out the thermal history file may take several hours



## Step 13c: Thermal History File Directory

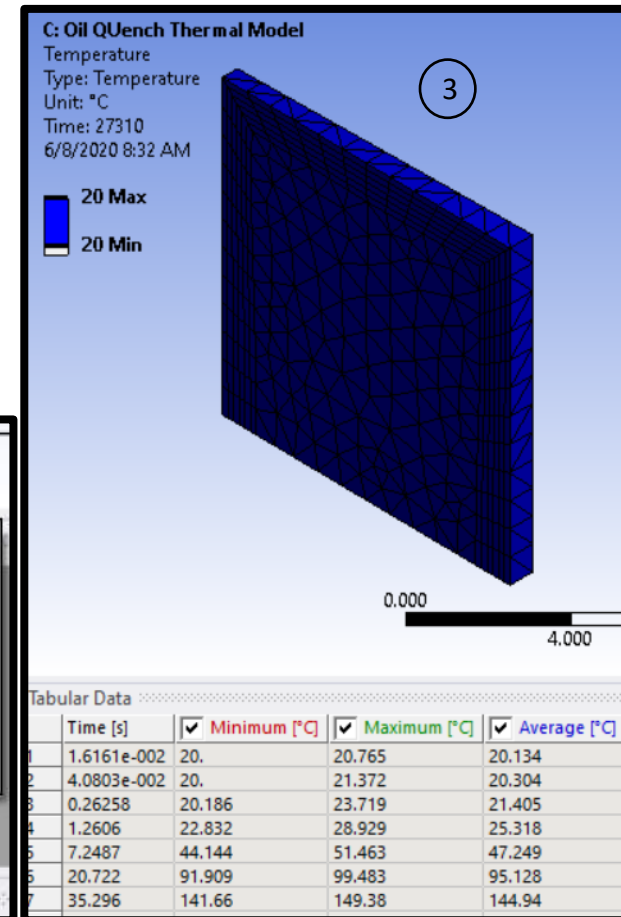
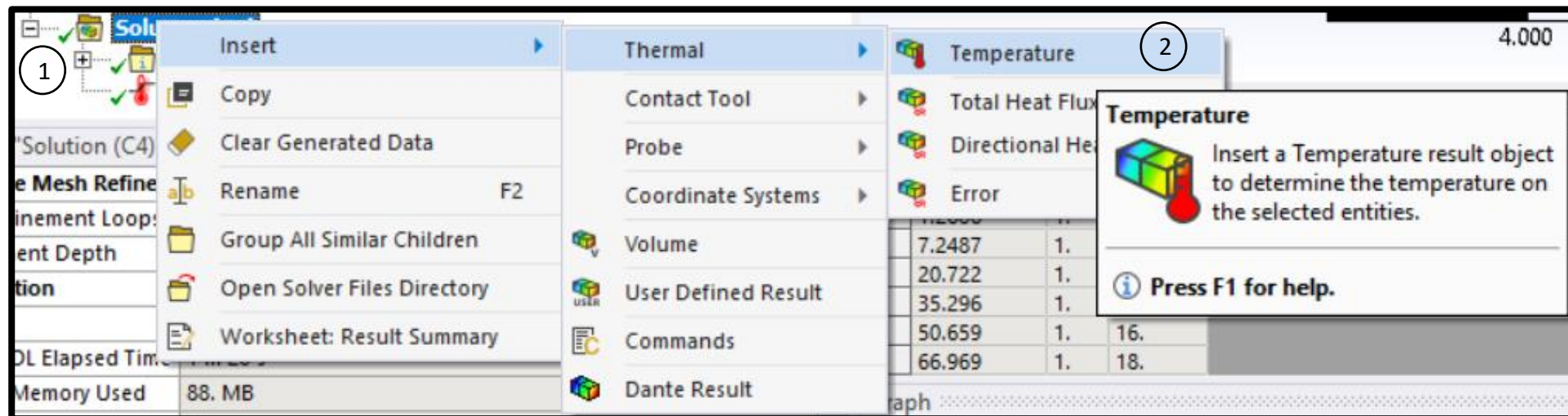
Saved file in Project directory (the directory in which you saved your current Project) with a folder name as “dante\_files” as a .tem file





# Step 14: Thermal Model Post-Processing Temperature

1. Expand the Thermal tab in the Solution toolbar and select **Temperature**
2. Right click on Temperature under Solution in the Project Tree and select **Evaluate All Results**
3. The temperature contour at the last substep (end of the simulation) is shown. A table is also shown in Tabular Data with the time, minimum temperature, and maximum temperature at every saved substep



# Step 14b: Thermal Model Post-Processing, Temperature

Utilizing the Tabular Data, it is possible to examine the temperature profile at any saved substep throughout the entire analysis:

1. Click on the substep number to be examined (in this case substep 20)
2. Right click in the blue highlighted area and select Retrieve This Result
3. The temperature contour at this time is then displayed

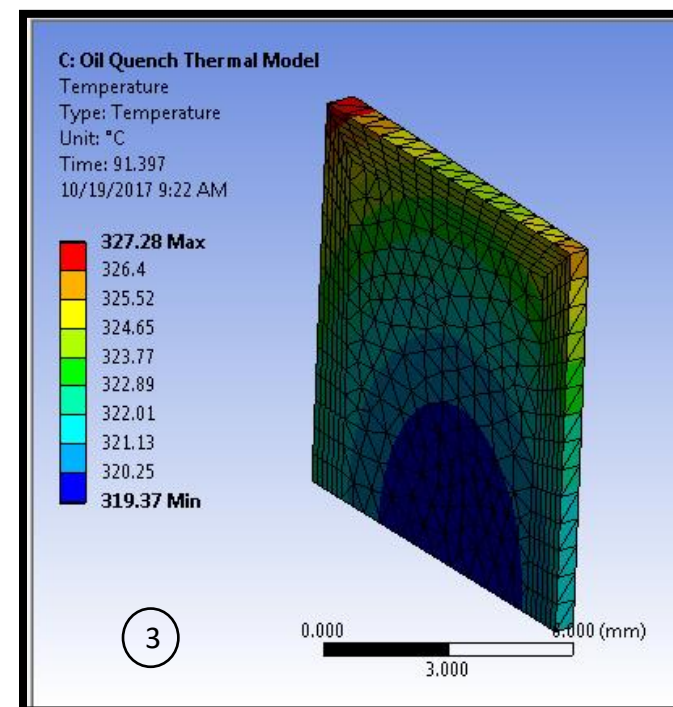
**Note: Results may differ from those shown due to step size calculation differences.**

1

	Time [s]	✓ Minimum [°C]	✓ Maximum [°C]
16	57.4	217.83	225.79
17	65.32	242.66	250.63
18	73.789	268.43	276.4
19	82.259	293.37	301.32
20	91.397	319.37	327.28
21	100.54	344.42	352.27
22	110.5	370.67	378.44
23	120.46	395.82	403.49

2

	Time [s]	✓ Minimum [°C]	✓ Maximum [°C]
16	57.4	217.83	225.79
17	65.32	242.66	250.63
18	73.789	268.43	276.4
19	82.259	293.37	301.32
20	91.397	319.37	327.28
			352.27
			378.44
			403.49
			429.89
			455.
			481.67
			506.83



## Step 14c: Thermal Model Post-Processing, Temperature

Hint: At this stage it is a good idea to check that the temperatures at the end of each step makes sense; i.e., ~900 °C at 1800 s , ~65 °C at 2110 s , ~20 °C at 5710 s, ~-30 °C at 12910 s, ~150 °C at 20110 s and ~20 °C at 27310 s.

If the temperatures are not approximately the ambient temperature set for that step, the most likely cause is in the HTC definition.

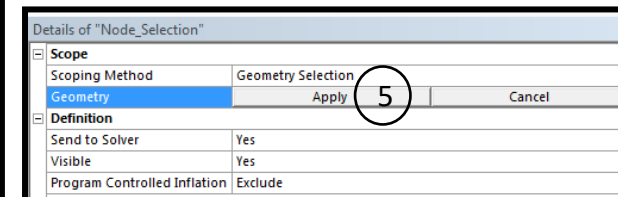
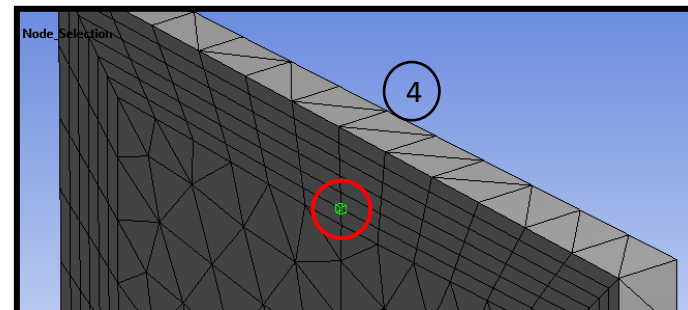
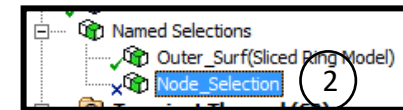
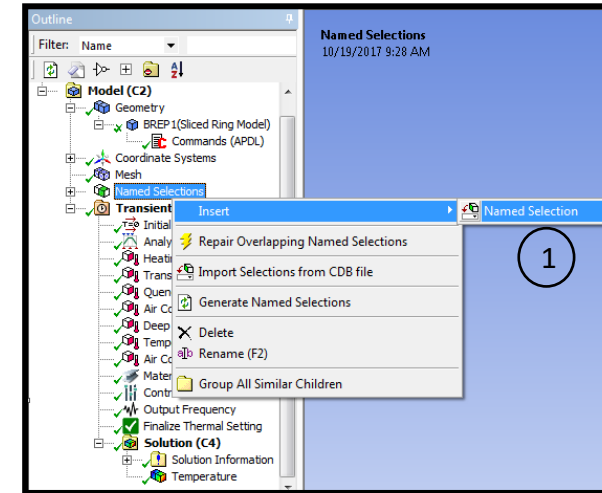
**The Deactivation of steps not using a particular HTC is critical. If this is neglected, Ansys will use an average value for the HTC and the ambient temperature.**

For example: the temperature at the end of heating (1800 s) should be 900 °C, but it is only 346 °C. That means no steps were deactivated and the ambient temperature used by Ansys was  $(900+400+65+20-30+150+20) / 7 = 217.8$

# Step 15: Temperature History Plot

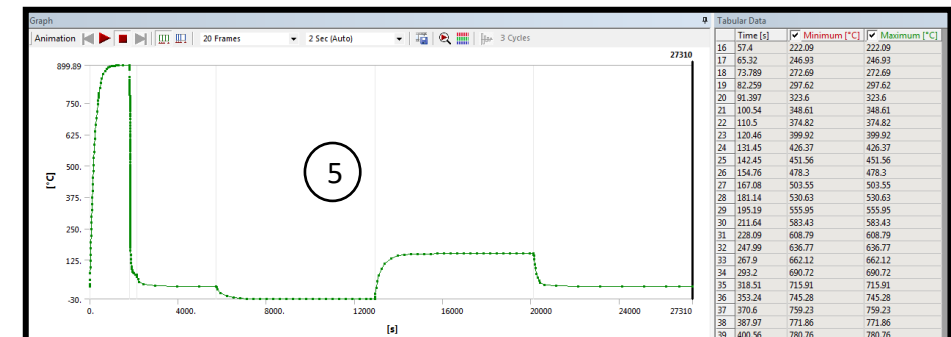
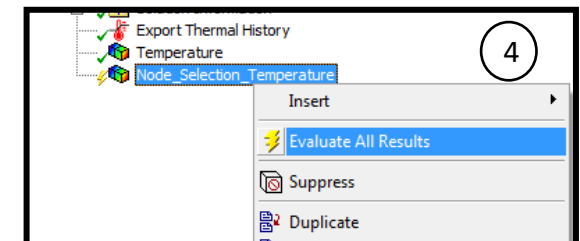
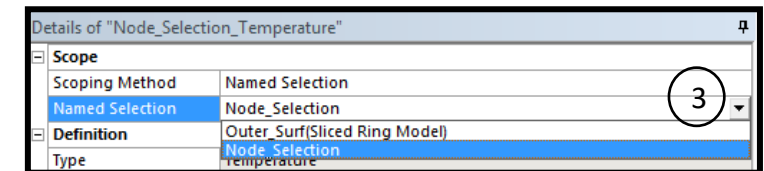
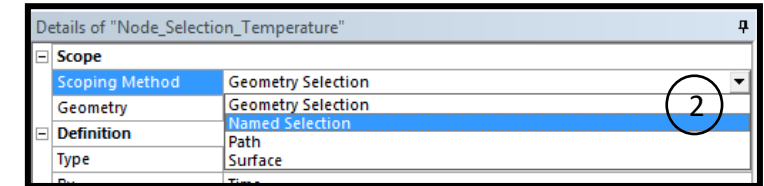
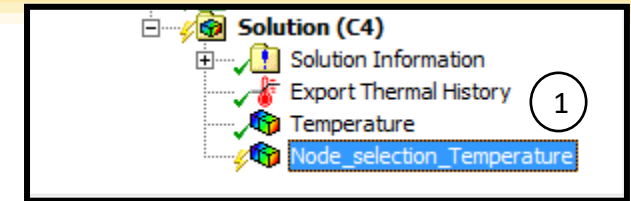
It is also possible to examine temperature versus time at a particular node. First, however, a **Named Selection** for the node must be created.

