Dante 6.0 Tutorial

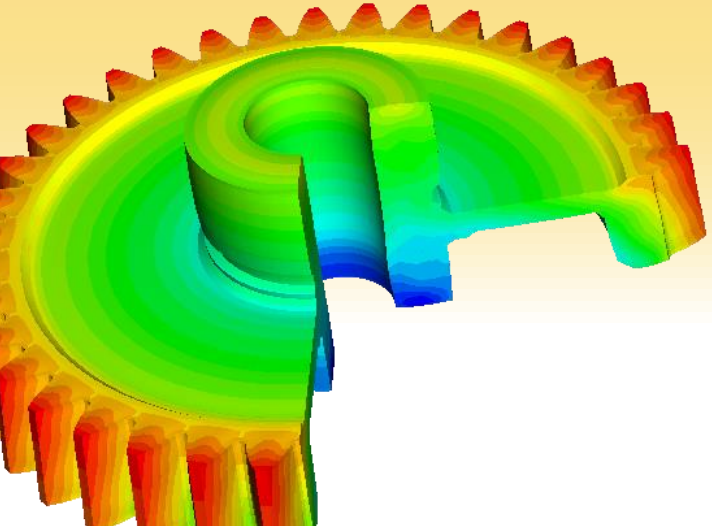
Nitriding 3D Ring Slice

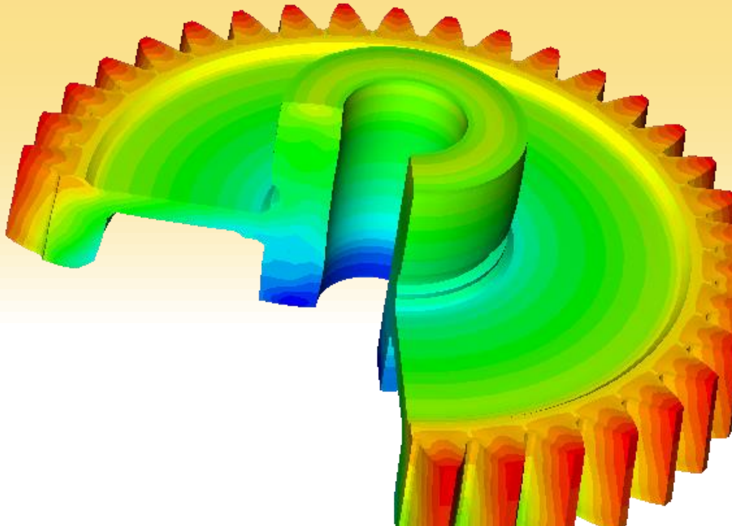
Coupled with ANSYS

Prepared By

DANTE Solutions, Inc.

Cleveland Oh





Introduction

Background

- This workshop will demonstrate nitriding modeling coupled between ANSYS 2020 R1 (or later) and DANTE using a simple sliced ring model
- This workshop includes sequentially coupled nitriding, thermal and stress models

Objectives

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

Thin Sliced Ring

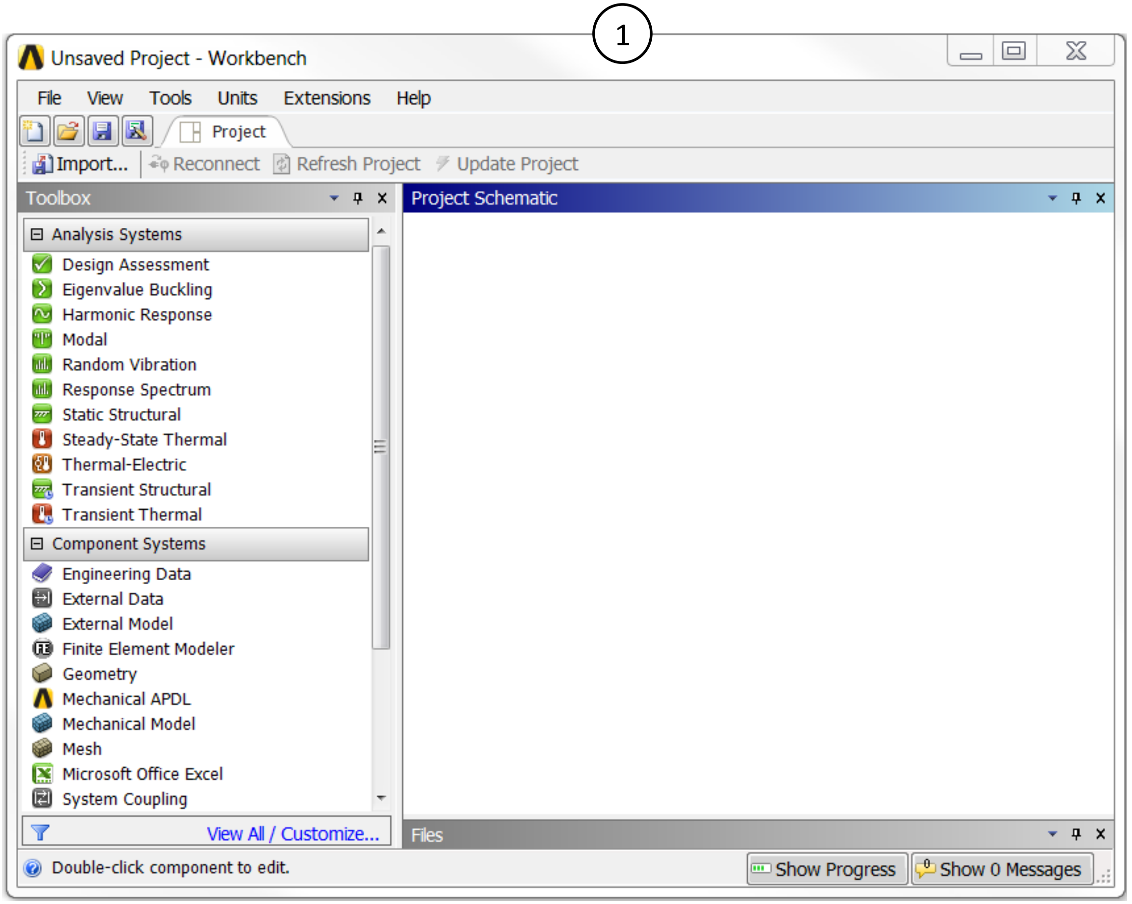
1. Model Geometry and Meshing
2. Nitriding Model
3. Nitriding Thermal Model
4. Nitriding Stress Model

Model Geometry and Meshing

Step 1: Start Ansys Workbench

1. Start ***Workbench***

NOTE: This workshop is developed under 2020-R1 but will work for any version 2020-R1 and later



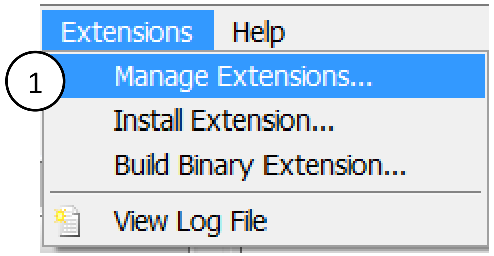
Software required for this project:

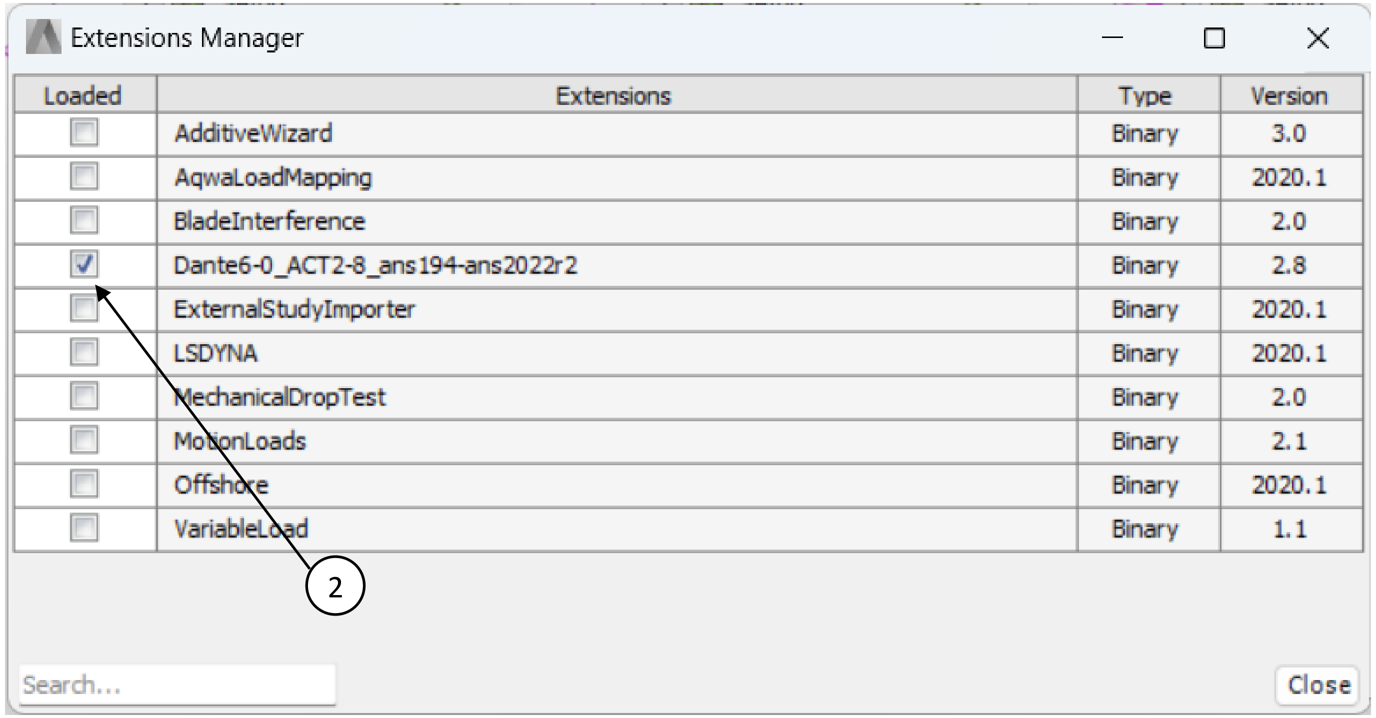
- ANSYS Mechanical Enterprise 2020 R1 or later
- DANTE (Compiled user subroutines and materials database)
- DANTE ACT

NOTE: All packages should be installed prior to this workshop

Step 2: Activate the DANTE ACT

1. Click ***Extensions*** → ***Manage Extensions***
2. In ***Extensions Manager***, check the ***DANTE ACT*** (Dante6-0_ACT2-8_ans194-and2022r2)





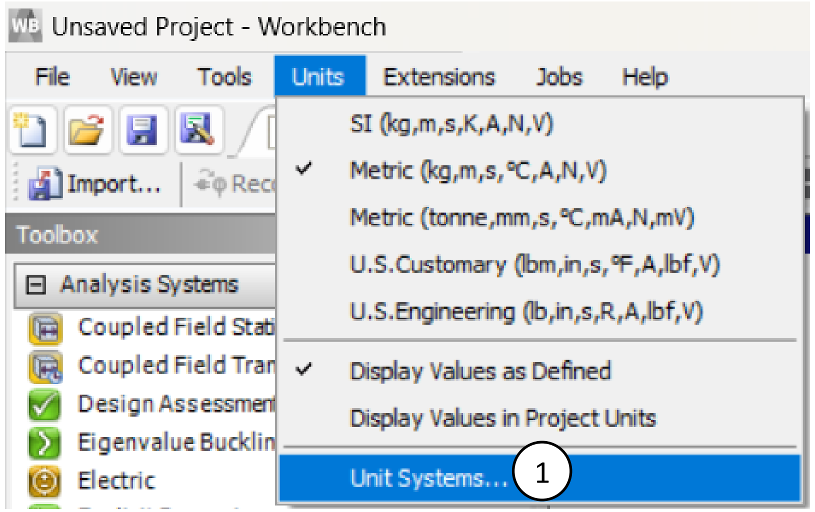
Step 3: Check Units

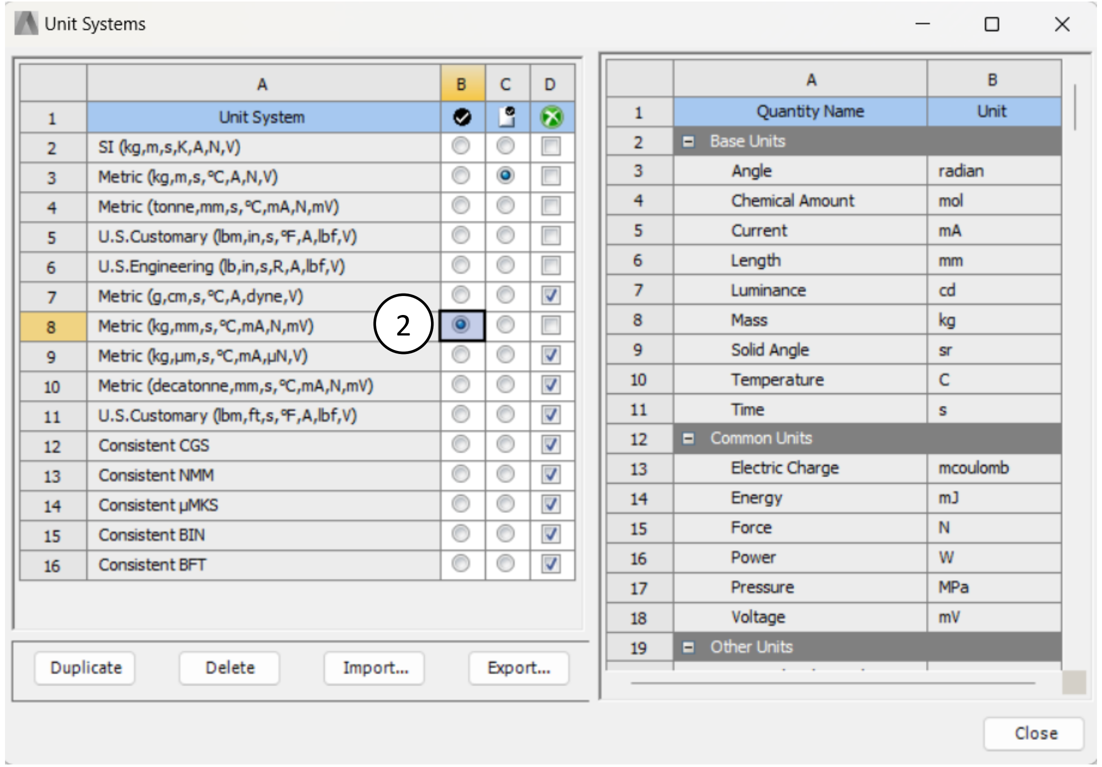
It is critical that the units be properly defined