1. Right click on **Named Selections** in the Project Tree, select **Insert**, and then select **Named Selection**
2. Rename the Named Selection for easy identification (Node\_Selection in this case)
3. Choose the **Node Selection Mode**
4. Select the desired node on the component
5. Select **Apply** for Geometry in the Details of "Node\_Selection"



# Step 15b: Temperature History

1. Add another **Temperature** from the Solution toolbar to the Solution and rename it for easy recognition later (Node Selection Temperature for this case)
2. In the Details of "Node Selection Temperature", expand the **Scoping Method** and select **Named Selection**
3. In the Details of "Node Selection Temperature", expand the **Named Selection** and select **Node Selection**
4. Right Click on **Node Selection Temperature** under Solution in the Project Tree and select **Evaluate All Results**
5. A line plot is generated in the Graph selection, along with all the data points in the Tabular Data section. Like the carbon profile, the Tabular Data can be exported to a text file or Excel file



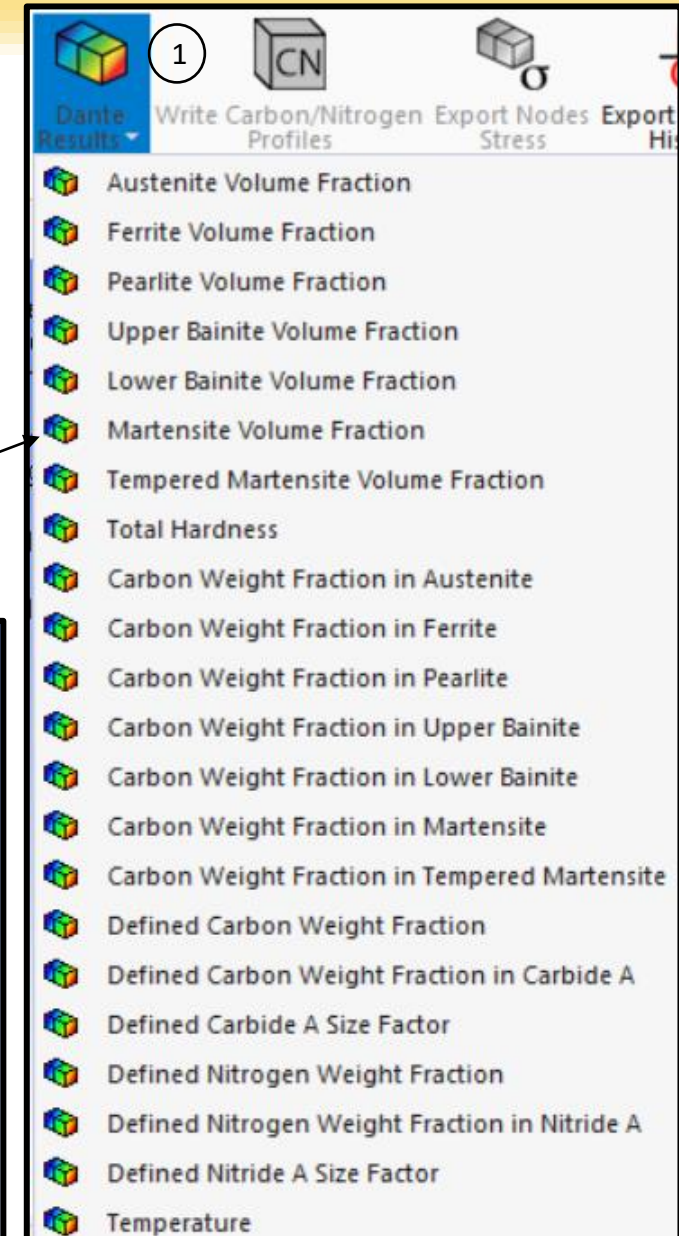
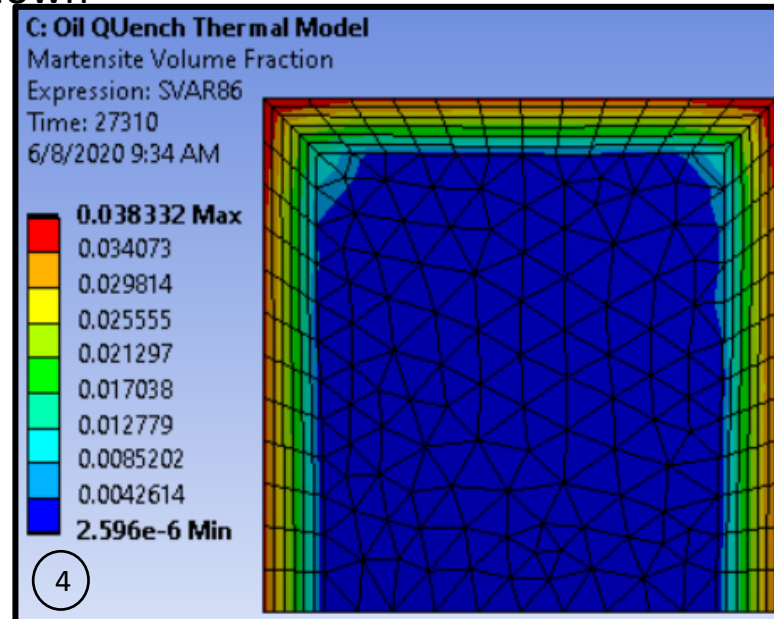




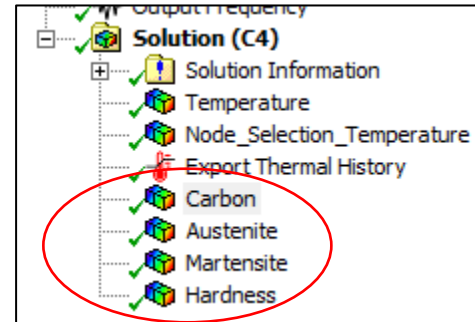
# Step 16: Thermal Model Post-Processing, Dante Results

1. Select **Dante Result** from the Dante toolbar to add a Dante Result to the Solution in the Project Tree
2. Available solution results in a thermal model ranges from volume fractions of microstructural phase constituents to hardness value and temperature
3. Select **Martensite Volume Fraction** as an example to create a Martensite Volume Fraction solution object in the solution tree, right click to evaluate
4. The Martensite contour plot is then shown

**NOTE:** Since a tempering stage was added with tempering kinetics, the amount of martensite at the end of simulation is low with around 4% volume content near the surface, evaluate a Tempered Martensite object to visualize the difference along with the other phase fractions.



## Step 16b: Thermal Model Post-Processing, Dante Results



Multiple Dante Results can be added to the Solution in the Project Tree and then renamed to make examining the Dante Results less time consuming

**That concludes the Thermal Model**

**Save the Project and close Ansys Mechanical**

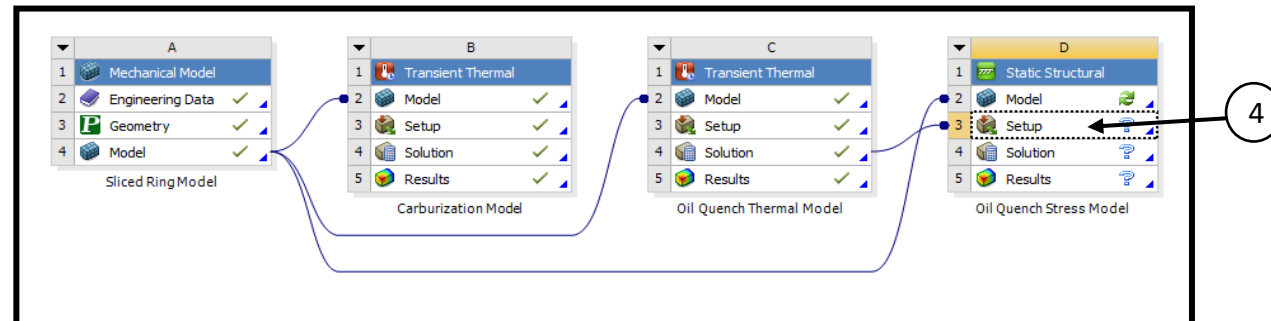
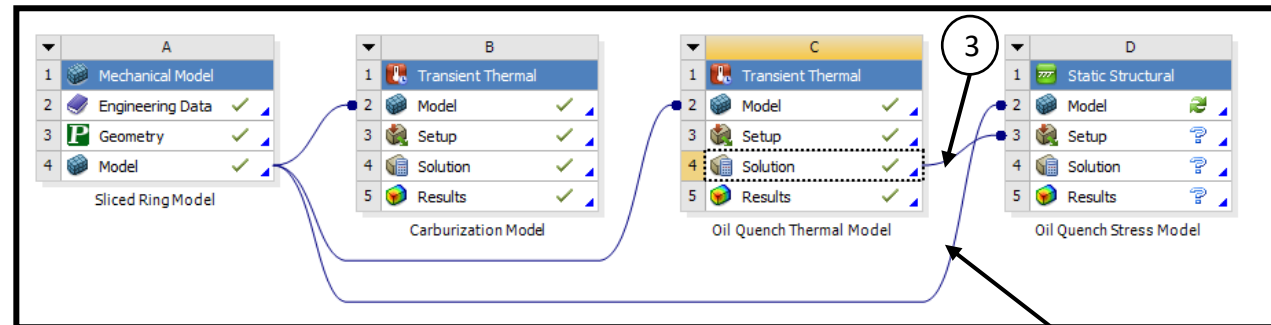
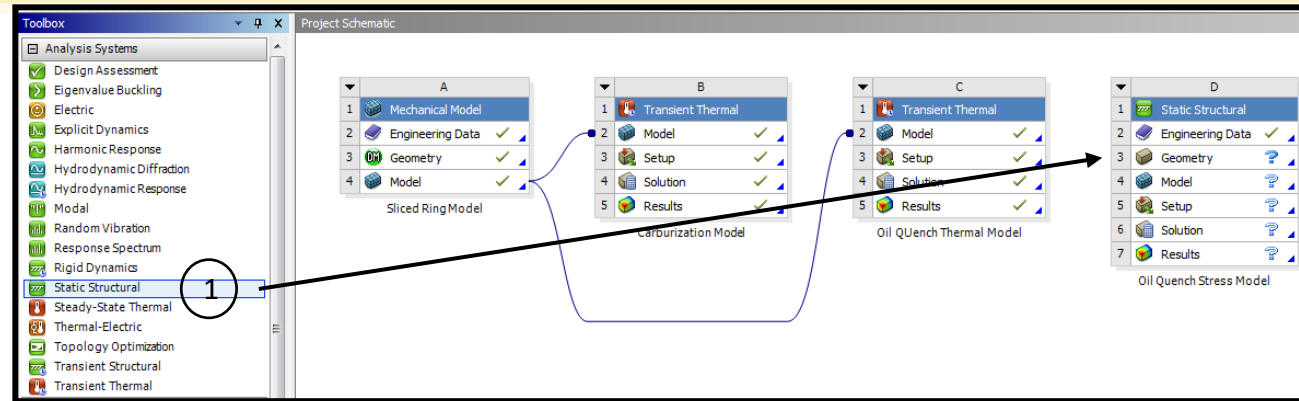
**The Stress Model can now be built**

# Quench Hardening Stress Model

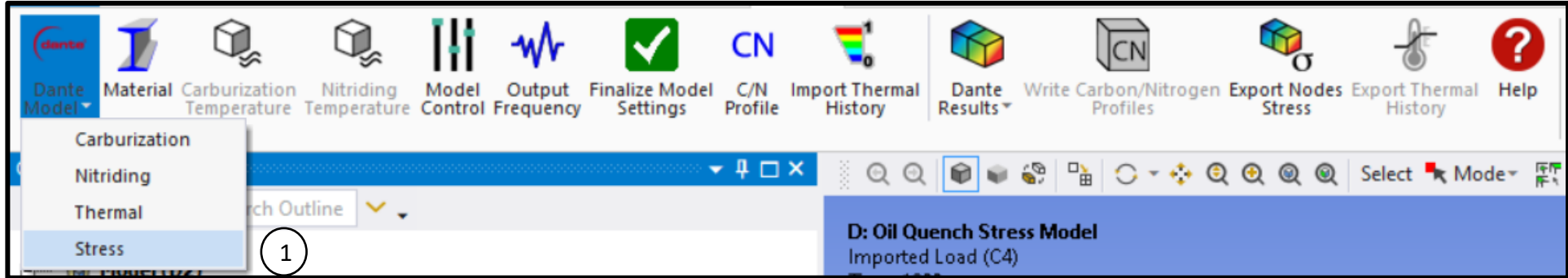


# Step 1: Stress Model Setup, Add Analysis System to Project Schematic

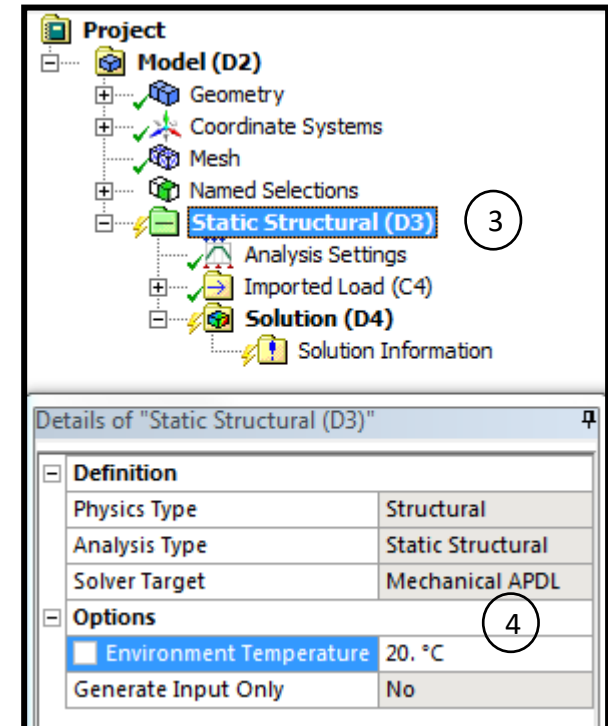
1. Drag and drop a Static Structural from Analysis Systems into the Project Schematic and Rename it as "Oil Quench Stress Model"
2. Drag and drop the Model from Sliced ring model to the model of Oil Quench Stress Model
3. Drag and drop the Solution from Oil Quench Thermal Model to the Setup of Oil Quench Stress Model
4. Double click Setup in the Oil Quench Stress Model to open Ansys Mechanical



## Step 2: Define DANTE Model & Initial Temperature



1. Expand **Dante Model** in the Dante toolbar and select **Stress**
2. The highlighted buttons are used to set up and post-process the Dante stress model
3. Click on Static Structural in the Project Tree
4. In the Details of "Static Structural", change **Environment Temperature** to 20 °C

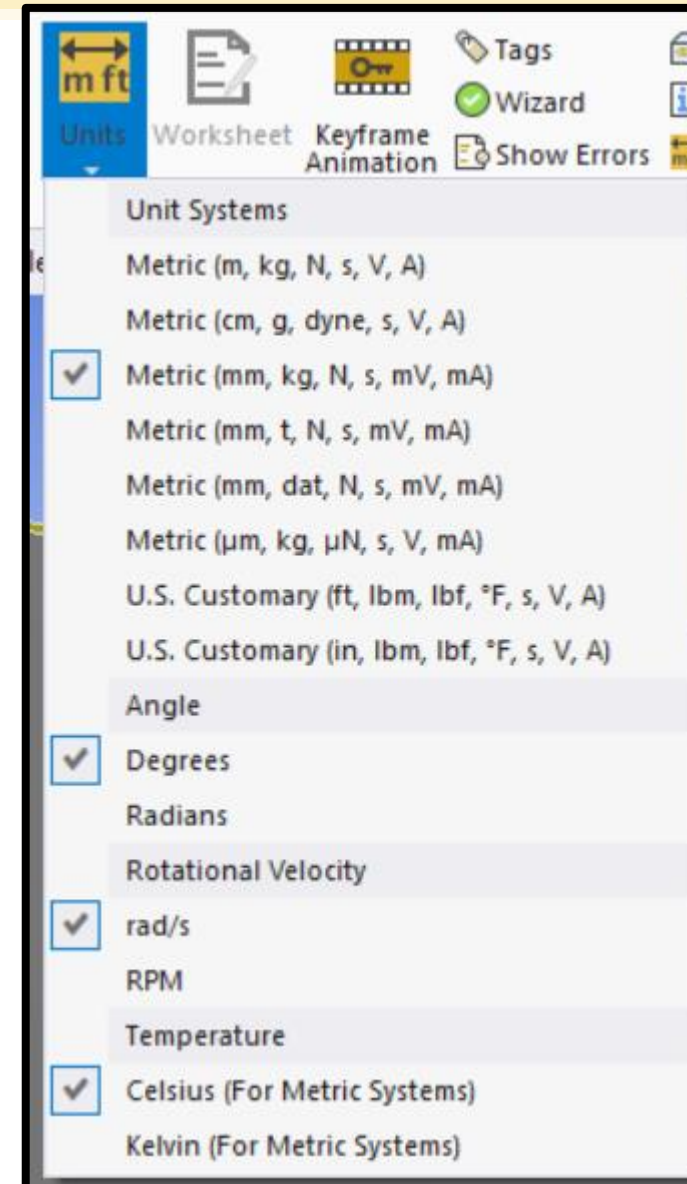


## Step 3: Define Units

It is critical that the units be properly defined.

1. Click on Units in the Home toolbar
2. Select Metric (mm, kg, N, s, mV, mA)
3. Select Degrees
4. Select rad/s (This isn't critical as there is no motion defined in the heat treatment models)
5. Select Celsius (For Metric Systems)

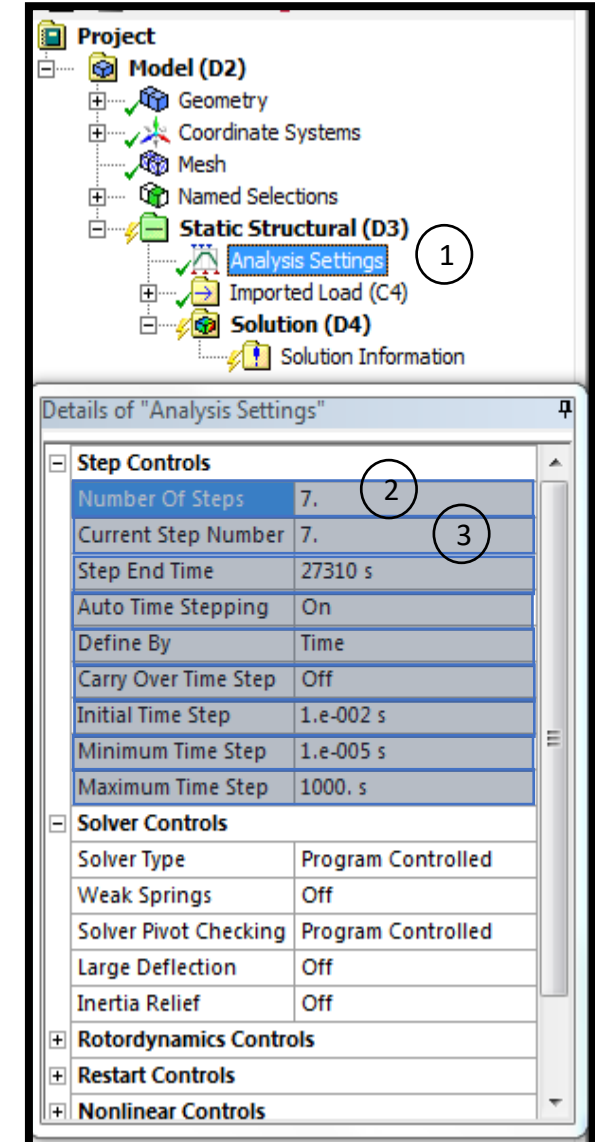
**NOTE:** It is absolutely critical that these units are selected. If different units are selected, the model may run, but the results will be WRONG.



# Step 4: Define Processing Steps

1. Click on **Analysis Settings** in the Project Tree
2. In the Details of "Analysis Settings", change the **Number of Steps** to 7
3. Beginning with Current Step Number 7, enter the information from the table into the Step Controls in Details of "Analysis Steps". It is necessary to work backwards with the step numbers because Ansys populates the Step End Time in 1 second increments for each step. Ansys will then reject the entry for Step End Time if it does not progress chronologically. **The Number of Steps and Step End Time for each step must be the same as the Thermal Model.**

Current Step Number	Step End Time	Auto Time Stepping	Define By	Carry Over Time Step	Initial Time Step	Minimum Time Step	Maximum Time Step
7	27310	On	Time	Off	1.00E-02	1.00E-05	1000
6	20110	On	Time	Off	1.00E-02	1.00E-05	300
5	12910	On	Time	Off	1.00E-02	1.00E-05	300
4	5710	On	Time	Off	1.00E-02	1.00E-05	1000
3	2110	On	Time	Off	1.00E-02	1.00E-05	100
2	1810	On	Time	Off	1.00E-02	1.00E-05	10
1	1800	On	Time	N/A	1.00E-02	1.00E-05	1000

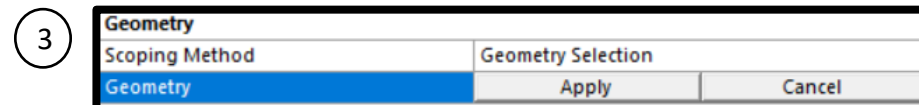
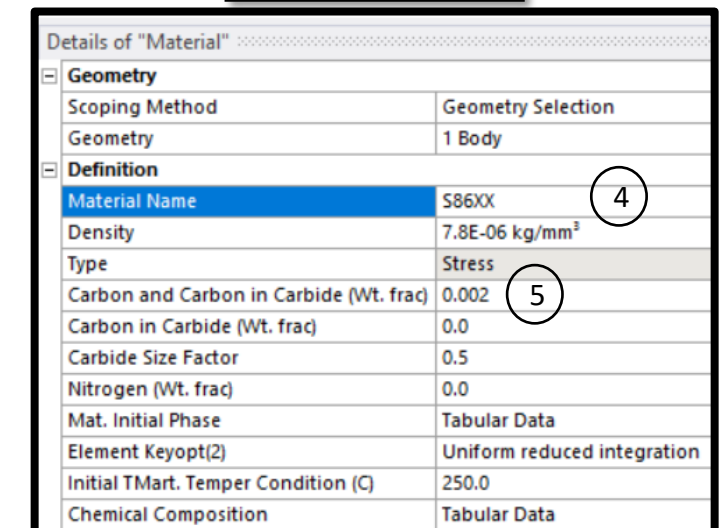
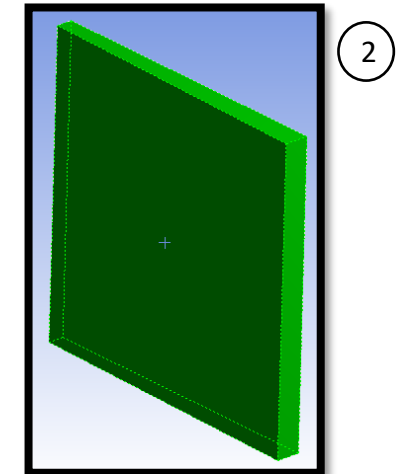
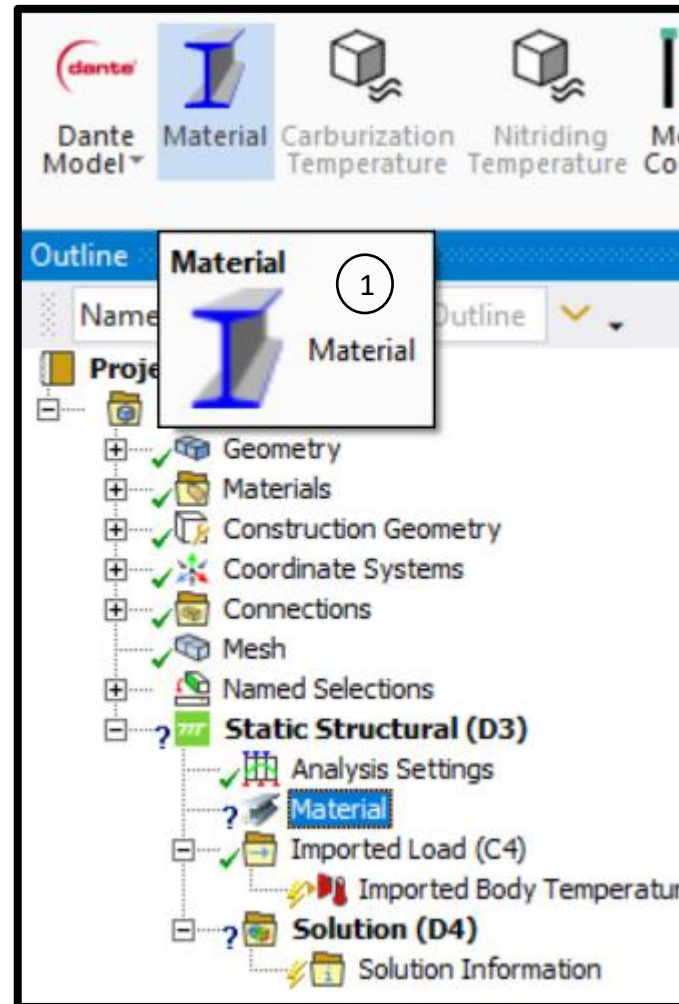


# Step 5: Assign Material

1. Select **Material** from the Dante toolbar to add it to Static Structural in the Project Tree

The following Steps apply to modifying values in the Details of "Material".

2. Clicking the yellow Geometry box with the **Scoping Method** set to **Geometry Selection**, simply click on the part to select the entire body
3. Select **Apply** for Geometry
4. Change **Material Name** to S86XX for the AISI 8600 series steel
5. Change the **Carbon and Carbon in Carbide (Wt. frac)** to 0.002 to indicate AISI 8620



# Step 6: Define Initial Phase

The following Steps apply to modifying values in the Details of “Material”.

1. The **Type** should be set to **Stress**. This tells the user subroutine which mathematical models to use
2. Click **Tabular Data** for **Mat. Initial Phase** to enter the initial phases of the material
3. Click **Apply** when finished (Default values of 30% Ferrite & 70% Pearlite are fine for this exercise)

Details of "Material"

[-] Geometry	
Scoping Method	Geometry Selection
Geometry	1 Body
[-] Definition	
Material Name	S86XX
Density	7.8E-06 kg/mm <sup>3</sup>
Type	Stress
Carbon and Carbon in Carbide (Wt. frac)	0.002
Carbon in Carbide (Wt. frac)	0.0
Carbide Size Factor	0.5
Nitrogen (Wt. frac)	0.0
Mat. Initial Phase	Tabular Data
Element Keyopt(2)	Uniform reduced integration
Initial TMart. Temper Condition (C)	250.0
Chemical Composition	Tabular Data

Carbon and Carbon in Carbide (Wt. frac)	0.002
Carbon in Carbide (Wt. frac)	0.0
Carbide Size Factor	0.5
Nitrogen (Wt. frac)	0.0
Mat. Initial Phase	Apply Cancel
Initial TMart. Temper Condition (C)	250.0
Chemical Composition	Tabular Data

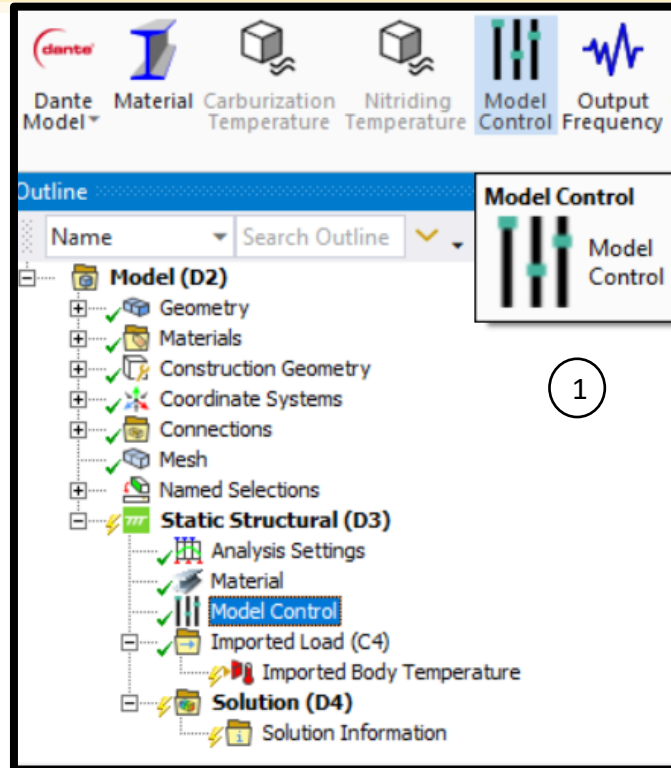
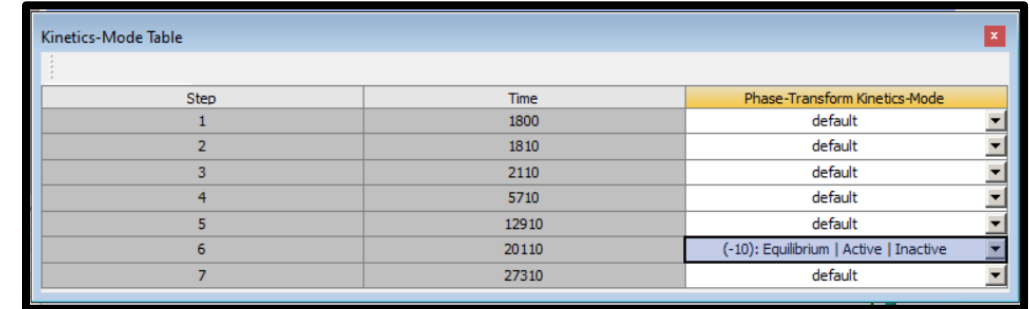
  

Mat. Initial Phase					
Ferrite	Pearlite	Upper Bainite	Lower Bainite	Martensite	Tempered Martensite
0.3	0.7	0.0	0.0	0.0	0.0