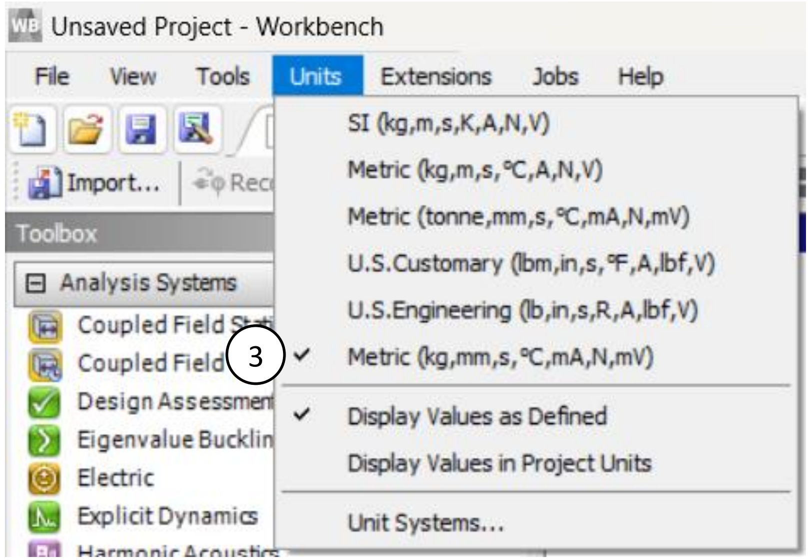
1. Select ***Units→ Unit Systems***
2. Toggle cell 8B for ***Metric (kg,mm,s,°C,mA,N,mV)***

   Note: If you prefer these units as default, also toggle cell 8C

3. Ensure ***Metric (kg,mm,s,°C,mA,N,mV)*** is checked





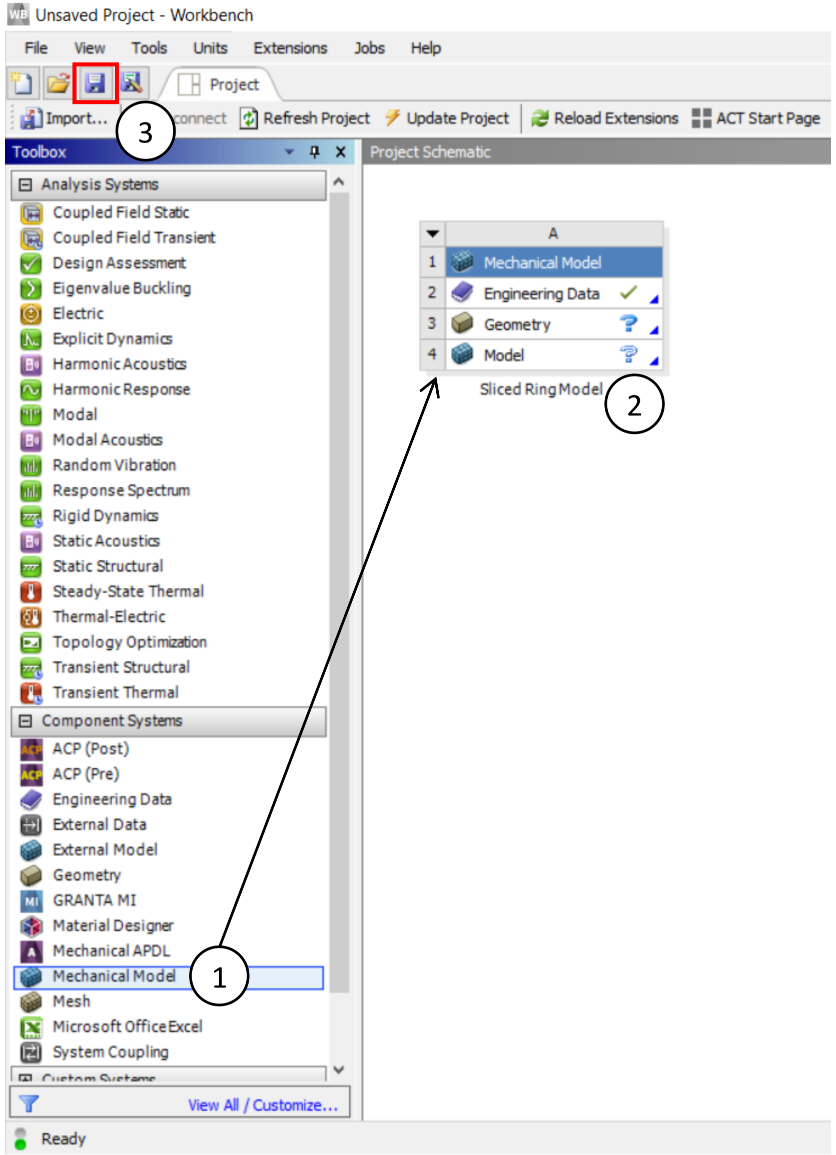


Step 4: Create Project

1. Drag and drop ***Mechanical Model*** from the **Component Systems** into the ***Project Schematic***

2. Rename it as "Sliced Ring Model"

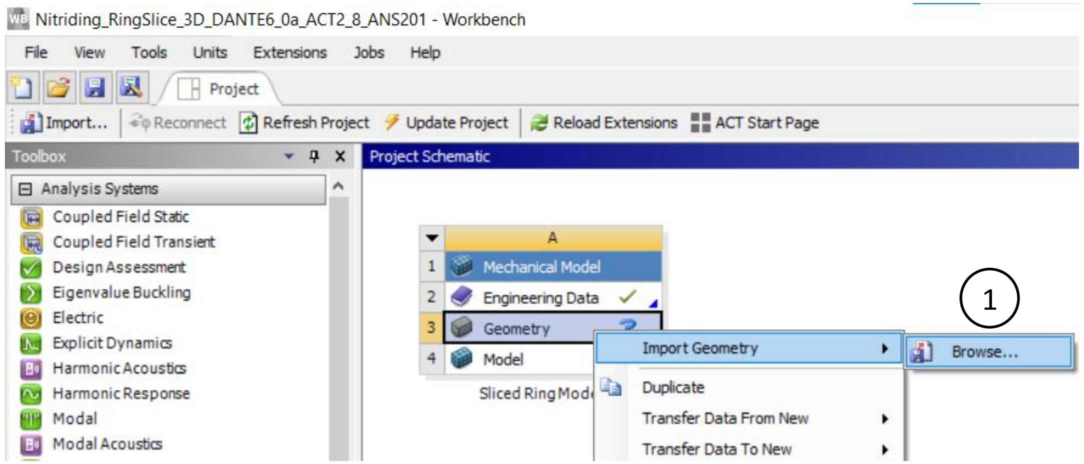
3. Save the project with a unique name

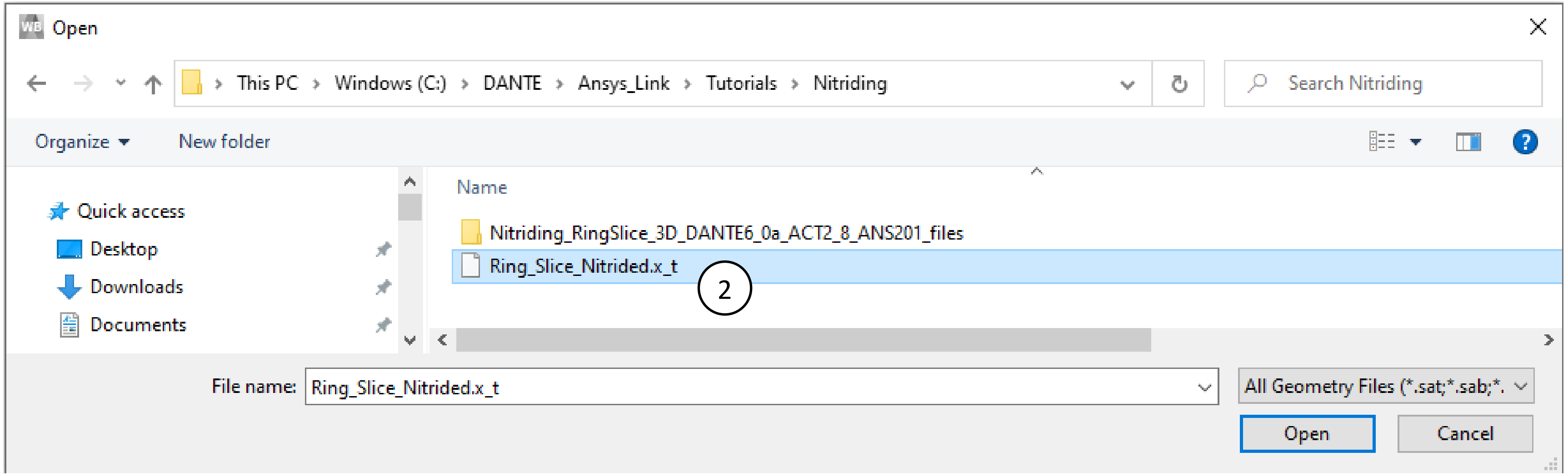


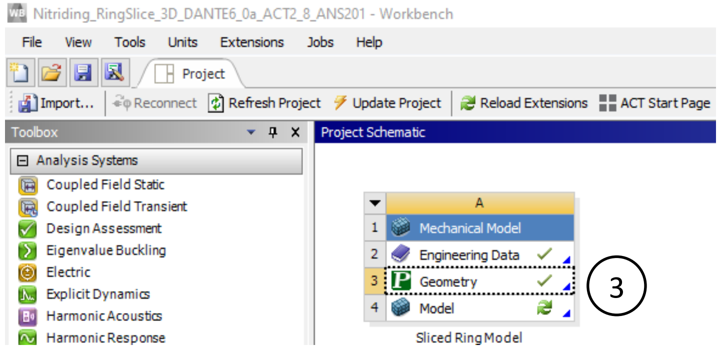
Step 5: Import the Geometry

The sliced ring geometry for this workshop will be imported from a CAD model created by the instructor

1. Right click ***Geometry*** → ***Import Geometry*** → ***Browse***
2. Open the ***Ring_Slice_Nitrided.x_t*** file provided
3. Double click ***Model***

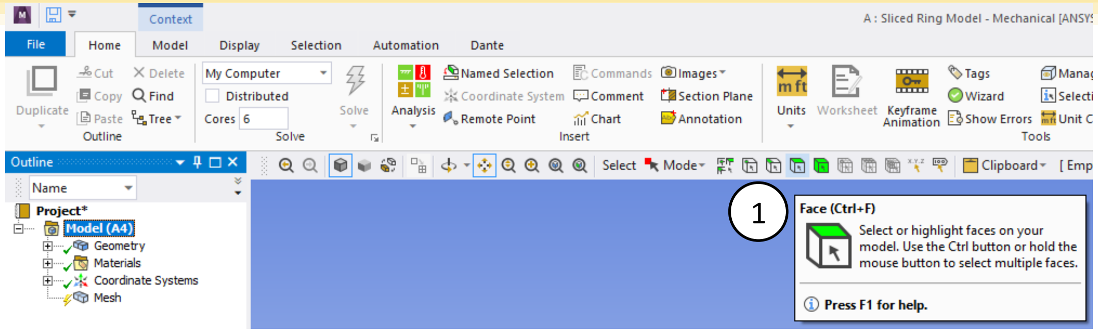


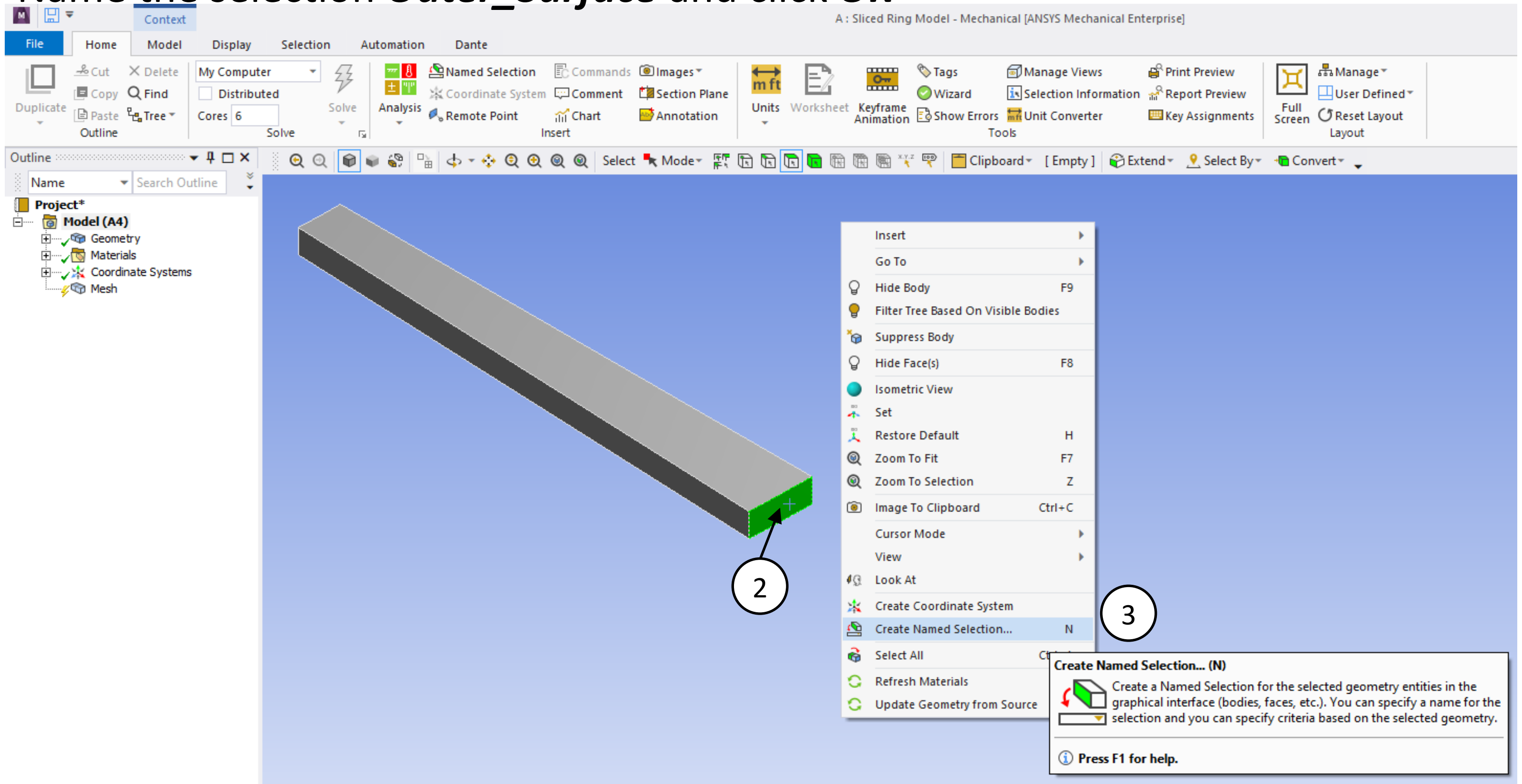


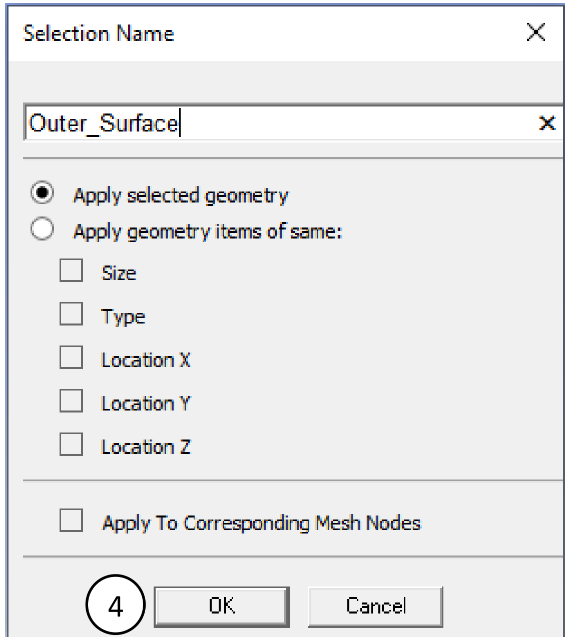


Step 6: Define Surfaces for Meshing & Boundary Conditions

1. Click ***Face Selection***
2. Select the outer surface
3. Right-click and select ***Create Named Section***
4. Name the selection ***Outer_Surface*** and click ***OK***



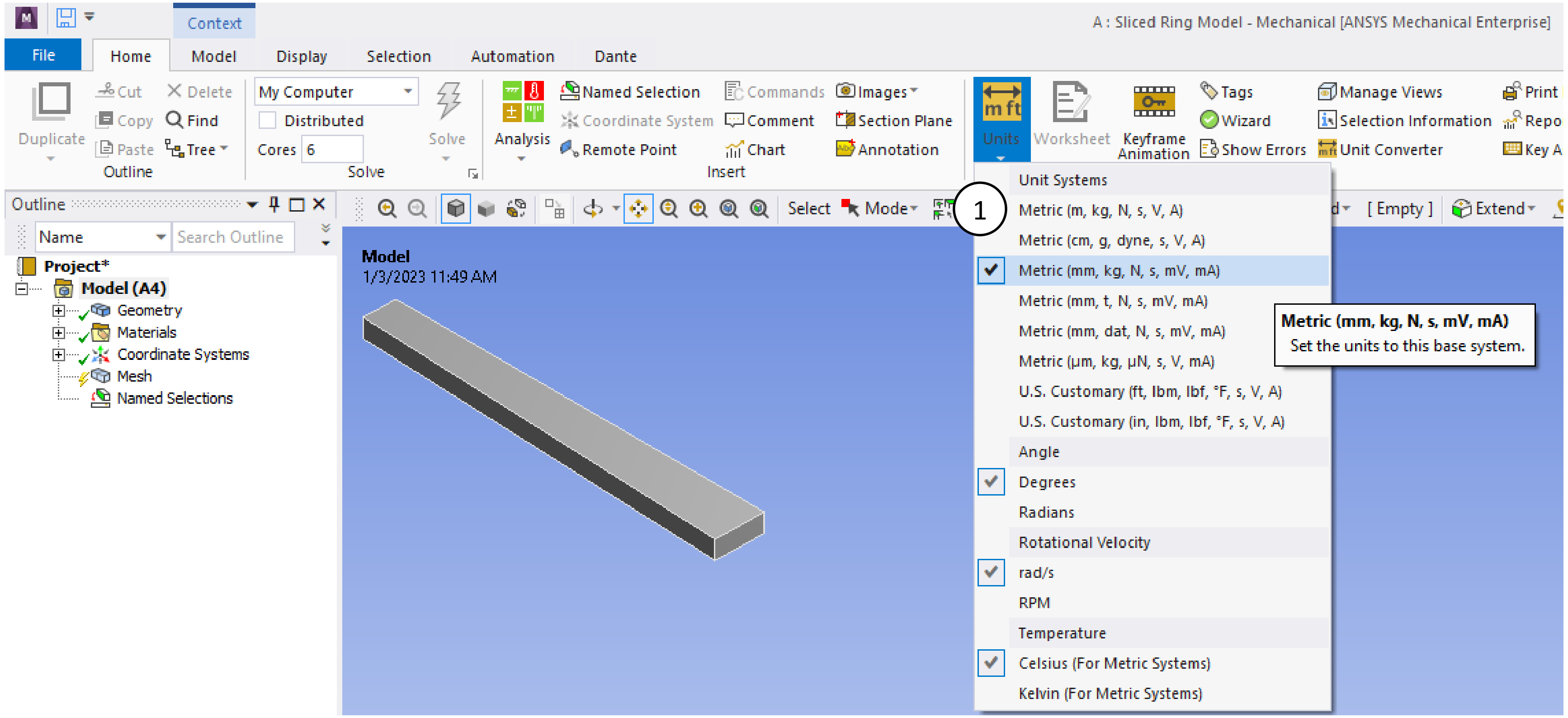




Step 7: Define Units

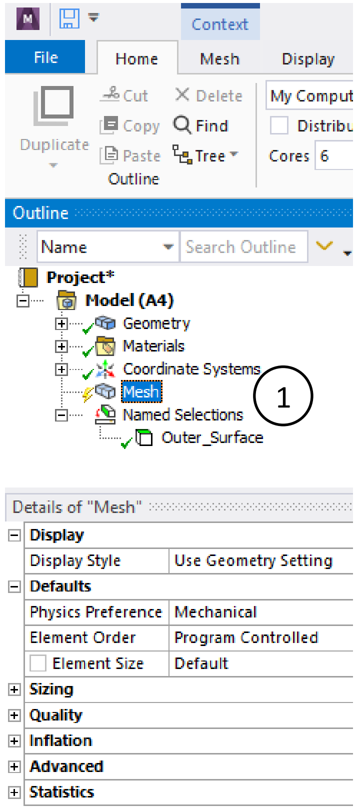
It is critical that the units be properly defined

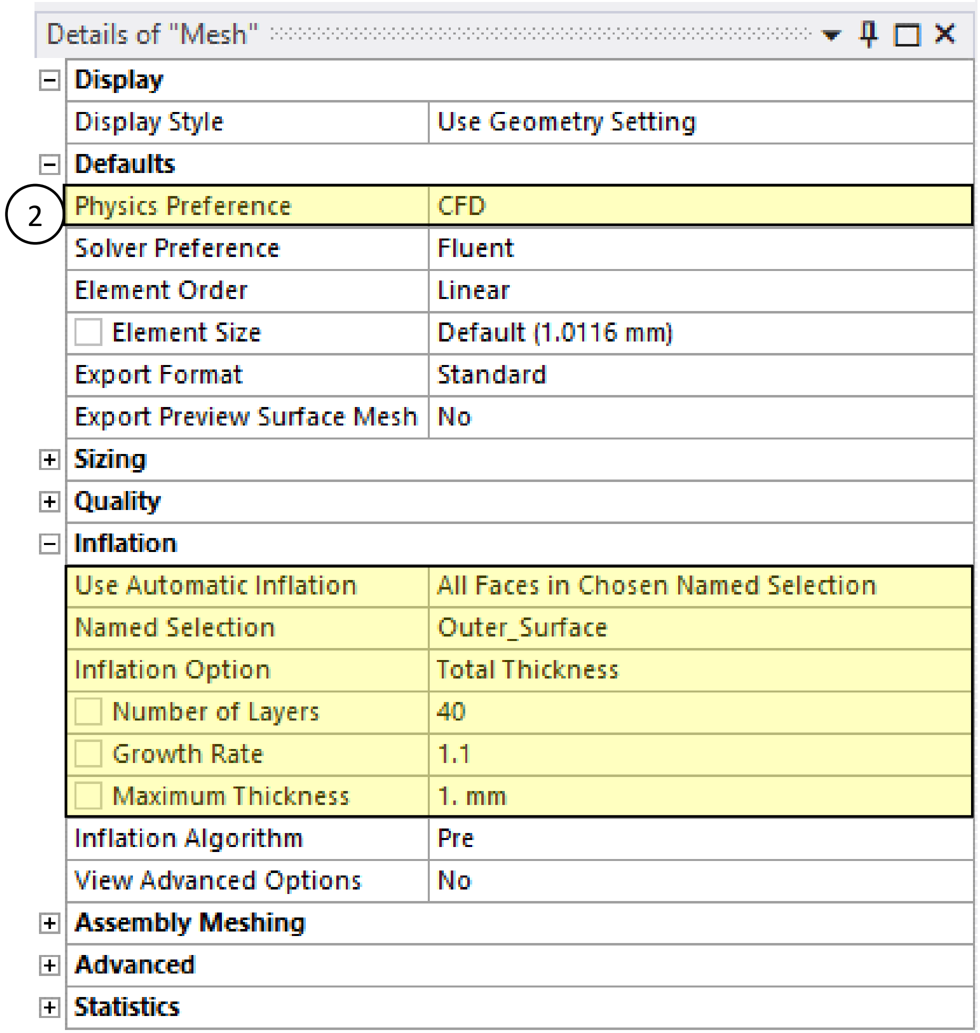
1. Select ***Units*** under ***Home***
   - ➢ ***Metric (mm, kg, N, s, mV, mA)***
   - ➢ ***Degrees***
   - ➢ ***rad/s***
   - ➢ ***Celsius (For Metric Systems)***



Step 8: Define Meshing Parameters

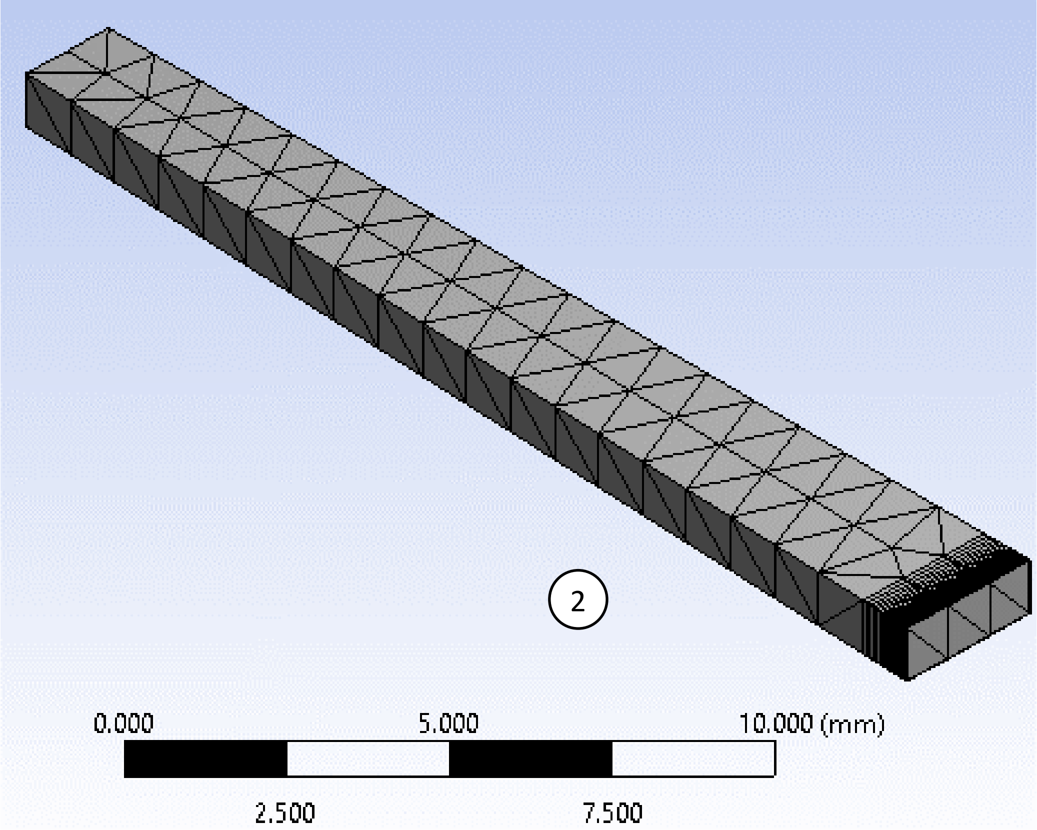
1. Select **Mesh** in the Project Tree
2. Modify the ***Details of "Mesh"***
   - ***Physics Preference → CFD***
   - ***Use Automatic Inflation → All Faces in Chosen Named Selection***
   - ***Named Selection → Outer_Surface***
   - ***Inflation Option → Total Thickness***
   - ***Number of Layers → 40***
   - ***Growth Rate → 1.1***
   - ***Maximum Thickness → 1.0 mm***

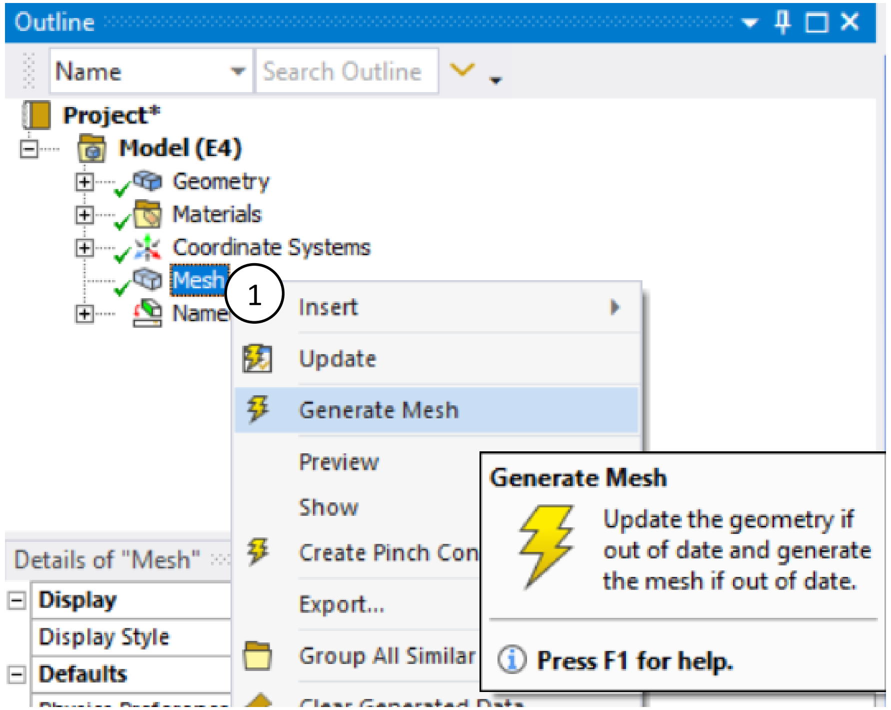


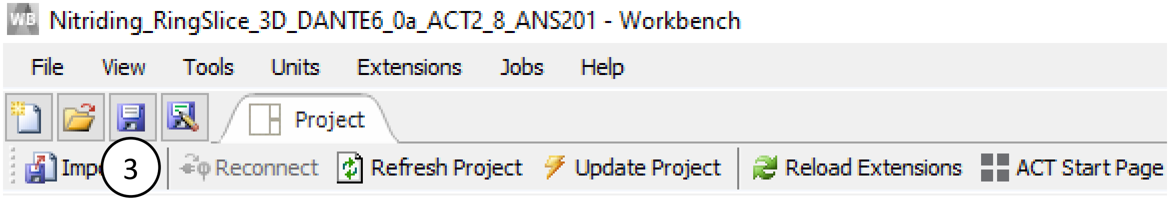


Step 9: Mesh the Component

1. Right click ***Mesh*** → **Generate Mesh**
2. The mesh should look like the image below
3. Close ANSYS Mechanical and save the project