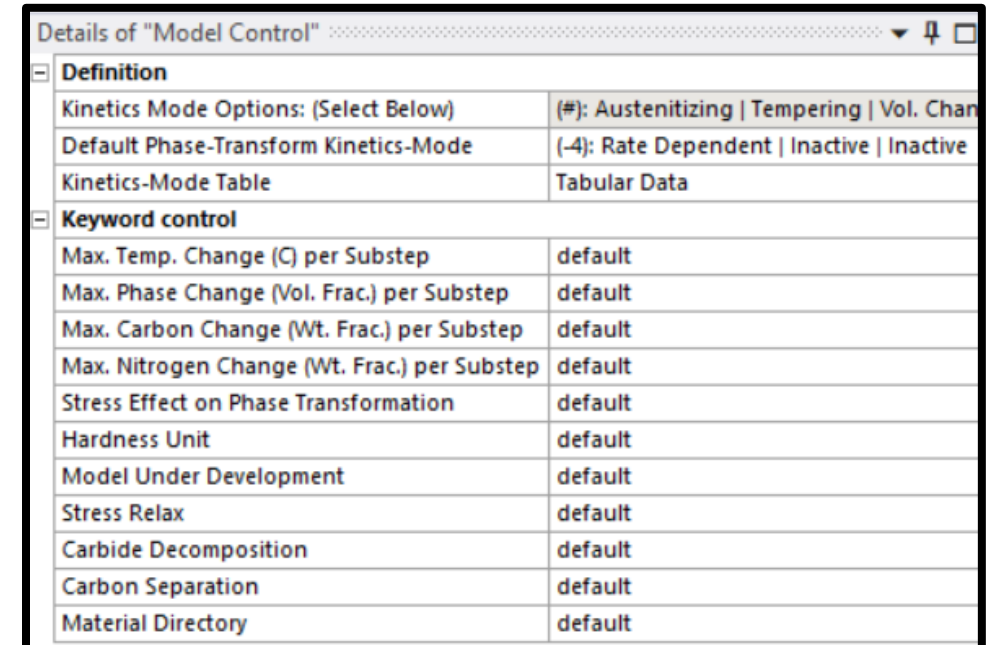
# Step 7: Define Control File

1. In the Dante toolbar, select the **Model Control** object to insert it into the Model Tree
2. Click on **Tabular Data** next to the **Kinetics-Mode Table** (which should be yellow) to review it and set the tempering kinetics (-10) at step 6 and click Apply to finish

The screenshot shows the 'Kinetics-Mode Table' dialog box. It contains a table with columns for Step, Time, and Phase-Transform Kinetics-Mode. The table has 7 rows. The 'Phase-Transform Kinetics-Mode' column has a dropdown menu for each row. The dropdown for step 6 is currently set to '(-10): Equilibrium | Active | Inactive'.

Step	Time	Phase-Transform Kinetics-Mode
1	1800	default
2	1810	default
3	2110	default
4	5710	default
5	12910	default
6	20110	(-10): Equilibrium   Active   Inactive
7	27310	default

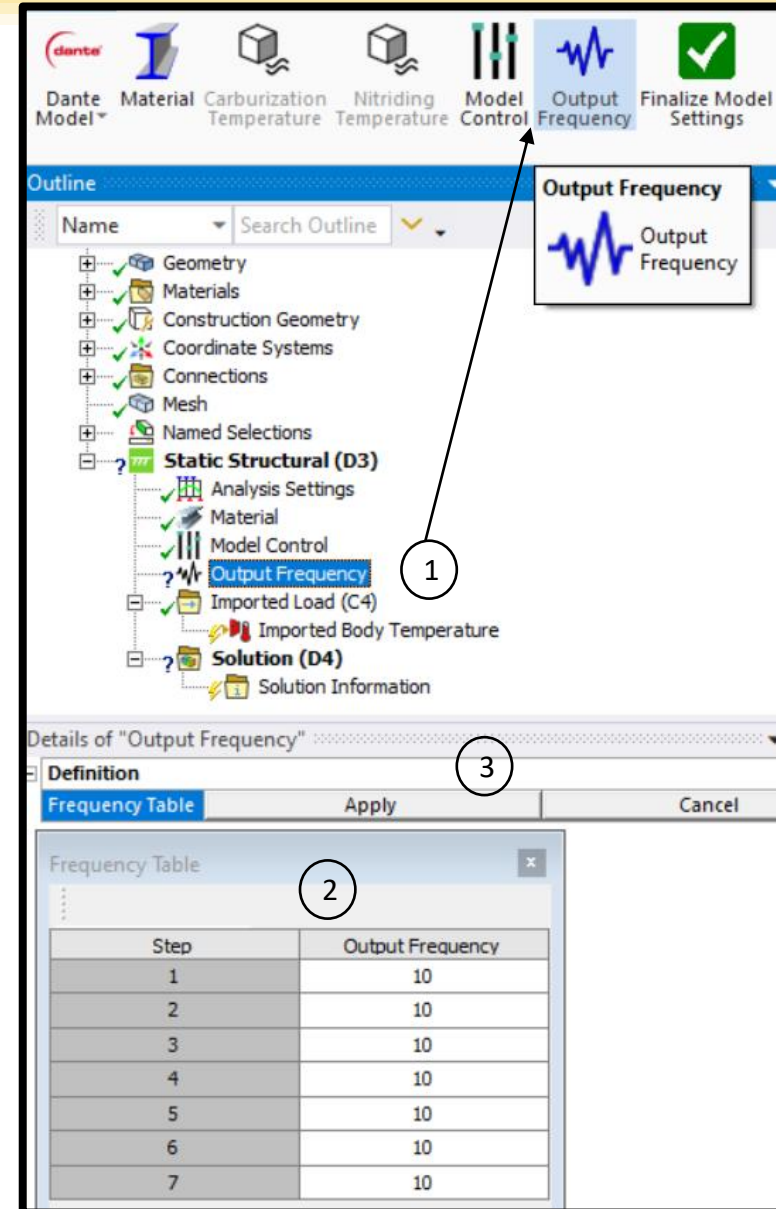


The screenshot shows the 'Details of "Model Control"' dialog box. It contains two sections: 'Definition' and 'Keyword control'. The 'Definition' section has three rows with dropdown menus. The 'Keyword control' section has a list of keywords with their corresponding default values.

Definition	
Kinetics Mode Options: (Select Below)	(#): Austenitizing   Tempering   Vol. Chan
Default Phase-Transform Kinetics-Mode	(-4): Rate Dependent   Inactive   Inactive
Kinetics-Mode Table	Tabular Data
Keyword control	
Max. Temp. Change (C) per Substep	default
Max. Phase Change (Vol. Frac.) per Substep	default
Max. Carbon Change (Wt. Frac.) per Substep	default
Max. Nitrogen Change (Wt. Frac.) per Substep	default
Stress Effect on Phase Transformation	default
Hardness Unit	default
Model Under Development	default
Stress Relax	default
Carbide Decomposition	default
Carbon Separation	default
Material Directory	default

# Step 8: Define Output Frequency

1. Select **Output Frequency** from the Dante toolbar to add it to Transient Thermal in the Project Tree. The Output Frequency is used to define the frequency of writing simulation results to the results file
2. In Details of “Output Frequency”, review the contents of the Frequency Table entries by clicking on **Tabular Data** next to **Frequency Table**. For a stress model, the default frequency values should be set to 10
3. Click Apply when done



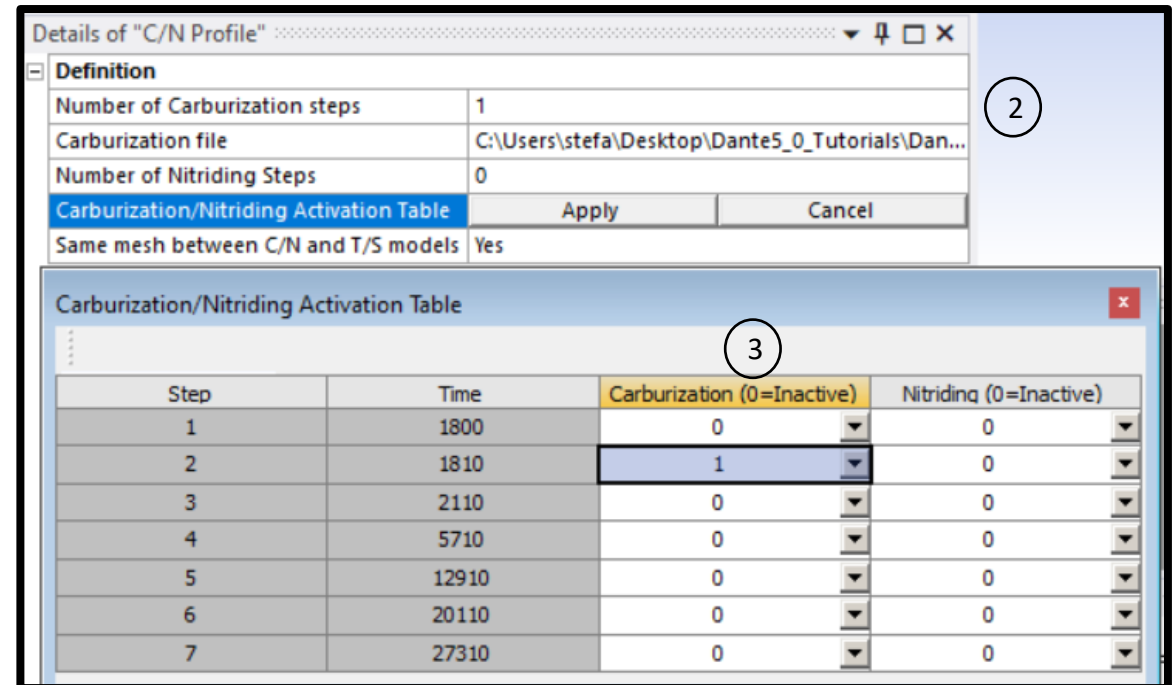
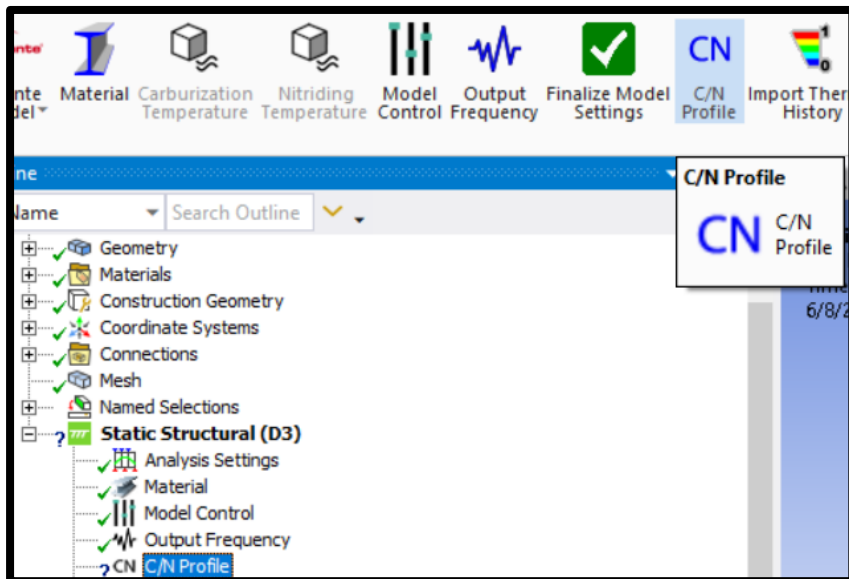
The screenshot shows the Dante software interface. At the top, the toolbar includes icons for Dante Model, Material, Carburization Temperature, Nitriding Temperature, Model Control, Output Frequency, and Finalize Model Settings. The 'Output Frequency' icon is highlighted with a blue box and a checkmark. Below the toolbar, the 'Outline' pane shows a tree view of the project structure. The 'Output Frequency' item is selected and highlighted with a blue box, with a circled '1' next to it. To the right of the 'Outline' pane, a 'Details of "Output Frequency"' dialog box is open. The 'Definition' tab is selected, and the 'Frequency Table' button is highlighted with a blue box, with a circled '2' next to it. The 'Apply' button is also highlighted with a blue box and a circled '3'. Below the dialog box, a 'Frequency Table' window is open, showing a table with 7 rows and 2 columns: 'Step' and 'Output Frequency'. The table contains the following data:

Step	Output Frequency
1	10
2	10
3	10
4	10
5	10
6	10
7	10



# Step 9: Map In The Carbon Profile

1. Click on the **C/N Profile** to insert it into the model tree
2. In the Details of "C/N Profile", set the **Number of Carburization steps** to 1 and for Carburization file, navigate to the sliced\_ring\_cc.cbn carbon profile in the dante\_files directory in the project directory
3. Then in the Carburization/Nitriding Activation Table, set the value under **Carburization** to "1" for step 2 to activate the carbon profile at that step