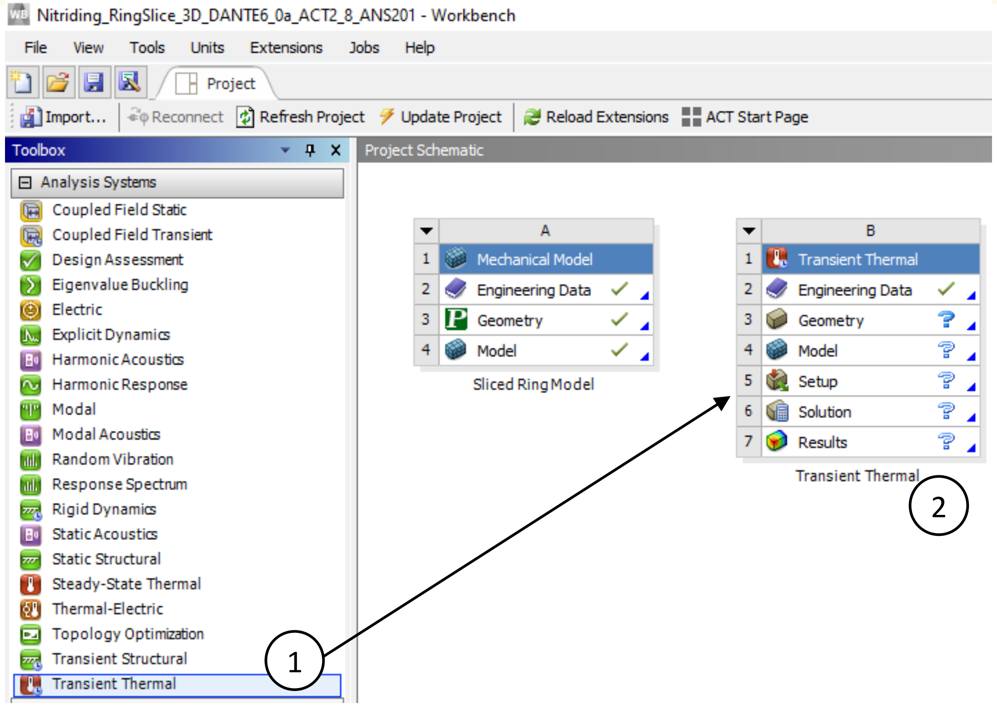


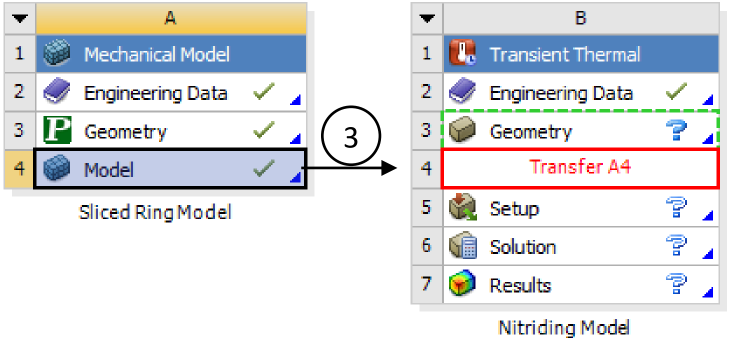


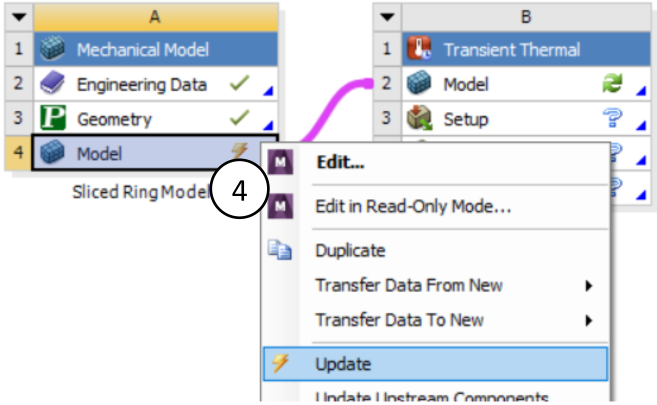
Nitriding Model

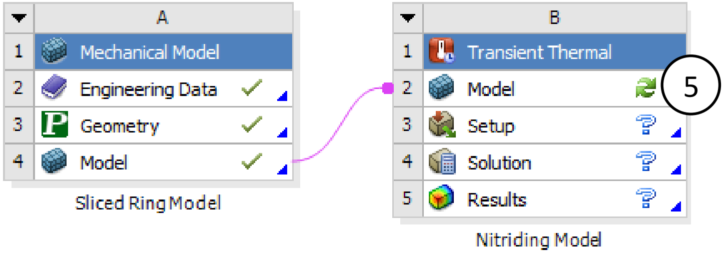
Step 1: Nitriding Model Analysis System

1. Drag and drop ***Transient Thermal Analysis System*** into the ***Project Schematic***
2. Rename it "Nitriding Model"
3. Drag and drop ***Model*** to ***Nitriding Model***
4. Right click ***Model*** → ***Update***
5. Double click ***Model***





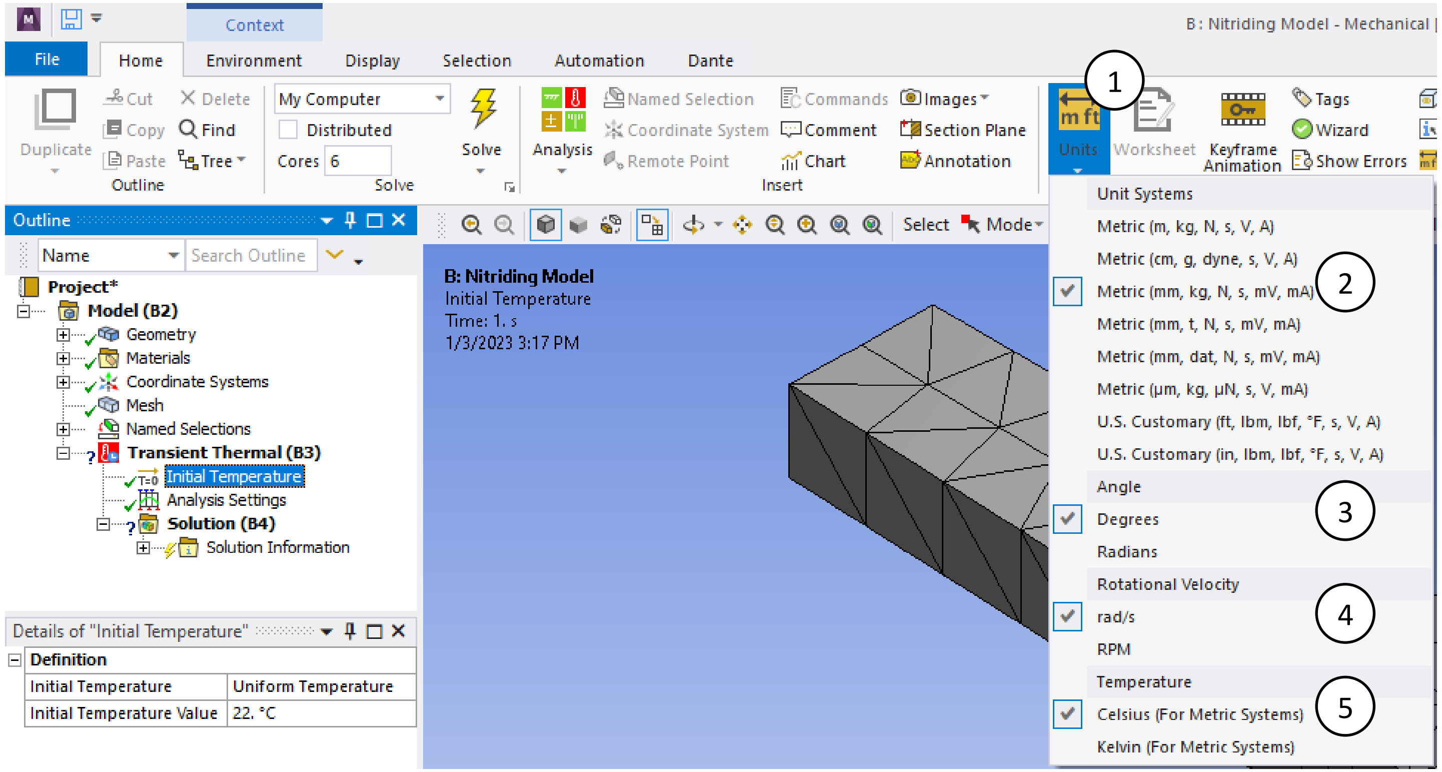




Step 2: Check Units

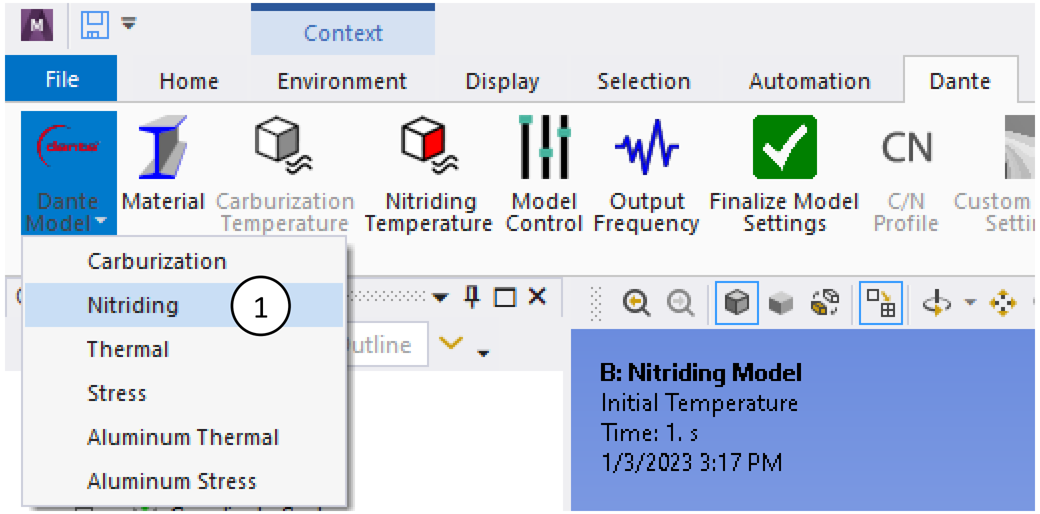
It is critical that the units be properly defined

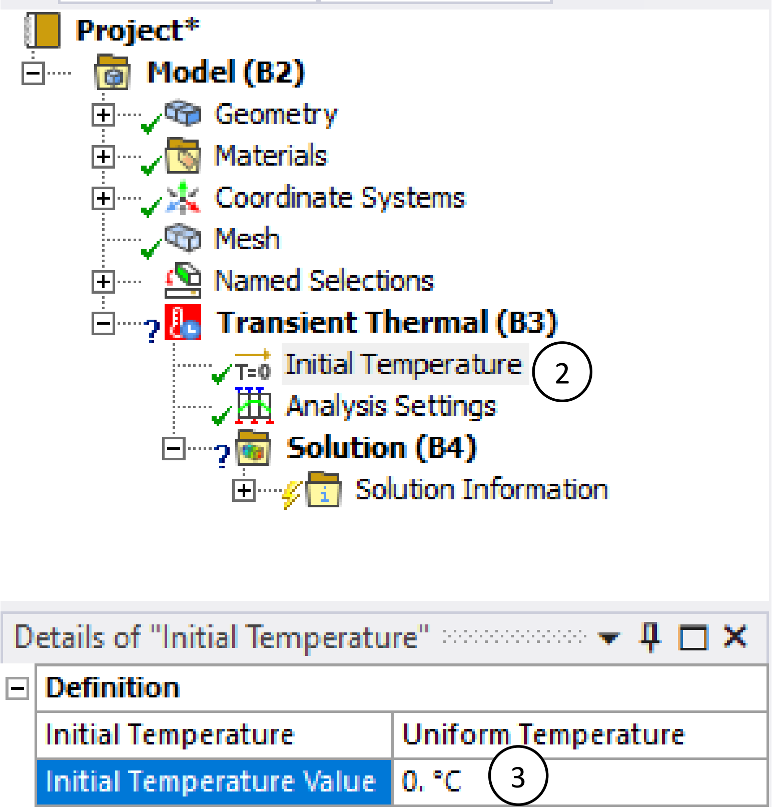
1. **Click on Units in the Home toolbar**
2. **Select Metric (mm, kg, N, s, mV, mA)**
3. **Select Degrees**
4. **Select rad/s (This isn't critical as there is no motion defined in the heat treatment models)**
5. **Select Celsius (For Metric Systems)**



Step 3: DANTE Model & Initial Temperature

1. Select ***Dante Model* → *Nitriding***
2. Click ***Initial Temperature***
3. Enter ***0*** for ***Details of "Initial Temperature"***





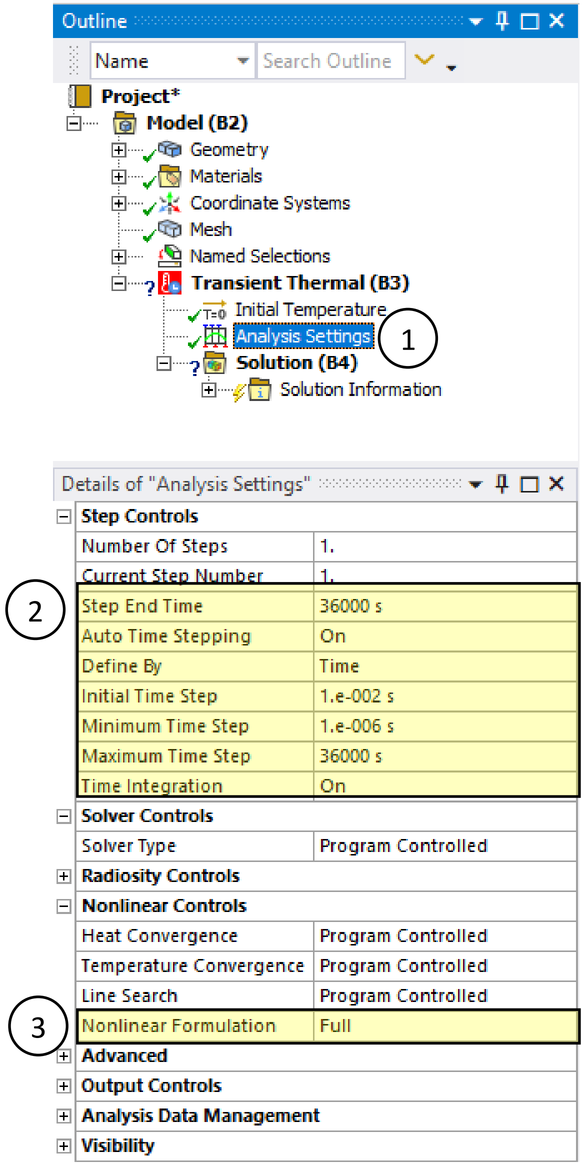
Step 4: Define Processing Steps

1. Select ***Analysis Settings***

2. Set the values under ***Details of “Analysis Settings”***:

   - ***Step End Time: 36000s***
   - ***Auto Time Stepping: On***
   - ***Minimum Time Step: 1e-6s***
   - ***Maximum Time Step: 36000s***

3. Set ***Nonlinear Formulation*** to ***Full*** under ***Nonlinear Controls***

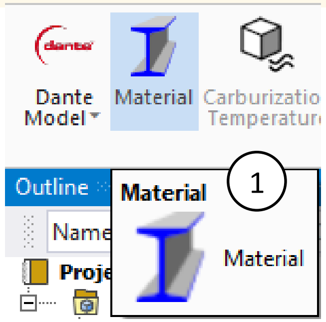


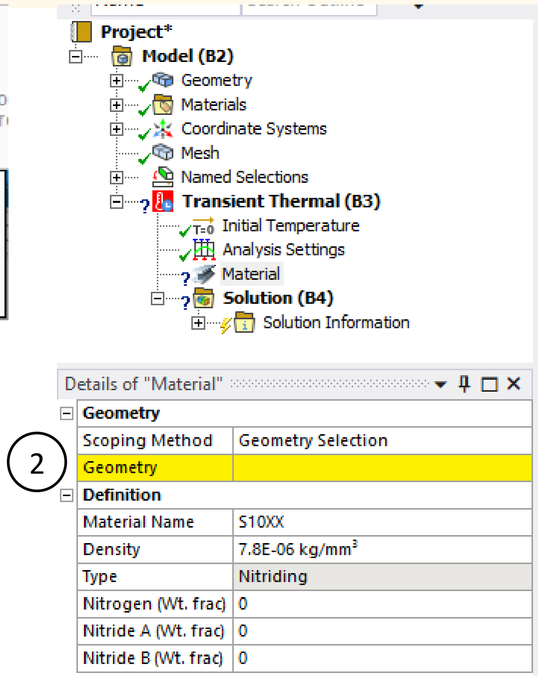
Step 5: Assign Material

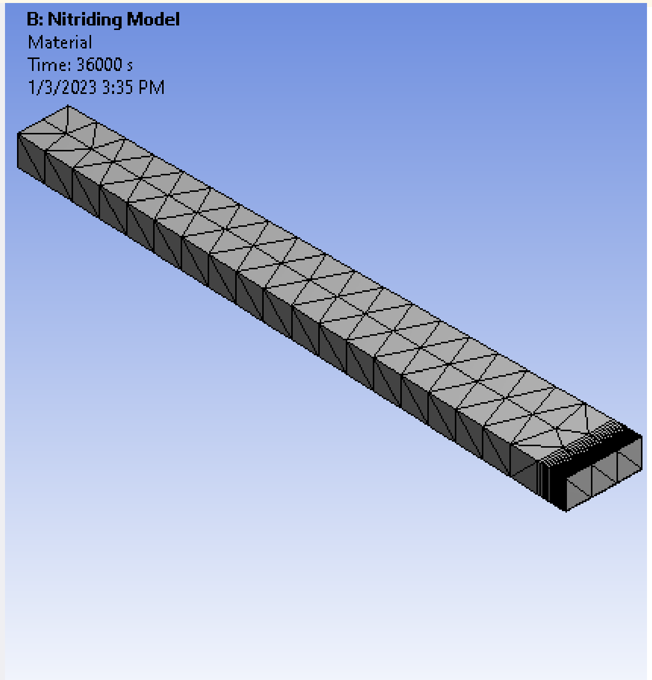
1. Select ***Material*** from the ***Dante Toolbar***

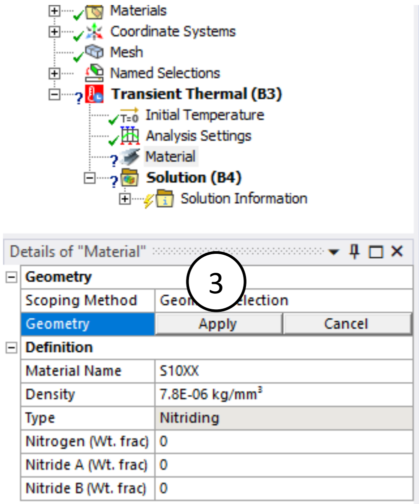
<u>Details of “Material”</u>:

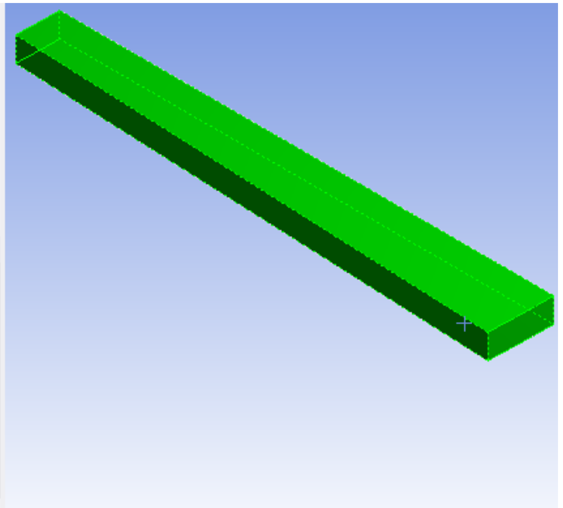
2. Click on the yellow box next to ***Geometry*** → Select the part body
3. Select ***Apply*** for ***Geometry***
4. Select ***Material Name*** → **SH13**

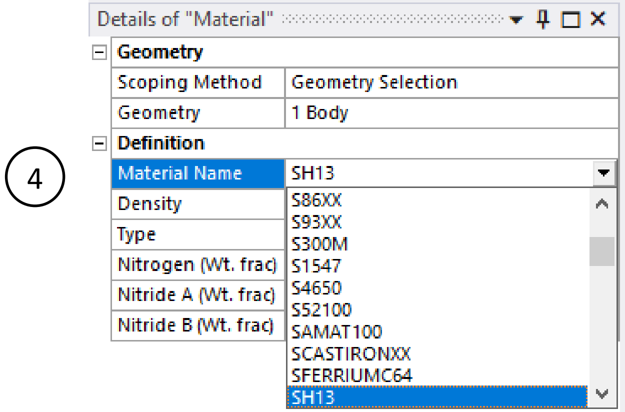


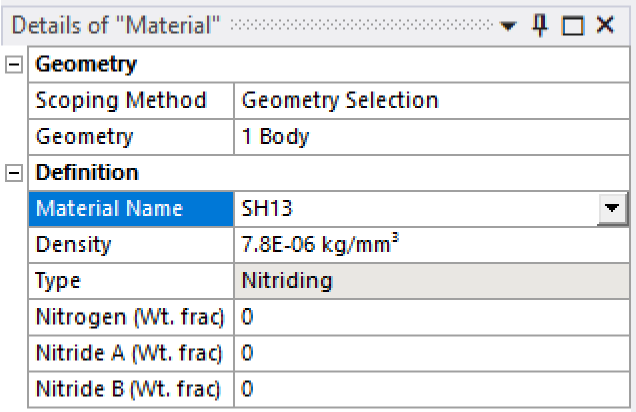












Step 6: Define Nitriding Temperature

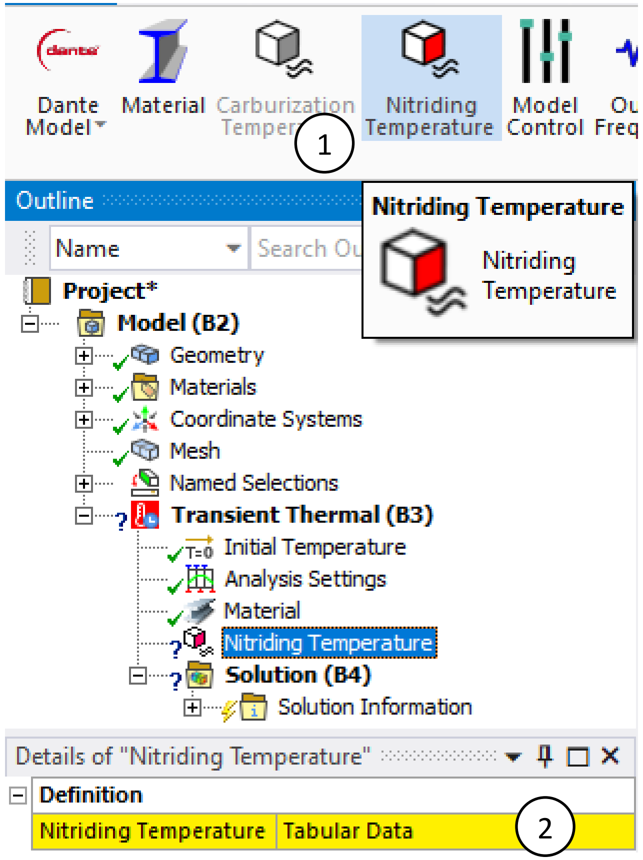
1. Select ***Nitriding Temperature*** from the ***Dante Toolbar***

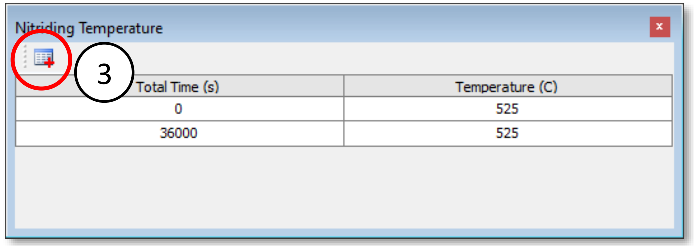
2. Click ***Tabular Data***

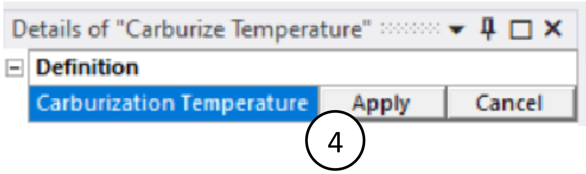
3. Add 2 rows to the table and input the time-temperature data

   - Constant 525°C for 36000s

4. Click ***Apply***

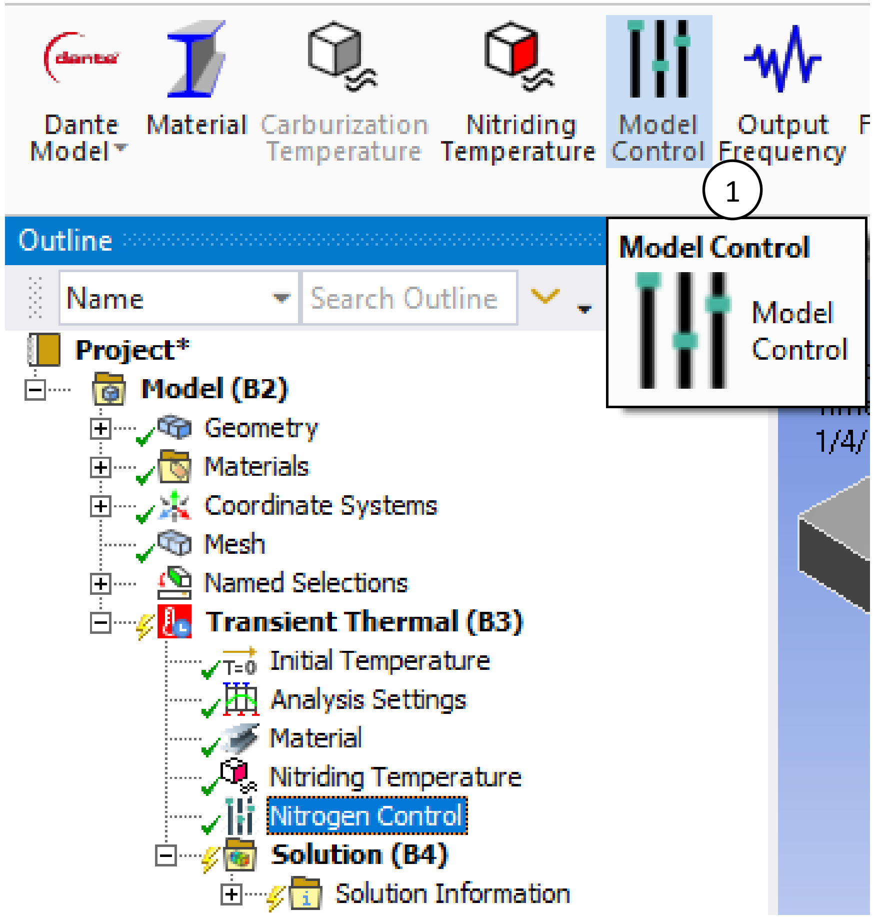






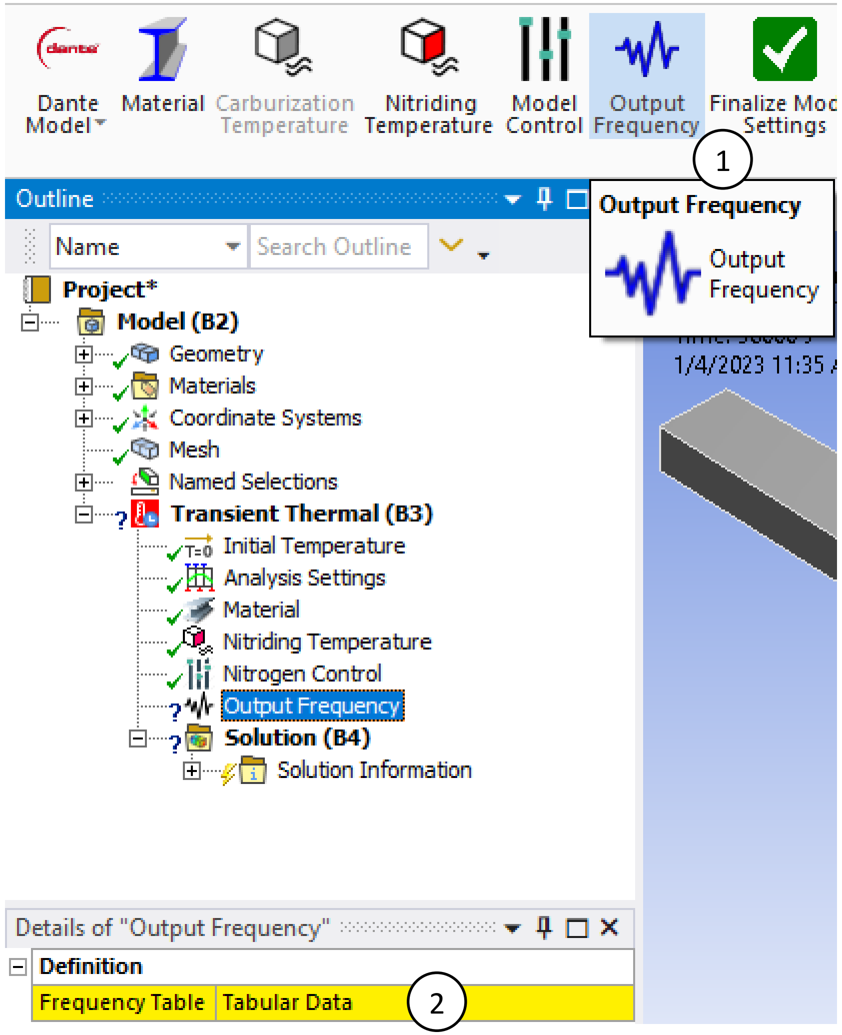
Step 7: Define Nitrogen Control

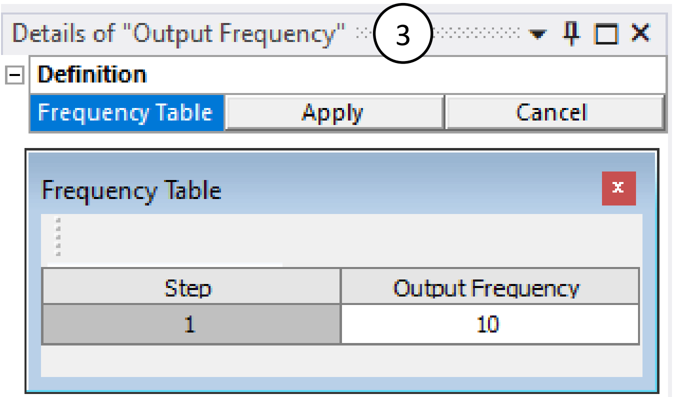
1. Select ***Model Control*** from the ***Dante toolbar***