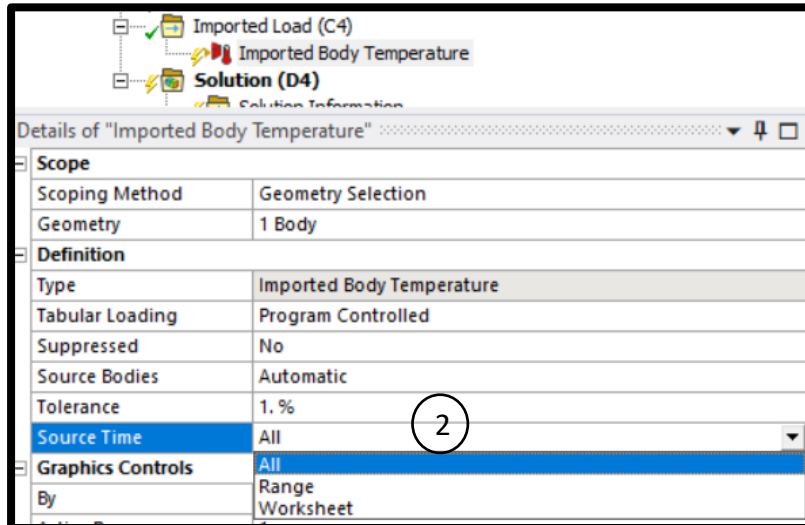


# Step 10: Import Thermal History

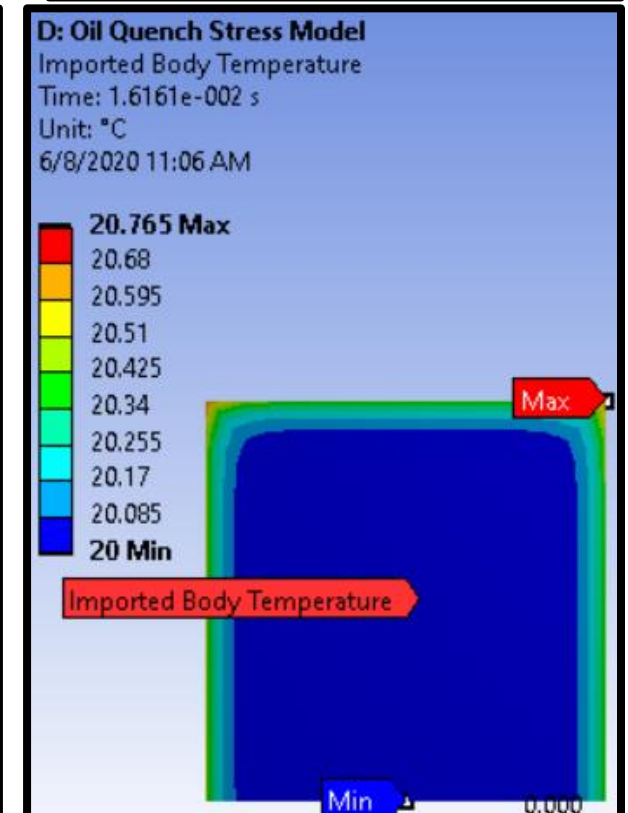
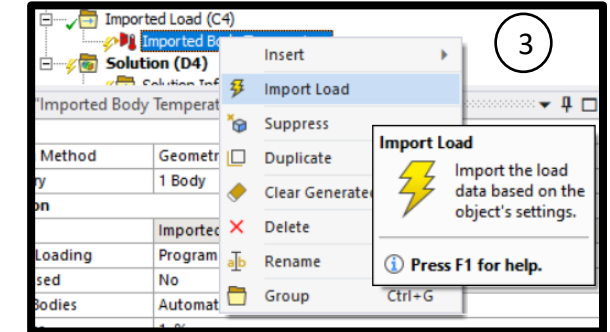
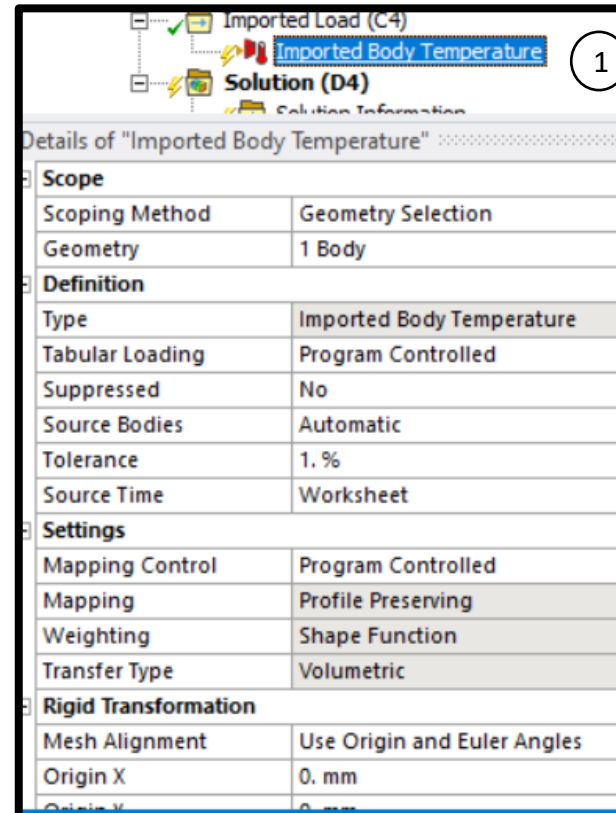
If the link between the Thermal model solution to the Stress model setup was made, the **Imported Load** object should have been added to the model tree automatically.

1. If this is the case, select the **Imported Body Temperature** under **Imported Load** and be sure that the Source Time is set to All instead of Worksheet, otherwise, the thermal history will not be mapped into the stress model
2. Right-click on the Import Body Temperature and select **Import Load** to load the temperature profile

**NOTE: This method of importing the thermal history is recommended**



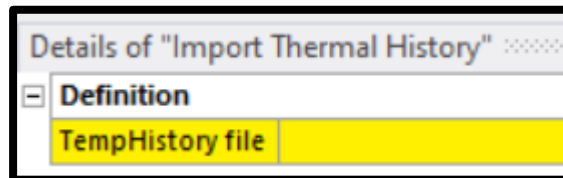
**NOTE: The Stress Model is “driven” by the thermal history predicted in the Thermal Model.**



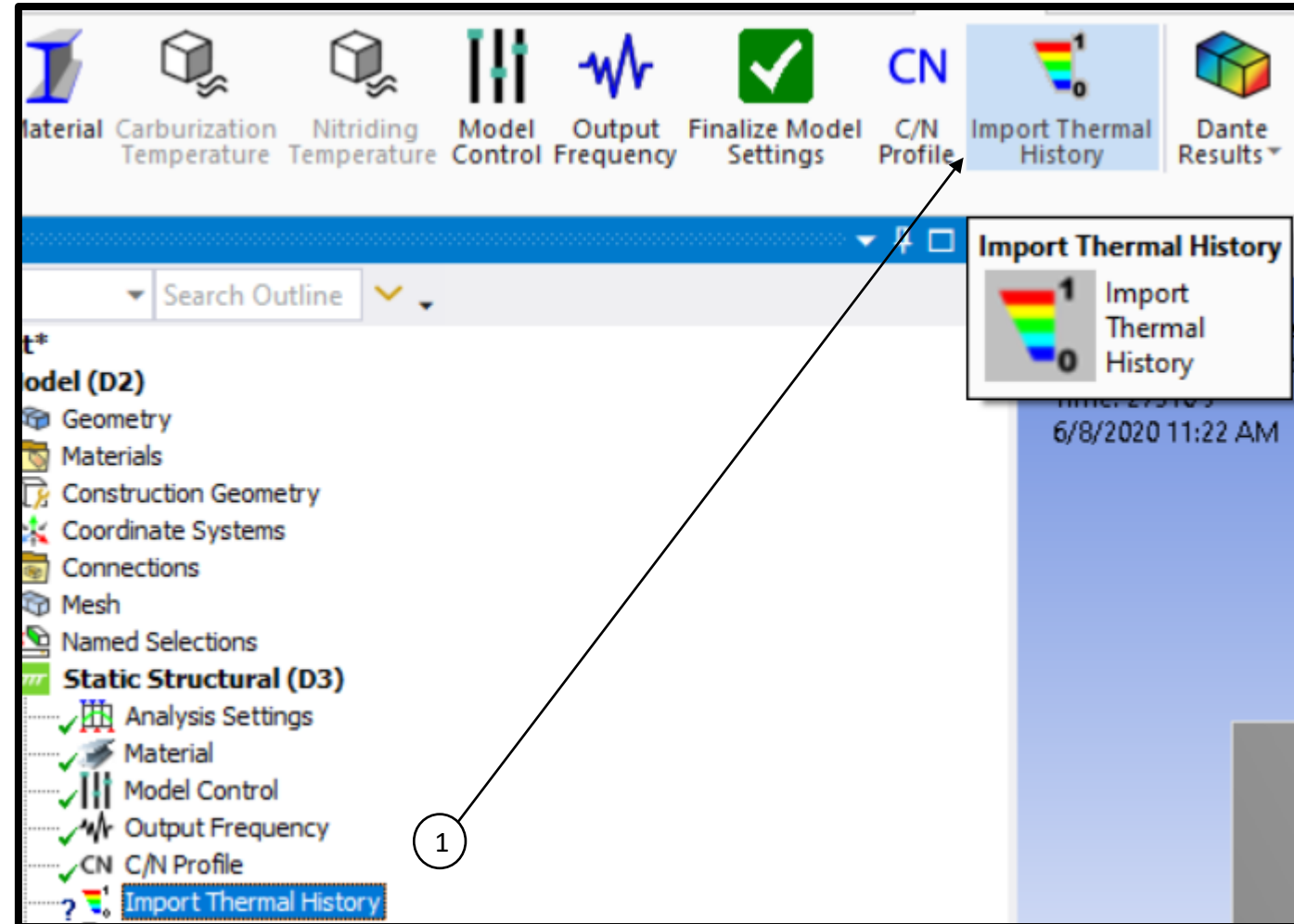
# Step 10b: Import Thermal History Alternative

**NOTE:** This is an alternative way to import the thermal history and should not be used if the previous method was used to import the thermal history

1. Click on the **Import Thermal History** button in the Dante toolbar to insert it into the Model Tree
2. In the details of Import Thermal History, navigate to the sliced\_ring.tem file in the dante files folder in the project directory



2



1

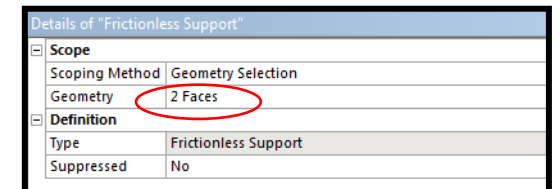
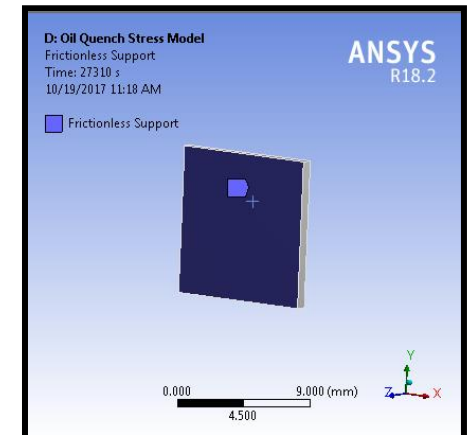
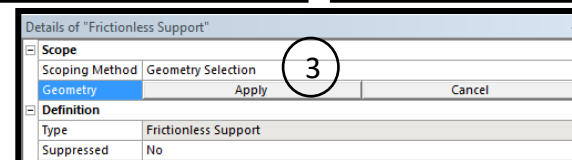
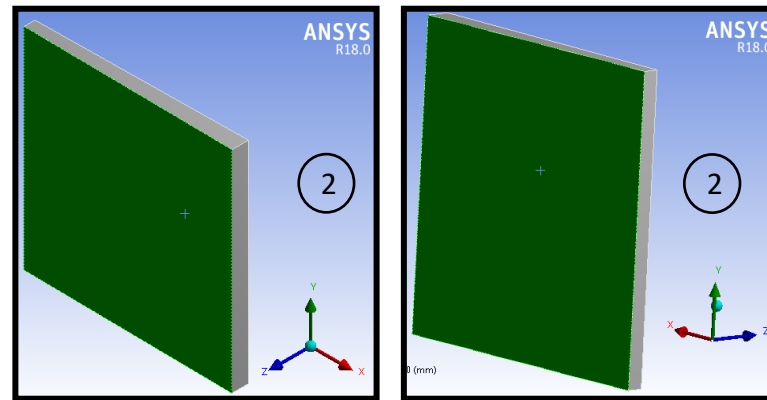
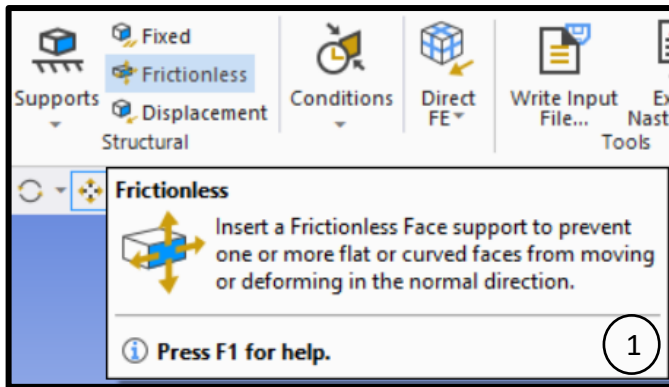
# Step 11: Define Mechanical Boundary Conditions

The geometry simplifications are built around two assumptions:

1. The thin slice is representative of ANY thin slice taken from the ring in the circumferential direction (cyclic symmetry)
2. The top half of the ring, in the y-direction, acts like the bottom half (symmetric about xz-plane)

To model the cyclic symmetry, Frictionless Supports will be used on the surfaces which would be internal to the ring

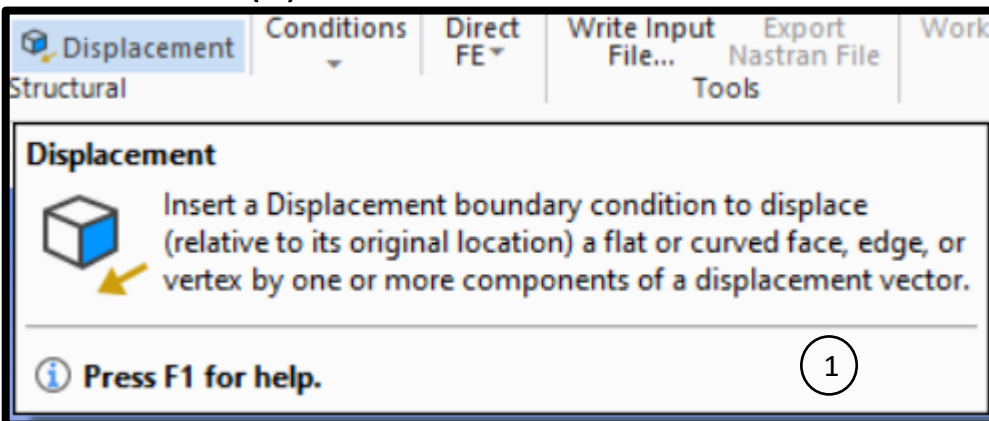
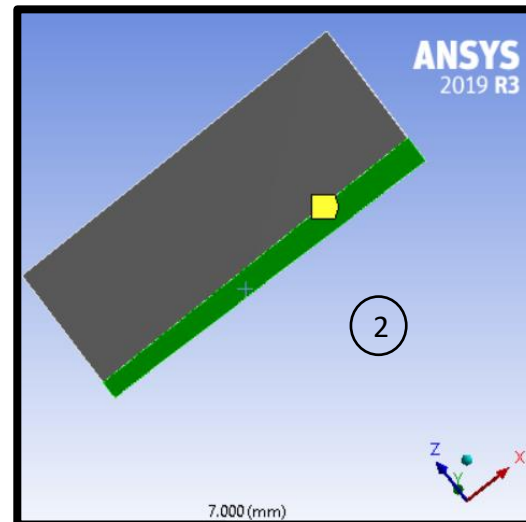
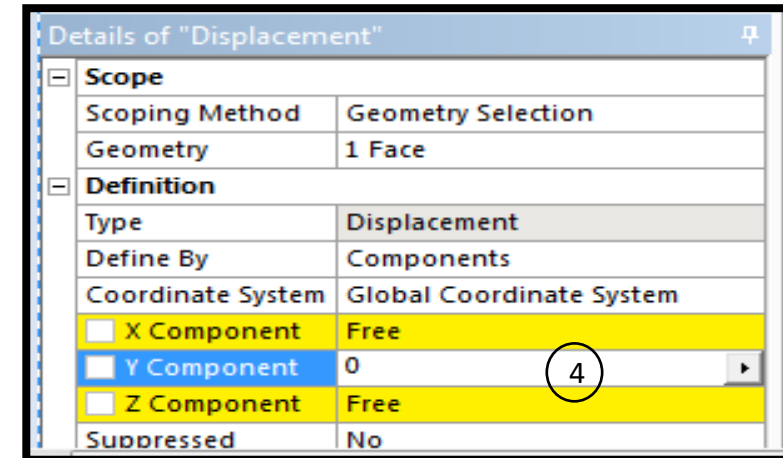
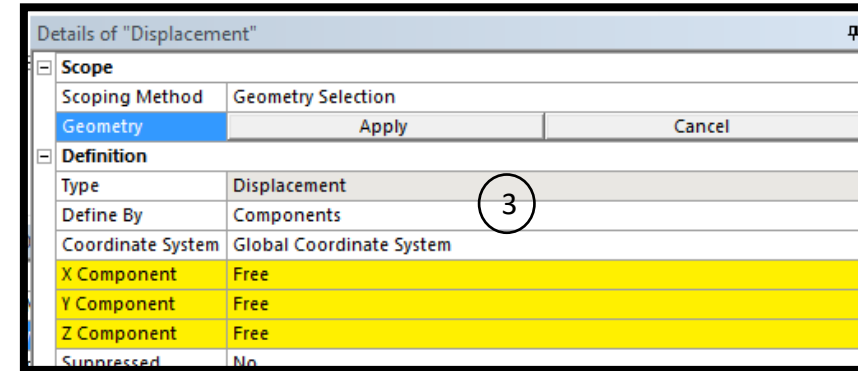
1. Expand Supports in the **Environment** toolbar and select **Frictionless Support**
2. Select the face shown in green, rotate the part 180°, hold the Ctrl button on the keyboard, and select the opposite face
3. Click **Apply** for Geometry in the Details of “Frictionless Support”



# Step 12: Define Mechanical Boundary Conditions

To model the symmetry about the xz-plane, Displacement will be used on the surface lying on the xz-plane which would be internal to the ring.