Step 8: Define Output Frequency

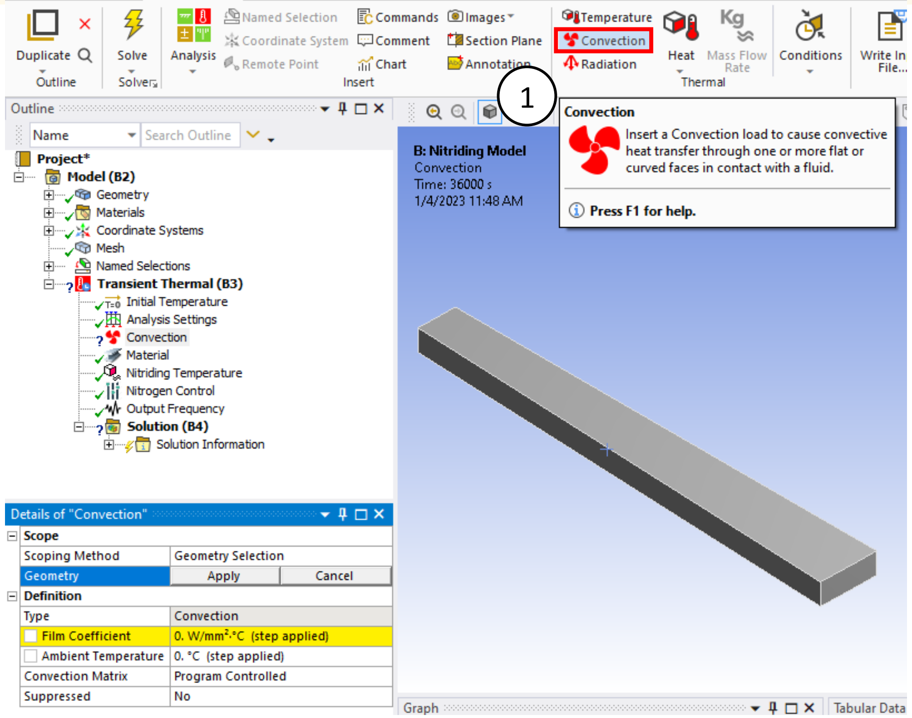
1. Select ***Output Frequency*** from the ***Dante toolbar***
2. Click ***Tabular Data***
3. Keep the default values → click ***Apply***

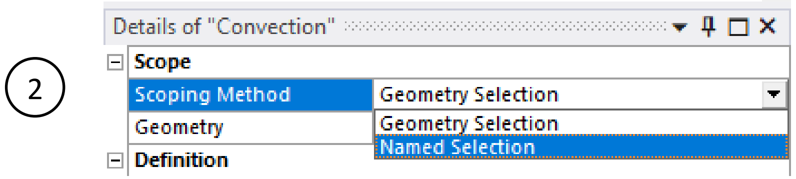


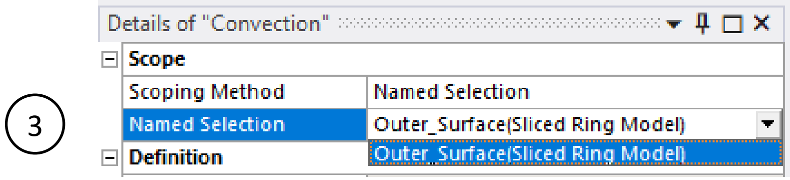


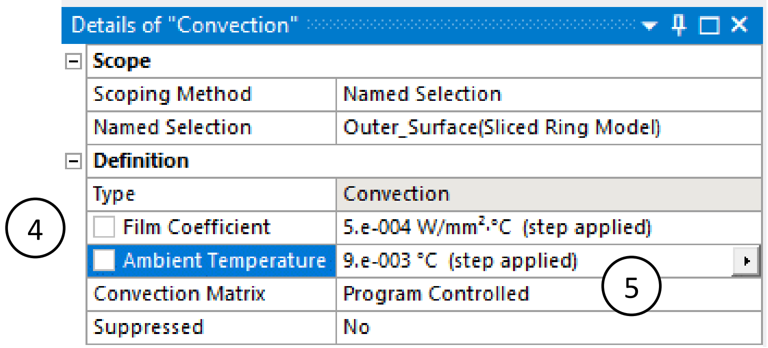
Step 9: Define Nitriding Boundary Conditions

1. Select ***Convection*** from ***Environment*** toolbar
2. Select ***Scoping Method → Named Selection***
3. Select ***Named Selection → Outer_Surface***
4. Enter ***5e-4 (W/mm^{2}C)*** for ***Film Coefficient***
5. Enter ***9e-3 (N Wt. Fraction)*** for ***Ambient Temperature***



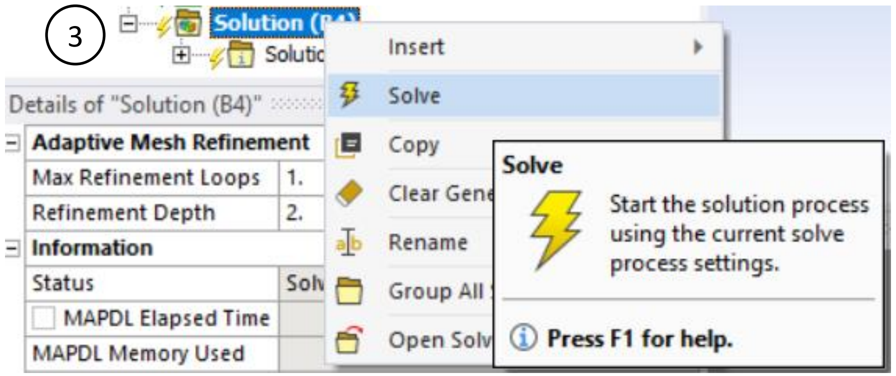


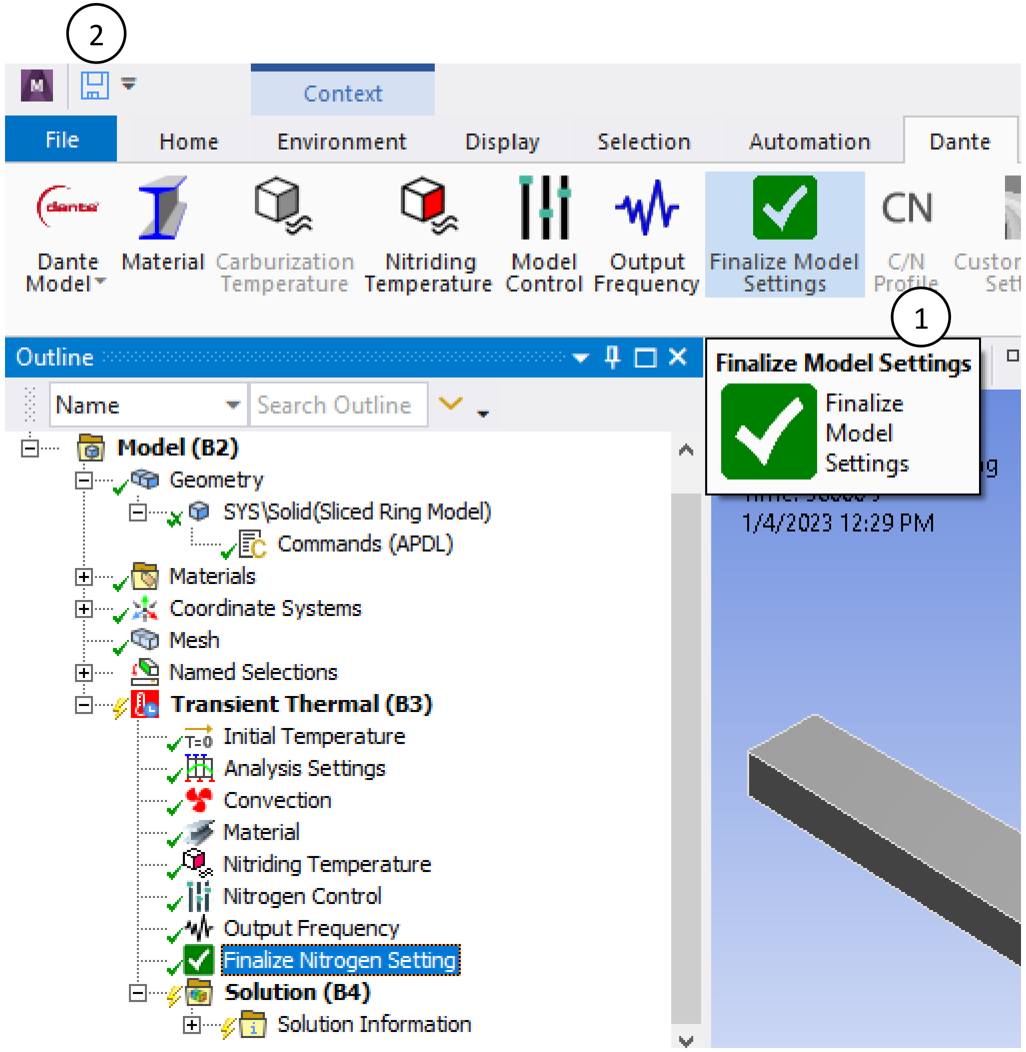




Step 10: Running the Nitriding Model

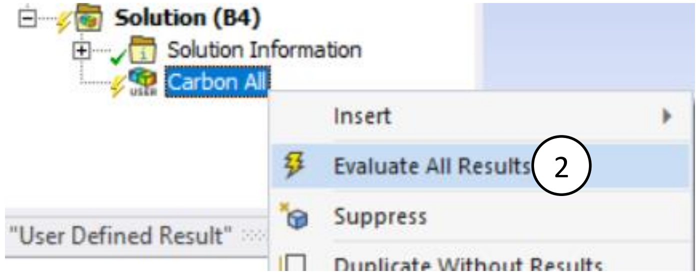
1. Select ***Finalize Model Settings*** from the ***Dante Toolbar***
2. Save the model
3. Right click ***Solution*** → ***Solve***

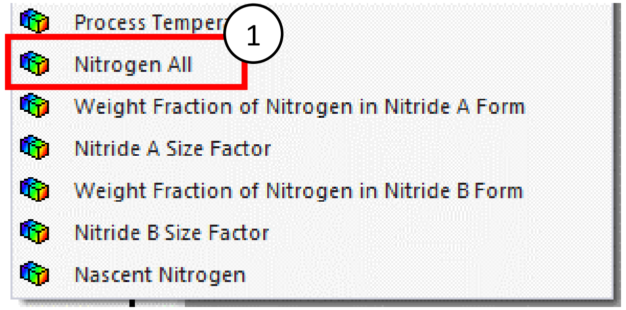


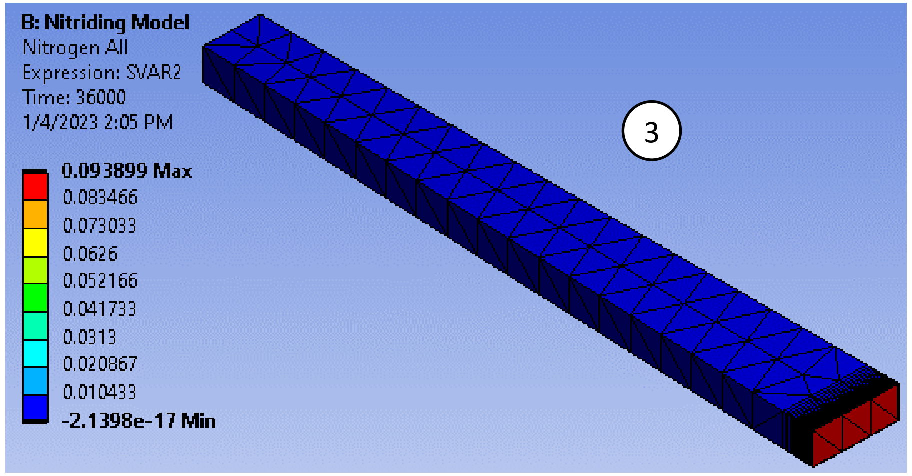


Step 11: Nitrogen Model Post-Processing

1. Under ***Dante Results*** select ***Nitrogen All***
2. Right click ***Nitrogen All → Evaluate All Results***
3. Check the result



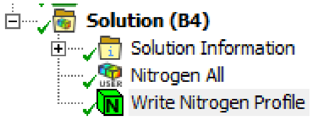


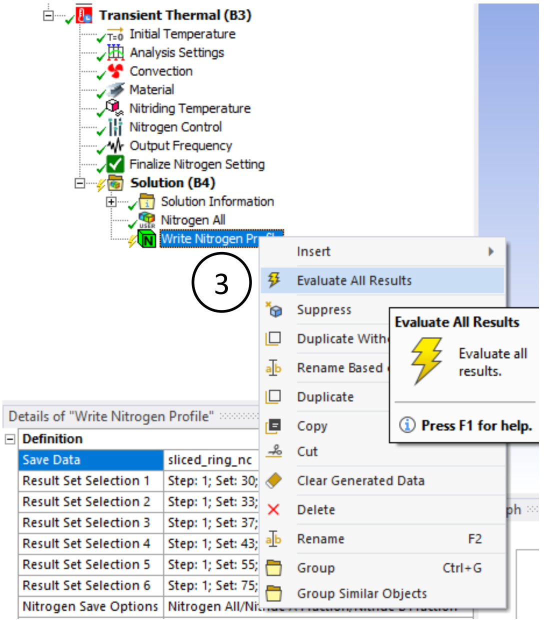


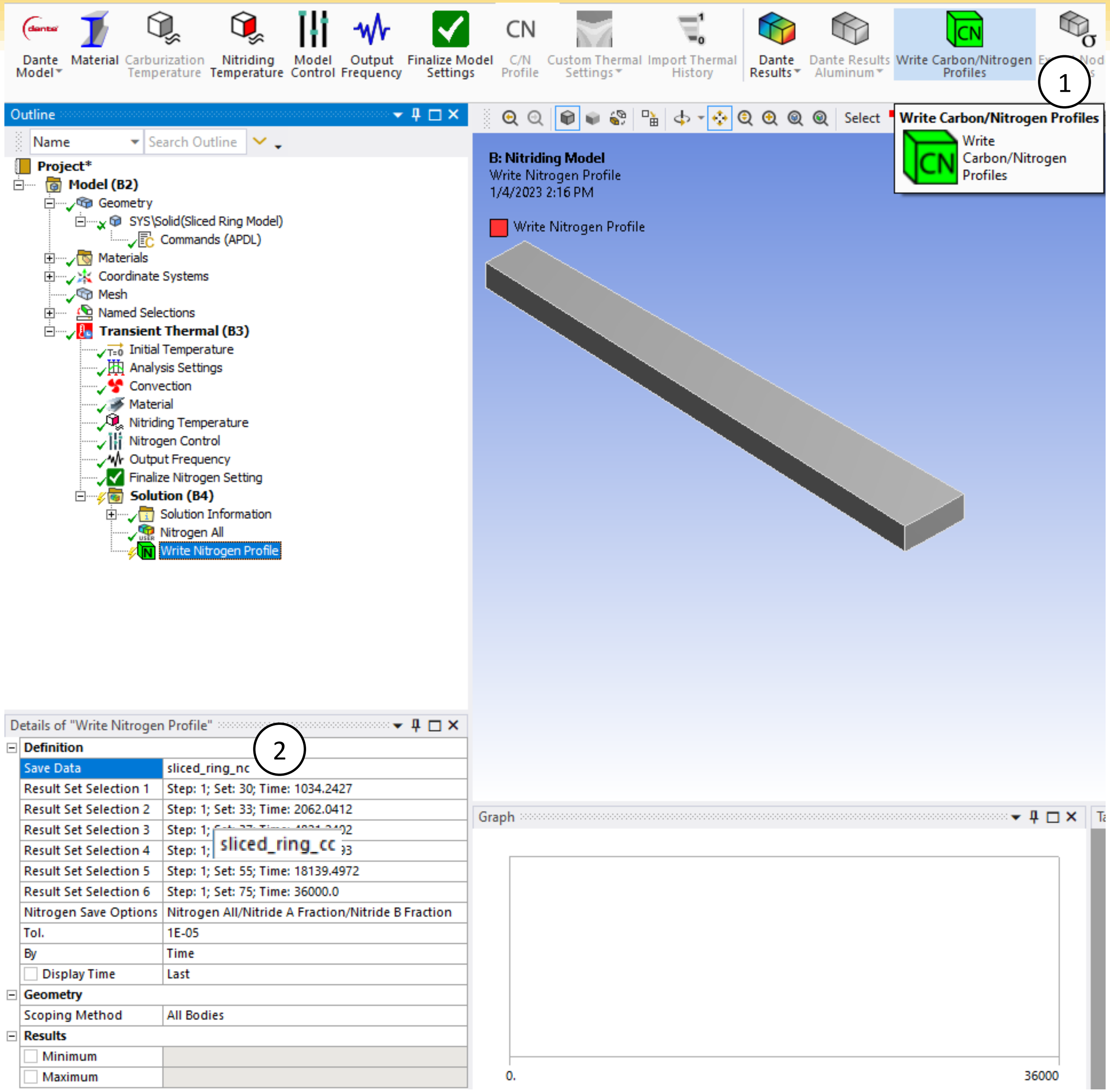
NOTE: These DANTE results can be added before running the model, when solving the model, they will be evaluated along with it