1. Expand Supports in the **Environment** toolbar and select **Displacement**
2. Select the face shown in green; note the coordinate system in the lower left of the figure
3. Click **Apply** for Geometry in the Details of "Displacement"
4. Symmetry about the xz-plane means the surface selected has no displacement in the y-direction; surfaces above this surface move in the positive y-direction and surfaces below this surface, if they were modeled, move in the negative y-direction. Therefore, change the Y Component in the Details of "Displacement" to zero (0)

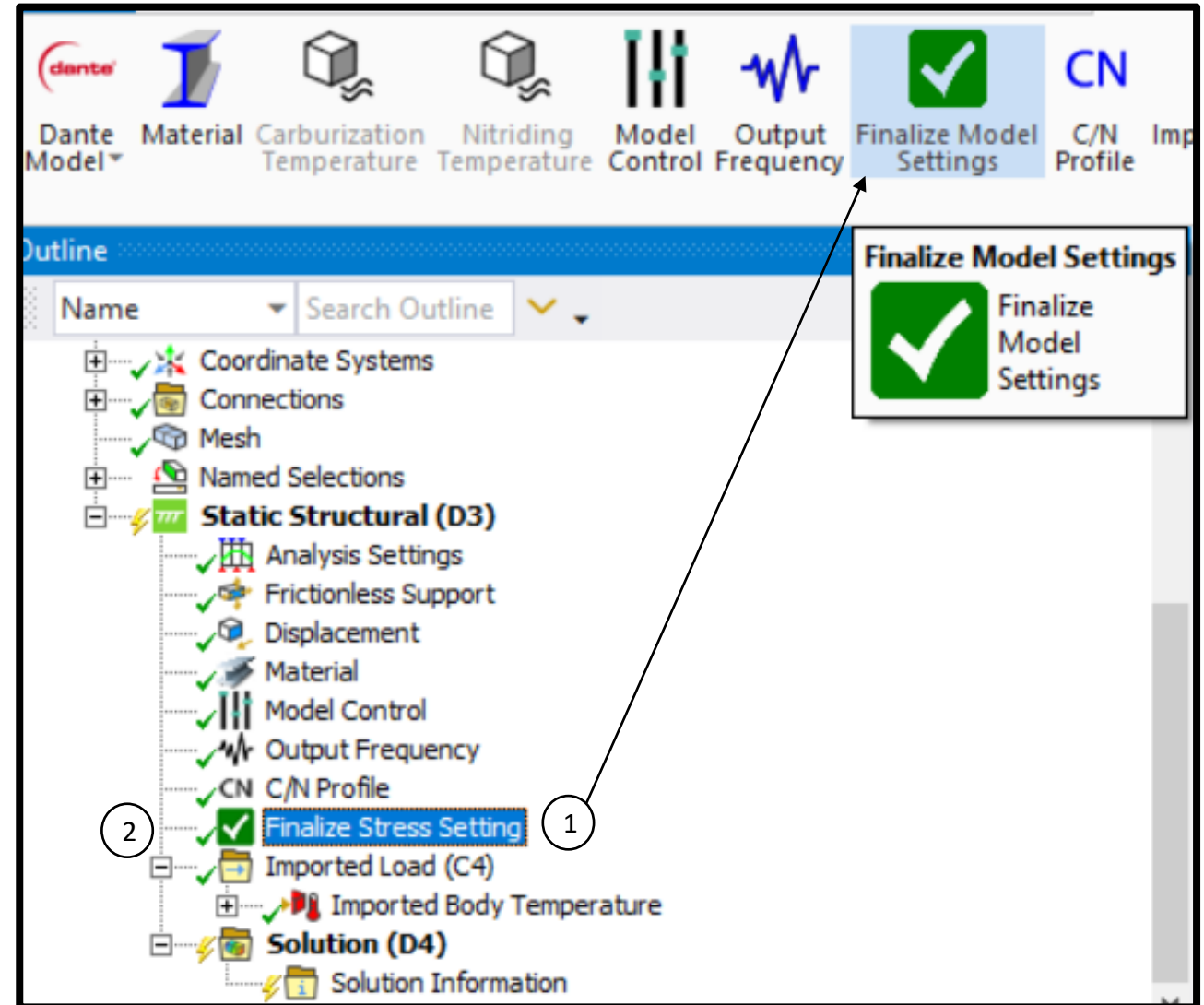
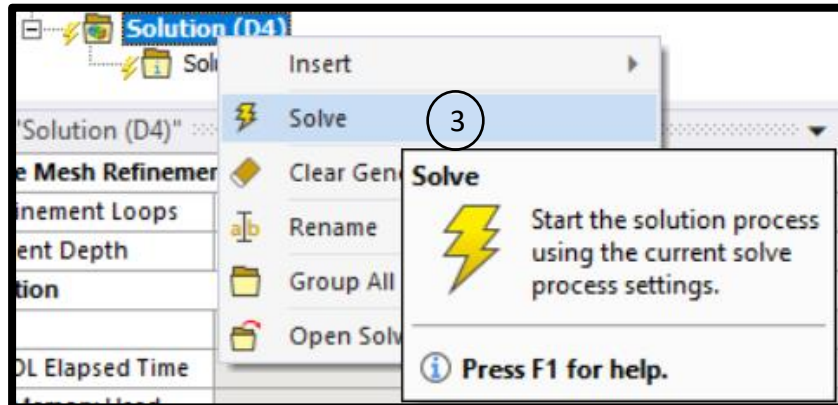




# Step 13: Running the Stress Model

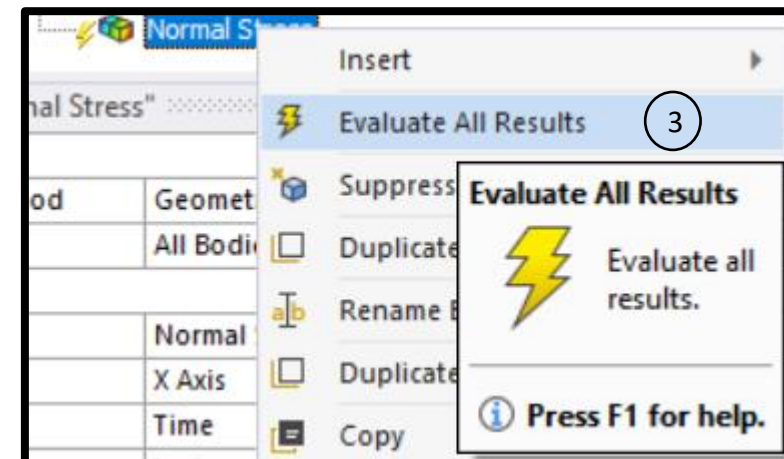
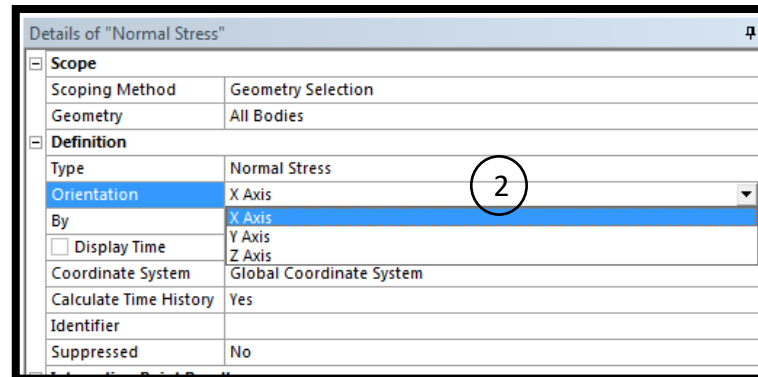
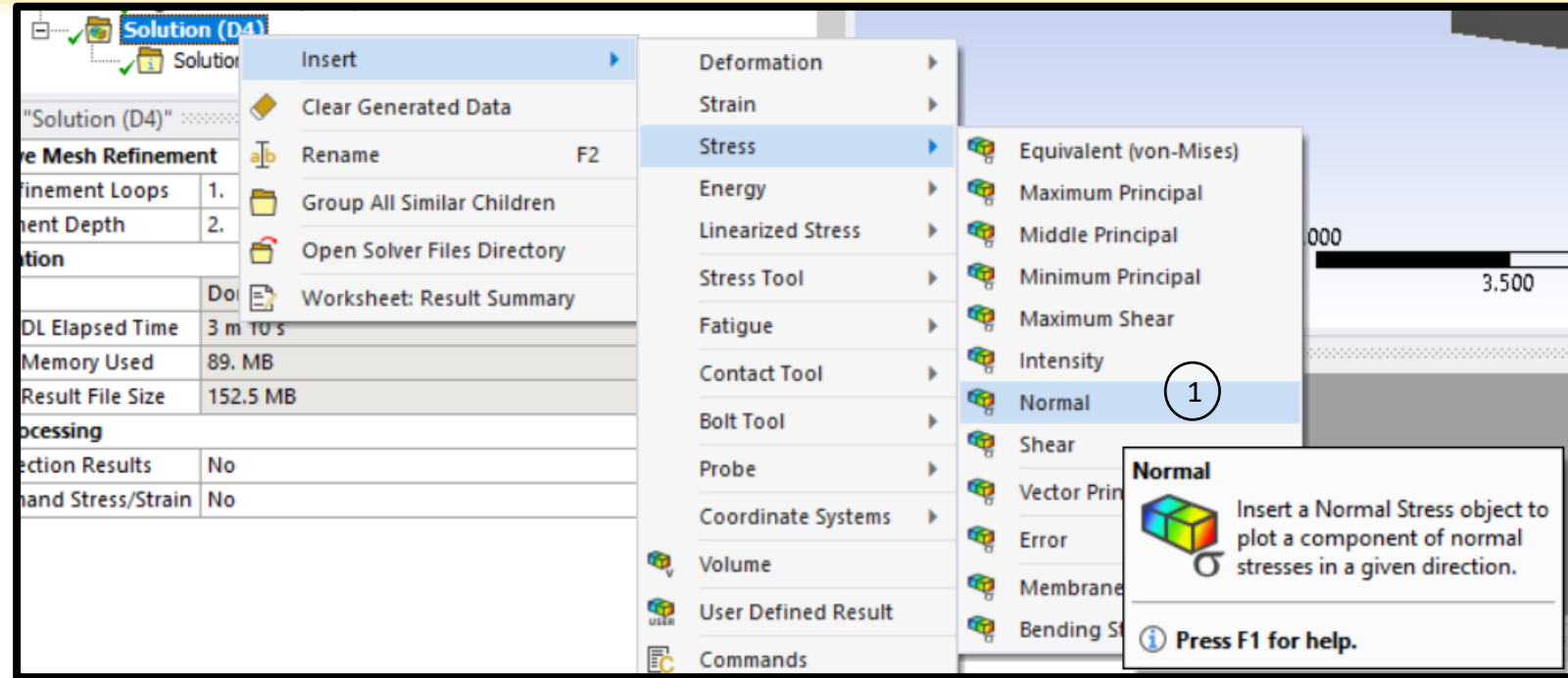
1. Select the **Finalize Model Settings** in the Dante toolbar to insert it into the Model Tree
2. This performs a series of checks on the stress model and if all pass, then a green check mark should appear next to the Finalize Stress Setting object in the Model Tree
3. Right-click on the Solution and select **Solve** to run the model

**NOTE: When the solution is complete, save the Project and move on to the next page to begin post-processing of the Stress Model**



# Step 14: Stress Model Post-Processing, Stress Results

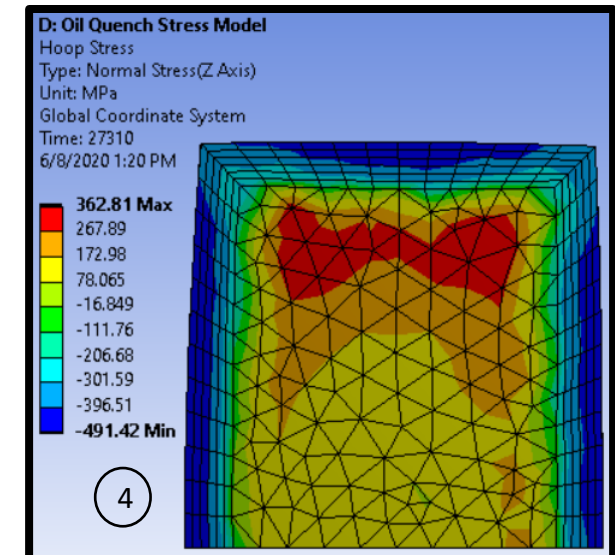
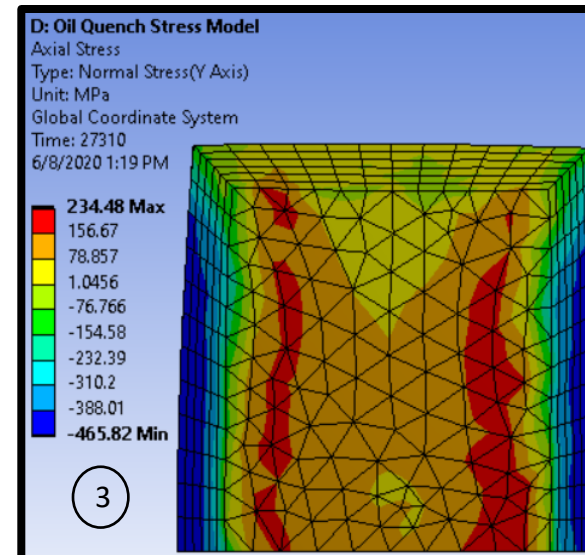
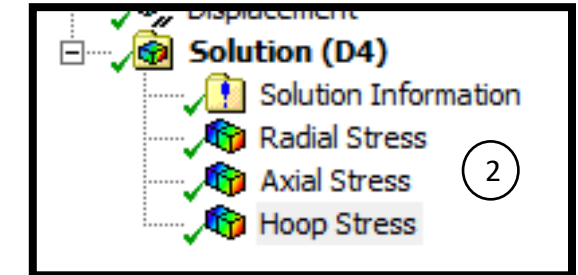
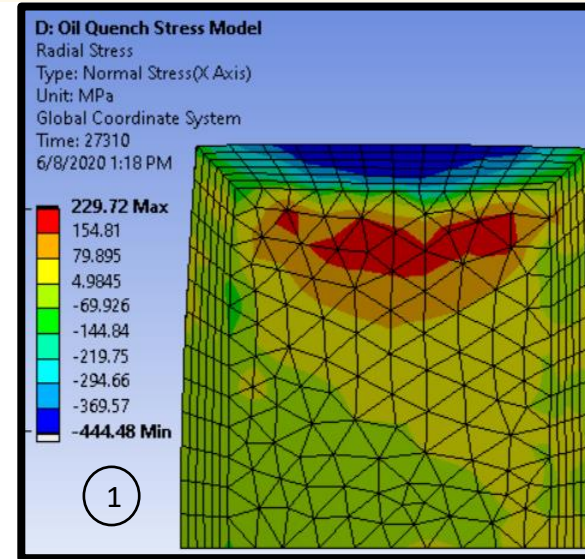
1. To view the stress results, either select stress from the Solution tab or right-click on **Solution**, hover over **Insert**, **Stress**, and select **Normal**
2. In the Details of "Normal Stress", expand the Orientation tab to select the direction of interest. For this example lets choose **X Axis**, which is the radial stress
3. Right click on Normal Stress in the Project Tree and select **Evaluate All Results**





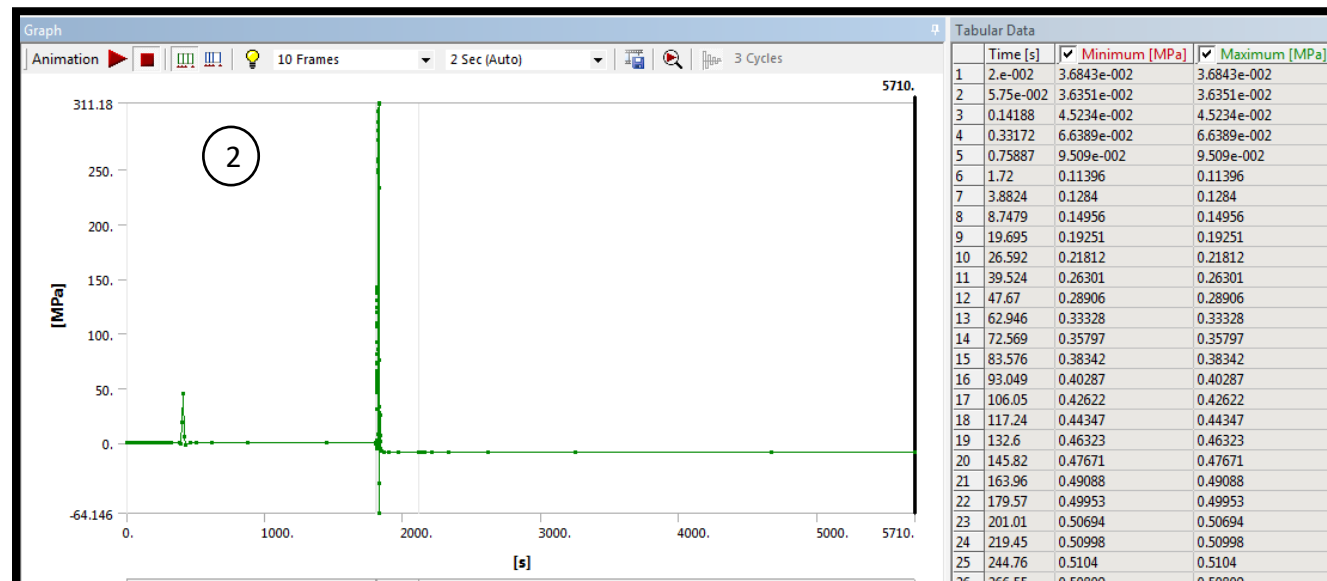
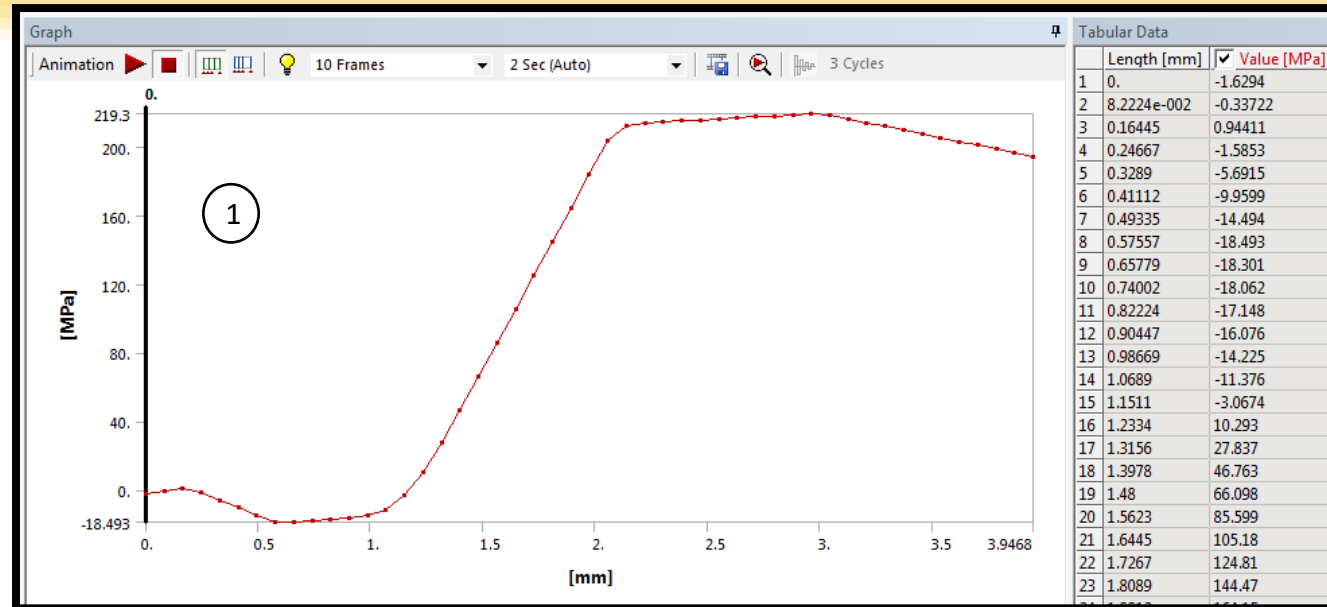
# Step 14b: Stress Model Post-Processing, Stress Results

1. The contour plot at the final time, along with a table displaying each saved substep with the corresponding time, maximum value, and minimum value is displayed in the Tabular Data
2. The Normal Stress result can then be renamed Radial Stress and two more Normal stress results can be added to the Solution in the Project Tree, renamed Axial Stress (choose Y Axis Orientation) and Hoop Stress (choose Z Axis Orientation), and then choose Evaluate All Results after right clicking on Axial Stress under Solution in the Project Tree
3. The contour for the Axial Stress
4. The contour for the Hoop Stress



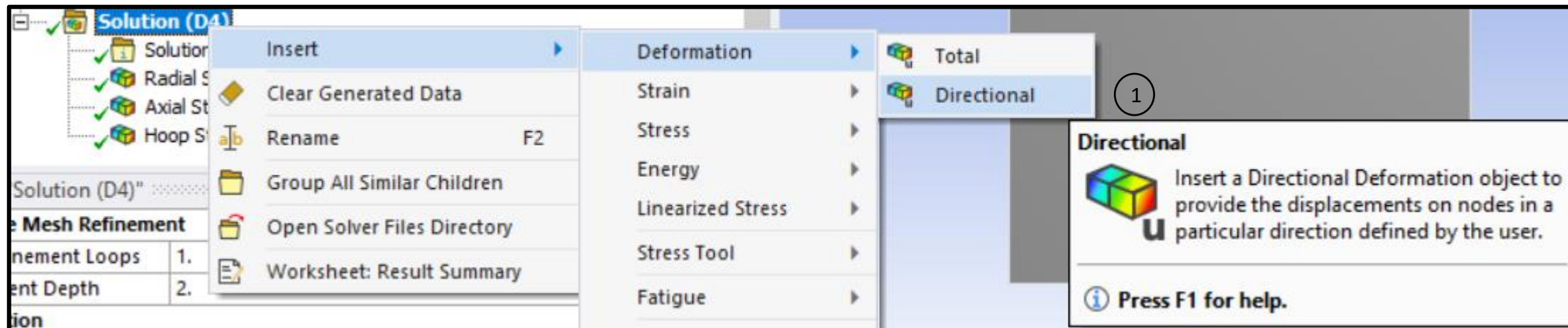
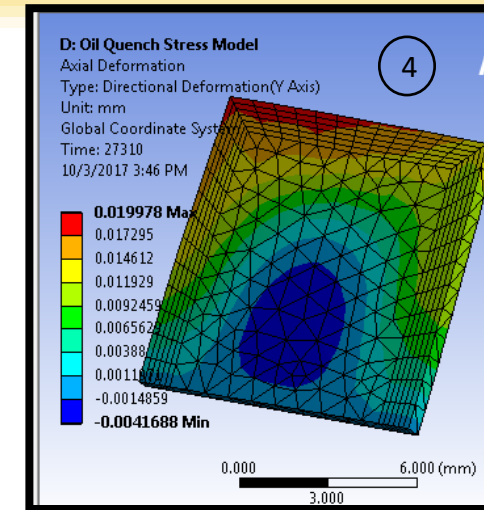
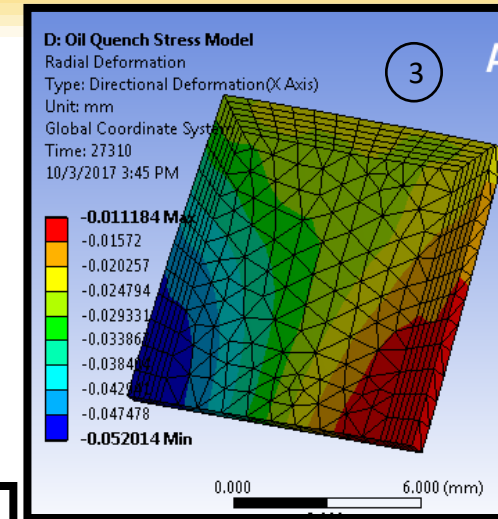
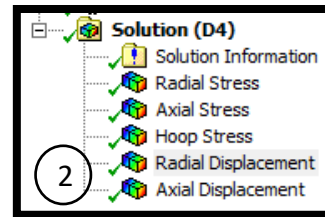
# Step 14c: Stress Model Post-Processing, Stress Results

1. A stress profile over a given path can also be viewed and exported to an external file as described in an earlier section of this tutorial; **Post-Processing Carburization Model: Plot Carbon Profile**
2. Stress versus time for a given node can also be viewed and exported to an external file as described in an earlier section of this tutorial; **Post-Processing Thermal Model: Temperature History Plot**



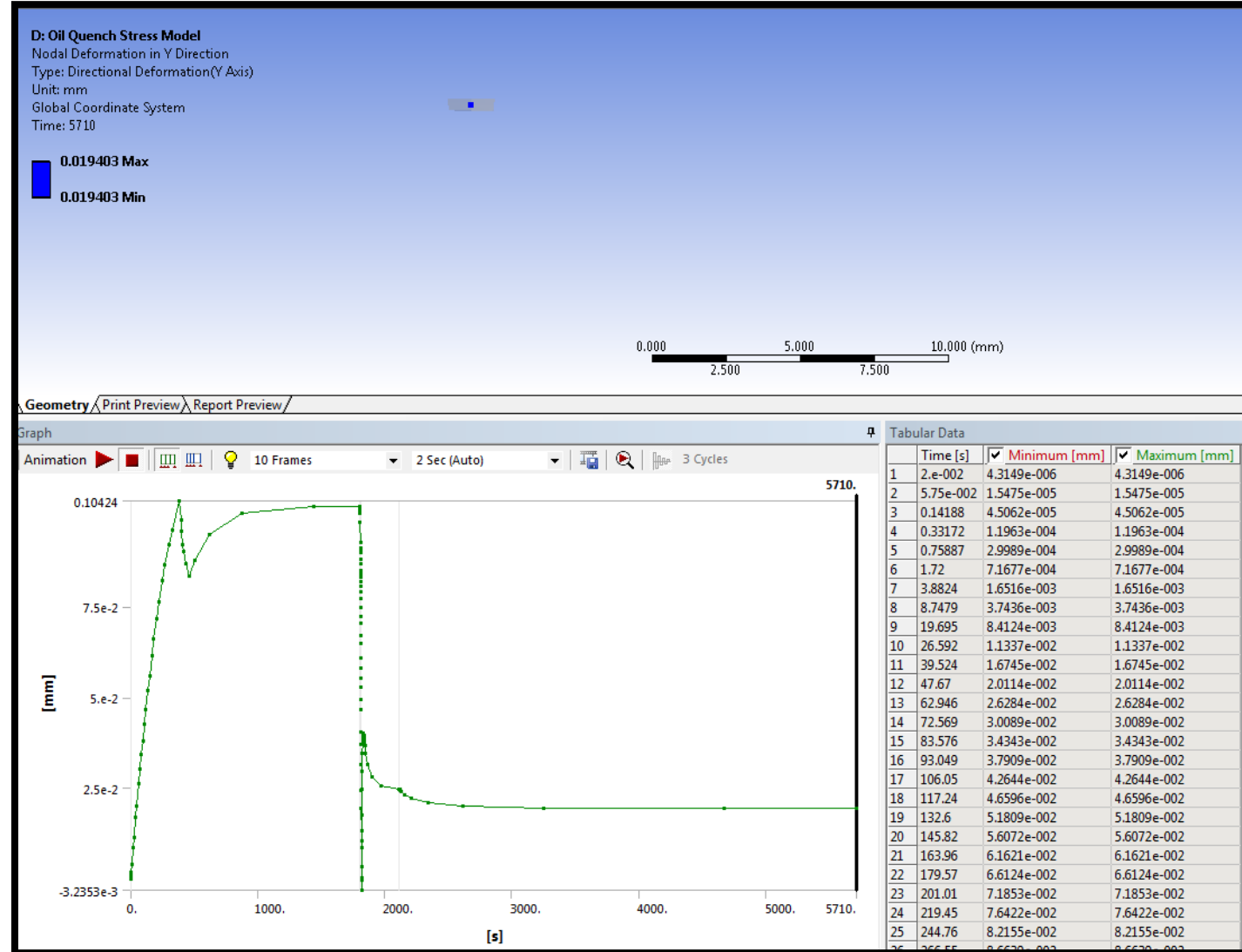
# Step 15: Stress Model Post-Processing, Displacement Results