Step 12: Exporting Nitrogen Profile

1. Select ***Write Nitrogen Profile*** from the ***Dante Toolbar***
2. Under ***Details of "Write Nitrogen Profile"***, change the file name to **"sliced_ring_nc"** for ***Save Data***
3. Right click ***Write Nitrogen Profile*** **→** ***Evaluate All Results***



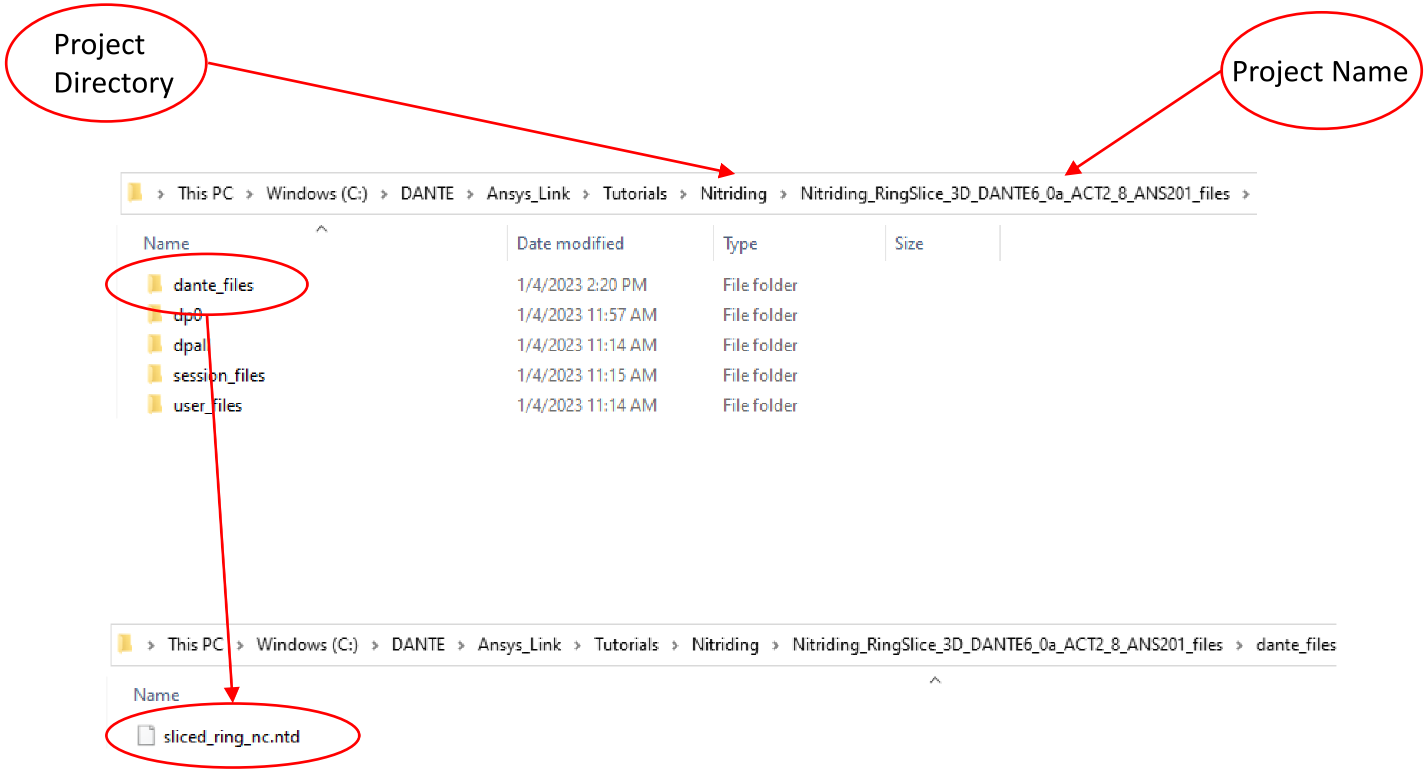




Step 13: DANTE Carbon Profile Directory

Saved nitrogen file in Project directory (the directory in which you saved your current Project) with a folder name as “dante_files” as a .ntg file

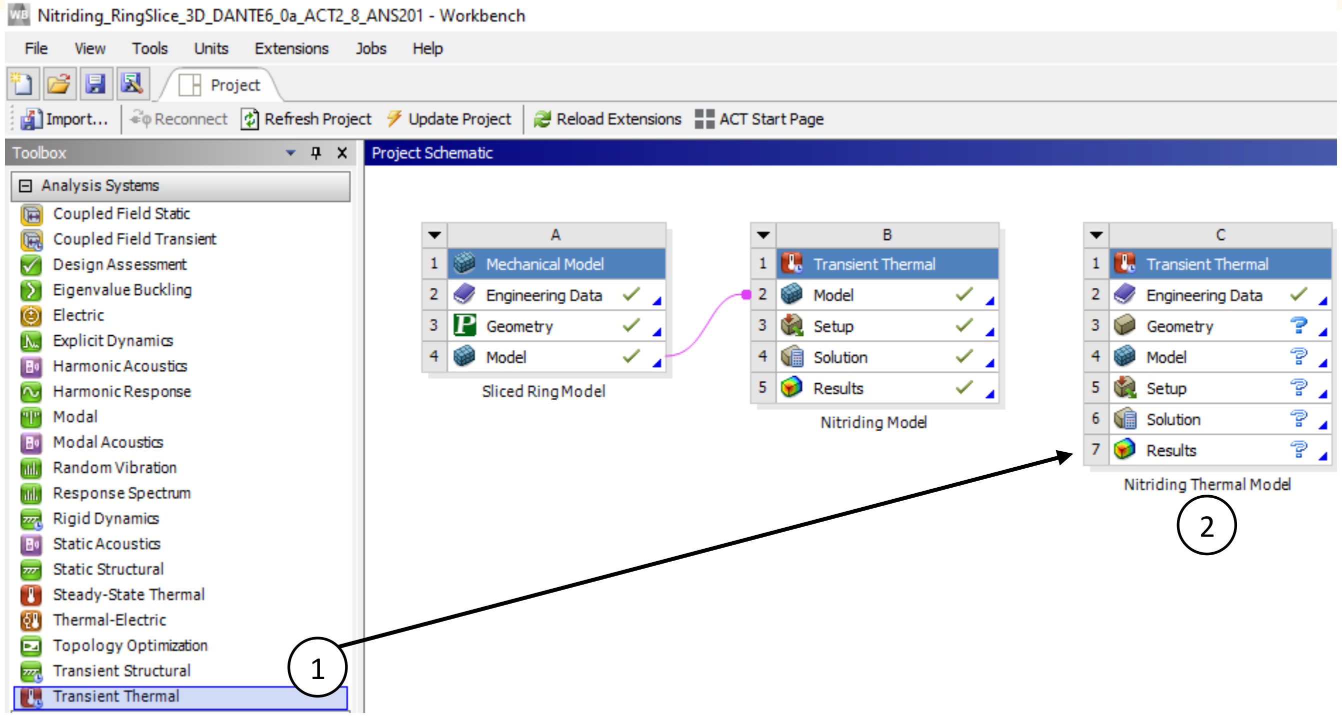
Save the Project and close Ansys Mechanical

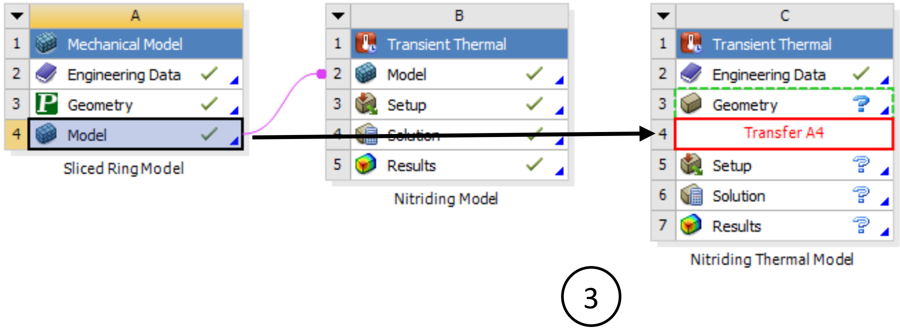


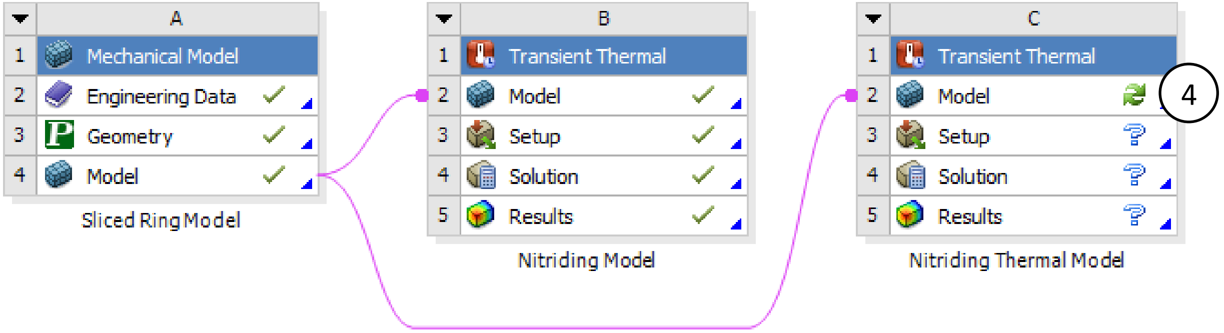
Nitriding Thermal Model

Step 1: Thermal Model Analysis

1. Drag and drop a ***Transient Thermal Analysis System*** into the ***Project Schematic***
2. Rename it as “***Nitriding Thermal Model***”
3. Drag and drop ***Model*** from ***Sliced Ring Model*** to ***Nitriding Thermal Model***
4. Double click ***Model*** in the ***Nitriding Thermal Model***



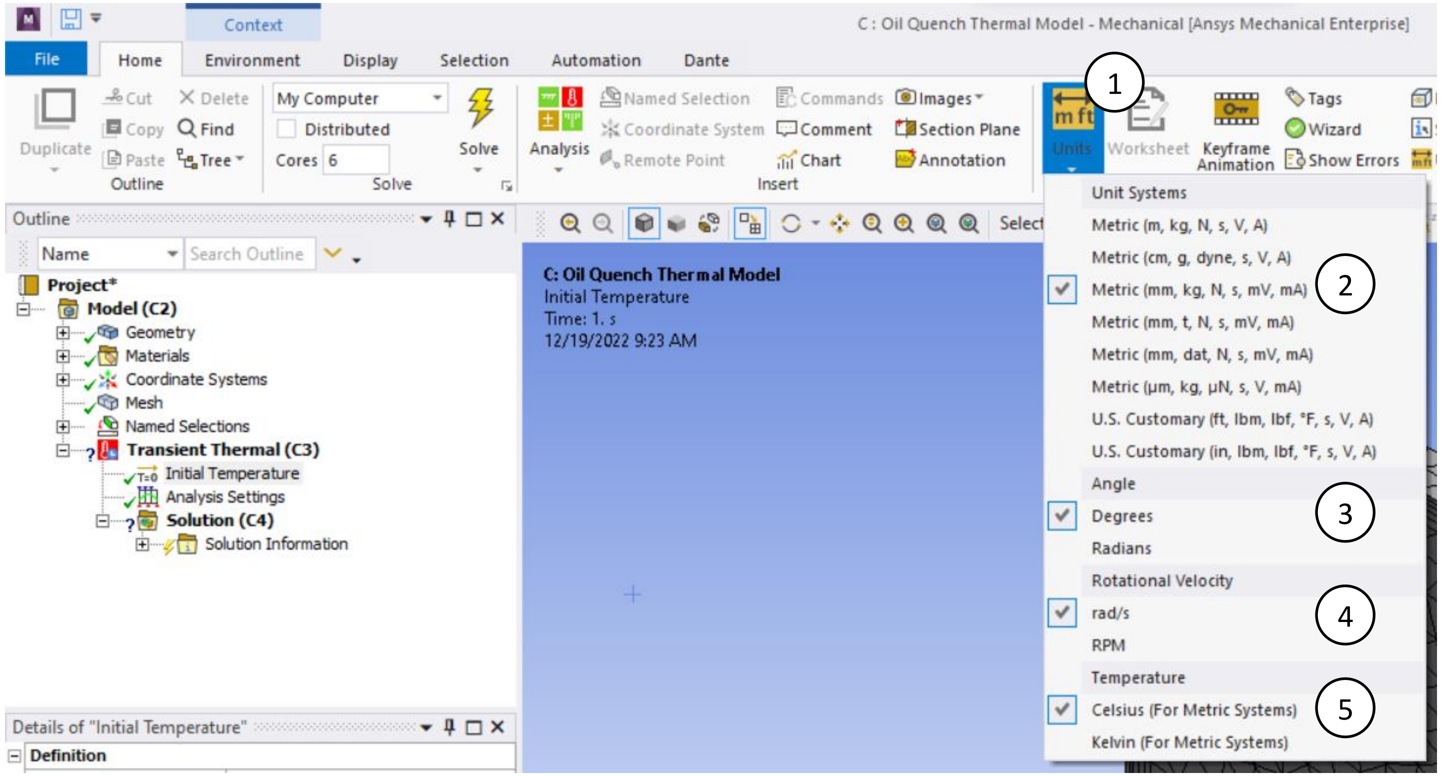




Step 2: Check Units

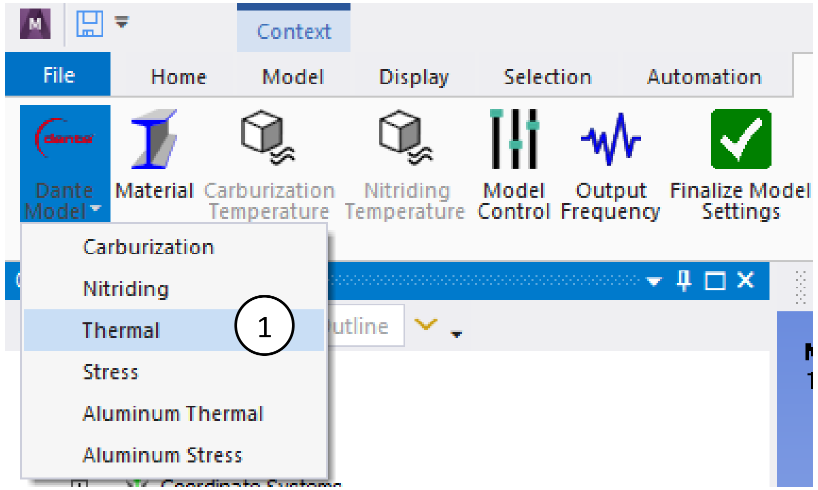
It is critical that the units be properly defined