1. The same procedure used to look at the Normal Stresses can also be used to examine the directional displacements (distortion). The only difference is the Deformation tab in the Solution toolbar is expanded and Directional is selected
2. Several Deformations can be added to the solution and renamed to make post-processing more efficient
3. The Radial Displacement (X Axis) contour is shown
4. The Axial Displacement (Y Axis) contour is shown



# Step 15b: Post-Process Stress Model, Displacement Results

- Displacement versus time for a given node can also be viewed and exported to an external file as described in an earlier section of this tutorial





# Step 16: Stress Model Post-Processing, Dante Results

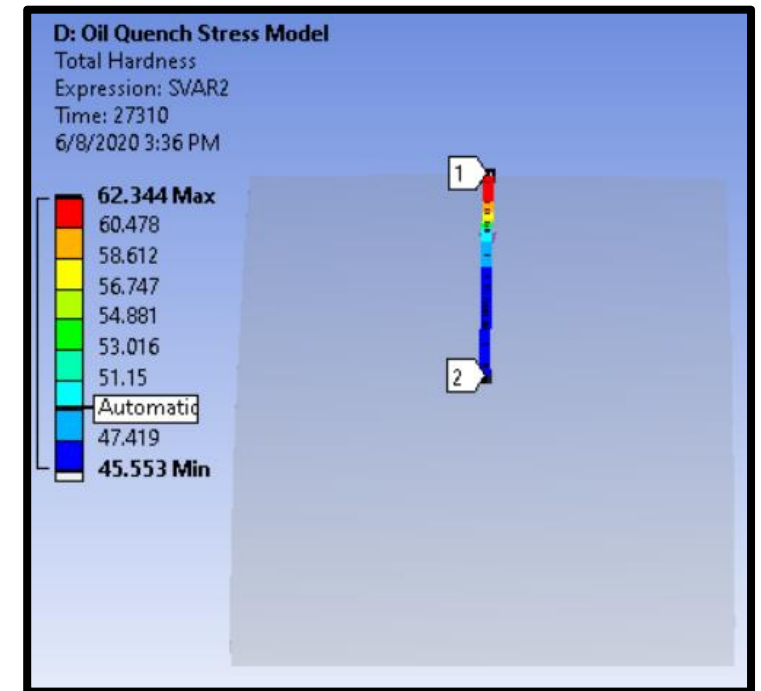
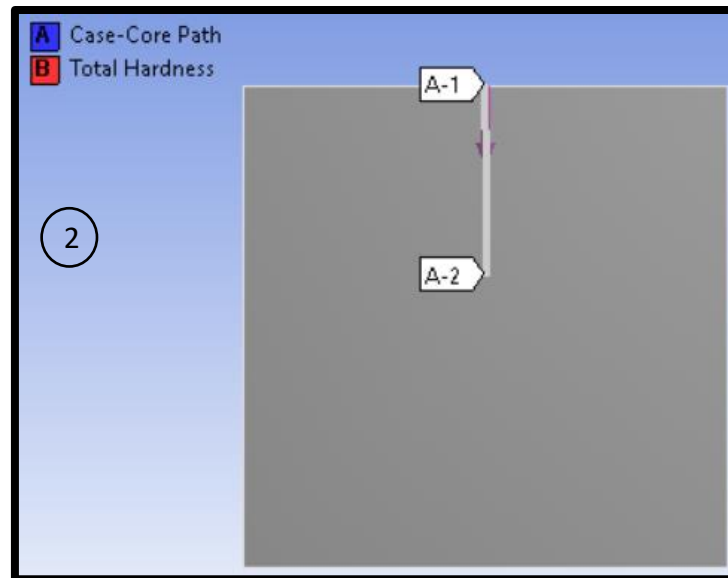
1. Viewing the **Dante Results** in the Stress Model is identical to viewing the Dante Results in the Thermal Model, as explained earlier in this tutorial
2. The results list is the same except that Effective Plastic Strain can be accessed. By inserting a Dante result object (generally called a User Defined Result)



# Step 17: Dante Results Path Plot

1. Path plots can be generated by setting the scoping method of any result object to Path and selecting a Path
2. In this case, the previously created Case-Core Path can be redefined to visualize a Dante result such as the hardness distribution

Details of "User Defined Result"	
<b>Scope</b>	
Scoping Method	Path <span style="border: 1px solid black; border-radius: 50%; padding: 2px;">1</span>
Path	Case-Core Path
Geometry	All Bodies
<b>Definition</b>	
Type	User Defined Result
Expression	= SVAR2
Input Unit System	Metric (mm, kg, N, s, mV, mA)



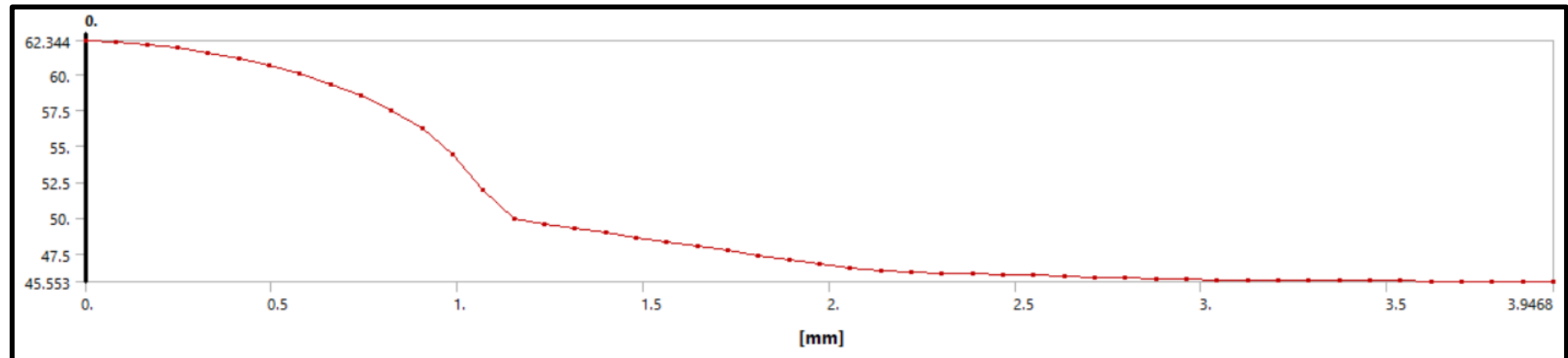
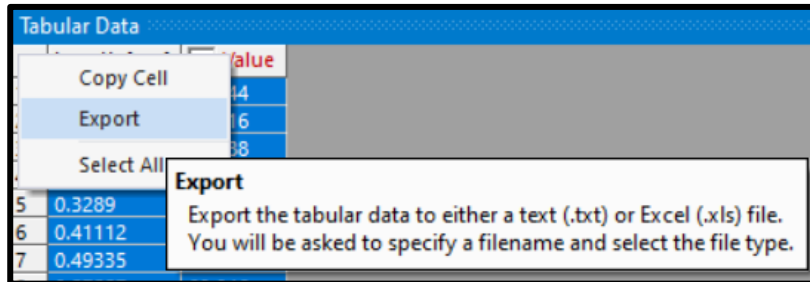
# Step 17b: Dante Results Path Plot

1. The hardness profile in Rockwell C for the path is tabulated and graphed in Ansys as well
2. By right-clicking on the top left corner of the tabular data table, the data can be exported as a .txt or .xls file format

1

Tabular Data		
	Length [mm]	Value
1	0.	62.344
2	8.2224e-002	62.216
3	0.16445	62.088
4	0.24667	61.828
5	0.3289	61.513
6	0.41112	61.114
7	0.49335	60.592
8	0.57557	60.046
9	0.65779	59.295
10	0.74002	58.544
11	0.82224	57.443
12	0.90447	56.262
13	0.98669	54.417
14	1.0689	51.958
15	1.1511	49.925
16	1.2334	49.58

2

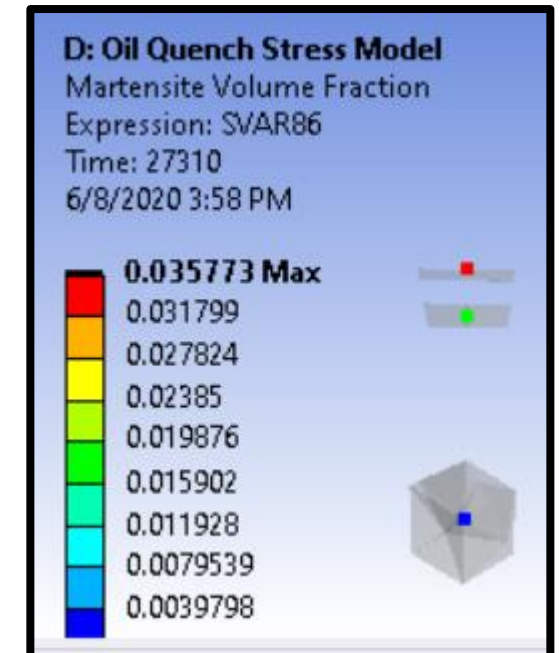
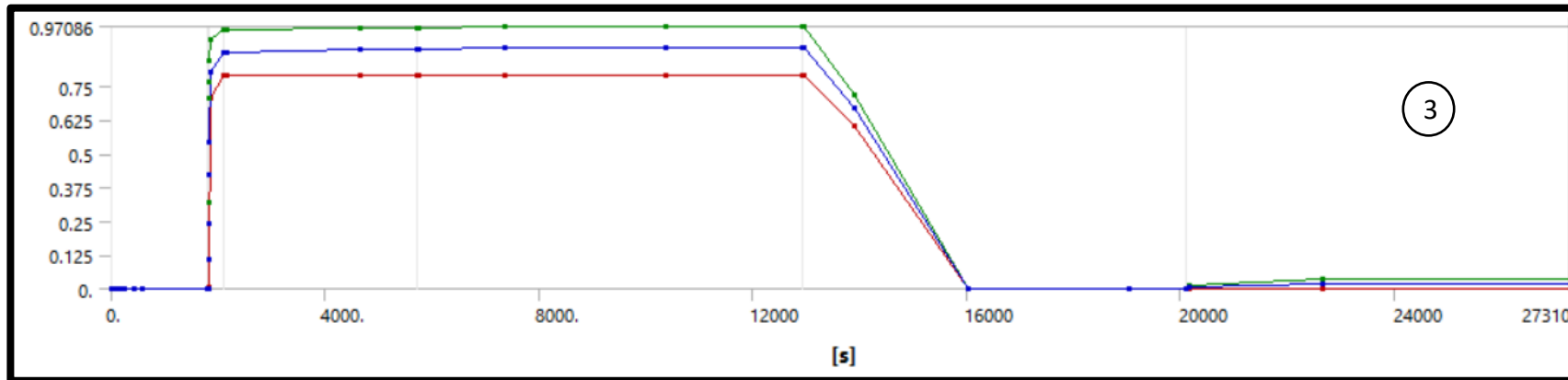




# Step 18: Dante Results History Plot

1. Define a nodal **Named Selection** as described in **Post-Processing Thermal Model: Temperature History Plot**. This time choose 3 nodes, hold ctrl on the keyboard to make multiple selections, and name the Named Selection "3 Nodes"
2. In a Dante result object, set the **Scoping Method** to **Named Selections** and select the **3 Nodes** selection
3. Evaluating this result will produce the time evolution of the amount of Martensite at the 3 points

Details of "Martensite Volume Fraction"	
Scope	
Scoping Method	Named Selection
Named Selection	3 Nodes
Definition	
Type	User Defined Result
Expression	= SVAR86
Input Unit System	Metric (mm, kg, N, s, mV, mA)
Output Unit	
By	Time



# Step 18b: Dante Results History Plot

- History data can also be exported in .txt or .xls format for further data analysis and post-processing

Tabular Data				
Time [s]	<input checked="" type="checkbox"/> Minimum	<input checked="" type="checkbox"/> Maximum	<input checked="" type="checkbox"/> Average	
1800.8	0.	0.	0.	
1805.5	0.	0.	0.	
1810.	0.	0.	0.	
1810.5	0.	0.	0.	
1810.9	0.	0.	0.	
1811.4	0.	0.	0.	
1811.9	0.	0.	0.	
1812.9	0.	0.	0.	
1814.7	0.	0.	0.	
1816.9	0.	0.31842	0.10614	
1819.	0.	0.7033	0.23954	
1821.7	0.	0.76607	0.42493	
1826.1	4.5173e-003	0.84452	0.54374	
1840.3	0.70266	0.92546	0.805	
2110.	0.78778	0.95731	0.87268	
2110.8	0.78778	0.95739	0.87284	
2154.3	0.7878	0.95977	0.87669	
4666.7	0.78784	0.96511	0.88487	
5710.	0.78784	0.96512	0.88489	
5710.8	0.78784	0.96515	0.88494	
5754.3	0.78784	0.96619	0.88644	
7367.5	0.78786	0.97082	0.89293	
10368	0.78786	0.97086	0.89299	

**Export**  
Export the tabular data to either a text (.txt) or Excel (.xls) file.  
You will be asked to specify a filename and select the file type.

# Step 19: Export Node Stresses

1. Another way of extracting data from the model is through the Dante post-processing tool Export Node Stress
2. Adding this object to the solution model tree and evaluating will save a .dat file with the nodal stress tensor information at the selected time frame in the dante\_files directory
3. Right-click the Solution and Open Solver Files Directory, navigate back to the project directory and then to the dante\_files directory