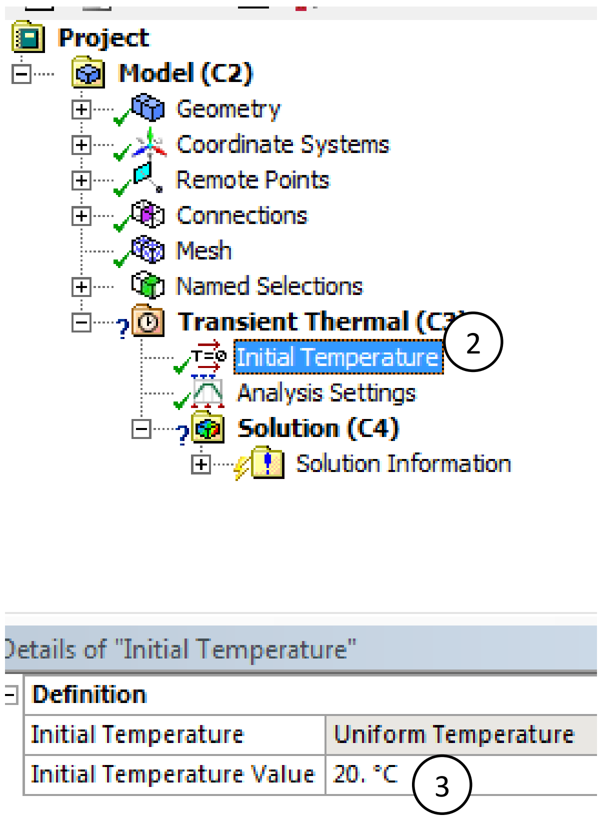
1. **Click on Units in the Home toolbar**
2. **Select Metric (mm, kg, N, s, mV, mA)**
3. **Select Degrees**
4. **Select rad/s (This isn't critical as there is no motion defined in the heat treatment models)**
5. **Select Celsius (For Metric Systems)**



Step 3: Define DANTE Model & Initial Temperature

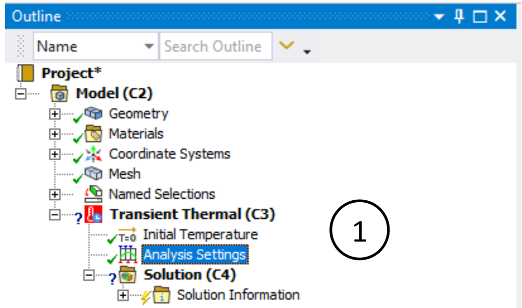
1. Click on ***Dante Model*** in the ***Dante Toolbar*** and select ***Thermal***
2. Select **Initial Temperature** in the ***Project Tree***
3. Change the ***Initial Temperature Value*** to 20.0 °C

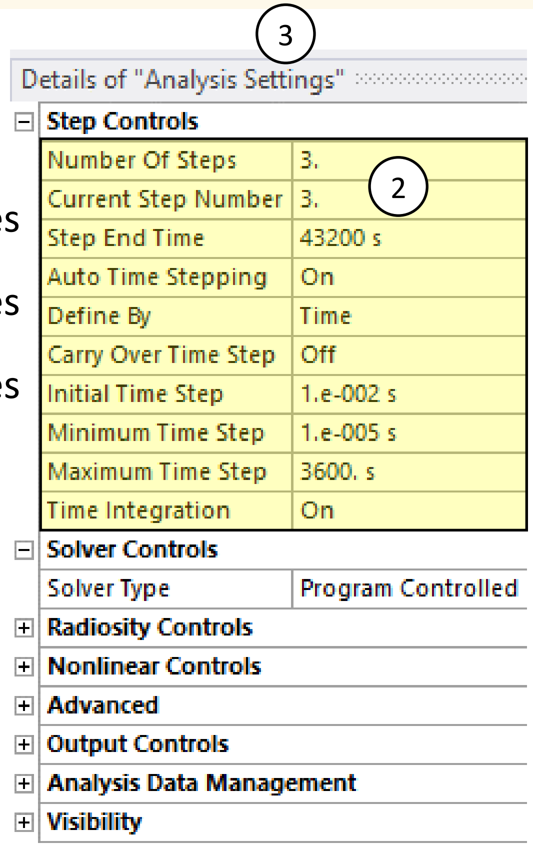


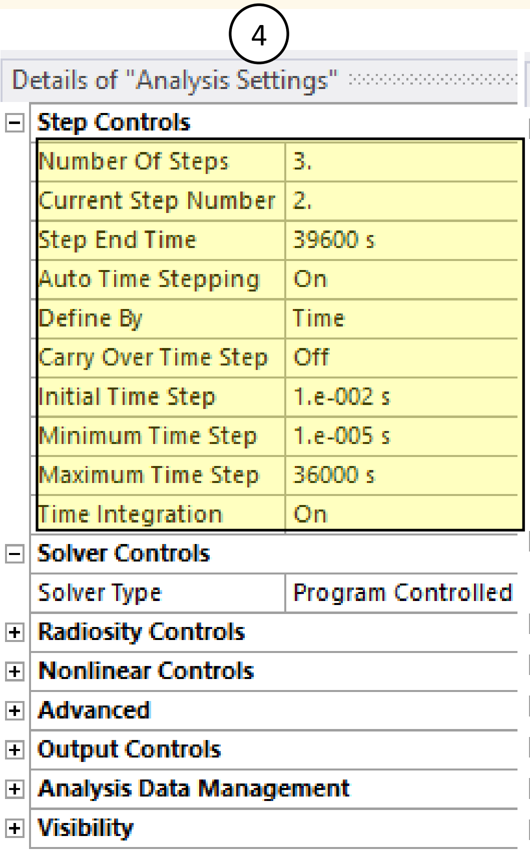


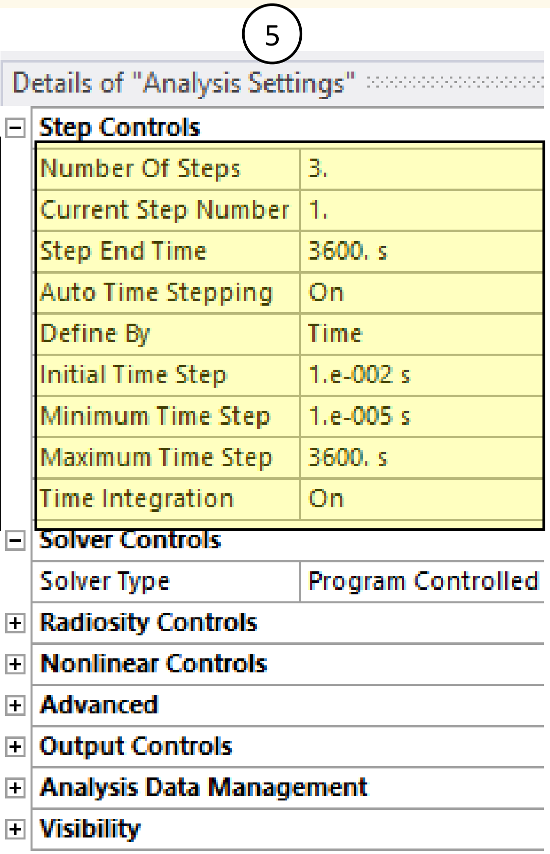
Step 4: Define Processing Steps

1. Select **Analysis Settings**
2. Enter ***3*** for ***Number of Steps***
3. Set ***Current Step Number*** to ***3*** and enter values
4. Set ***Current Step Number*** to ***2*** and enter values
5. Set ***Current Step Number*** to ***1*** and enter values
6. Set ***Nonlinear Controls → Nonlinear Formulation → Full***



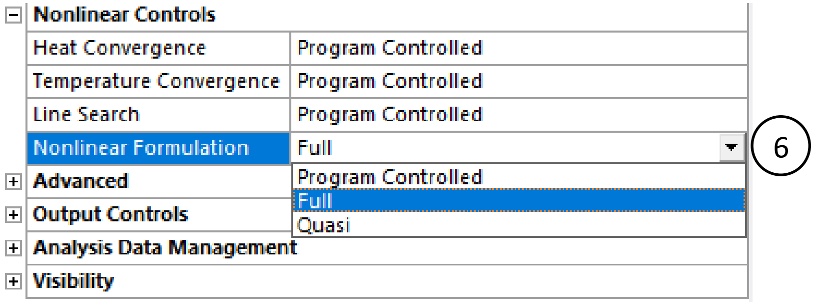






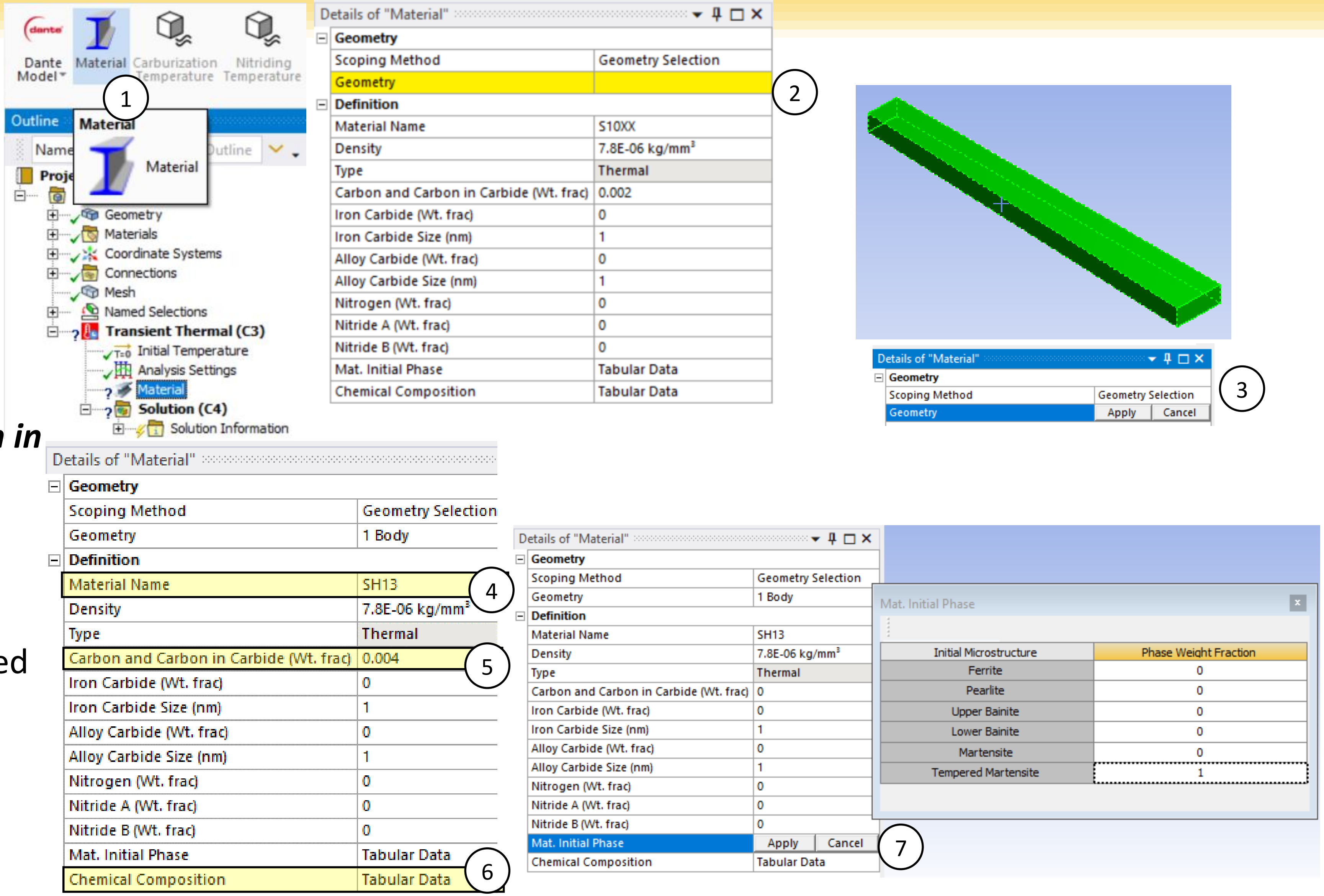
Thermal modeling of the nitriding process involves modeling of 3 steps: the part heat up, processing, and cooling down

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
3	43200	On	Time	Off	1.00E-02	1.00E-05	3600	On
2	39600						36000	
1	3600			N/A			3600	



Step 5: Assign Material

1. Select ***Material*** from the ***Dante Toolbar***
2. Click on the yellow box next to ***Geometry*** → Select the part body
3. Select ***Apply*** for ***Geometry***
4. Select ***SH13*** for ***Material Name***
5. Enter ***0.004*** for ***Carbon and Carbon in Carbide***
6. Click ***Tabular Data*** for ***Mat. Initial Phase*** and set values as shown
   - Initial microstructure composed fully of tempered martensite, set to ***1*** with other phases ***0***
7. Click ***Apply***



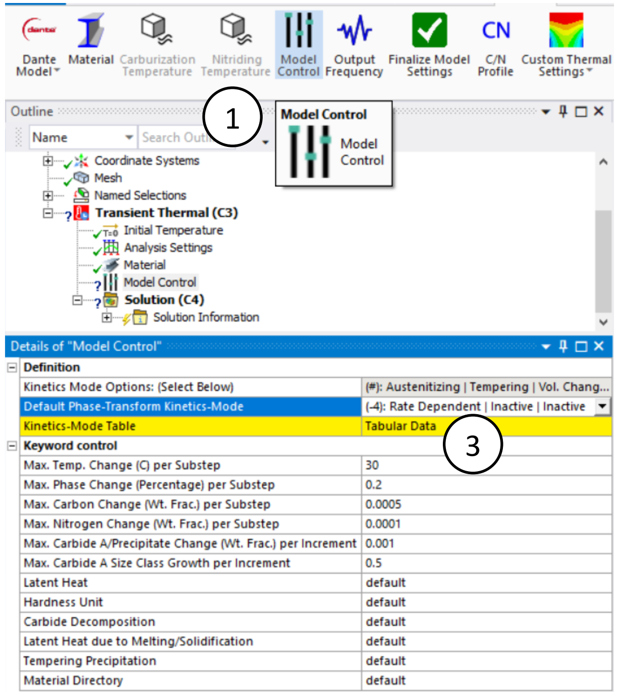
Step 6: Define Control File

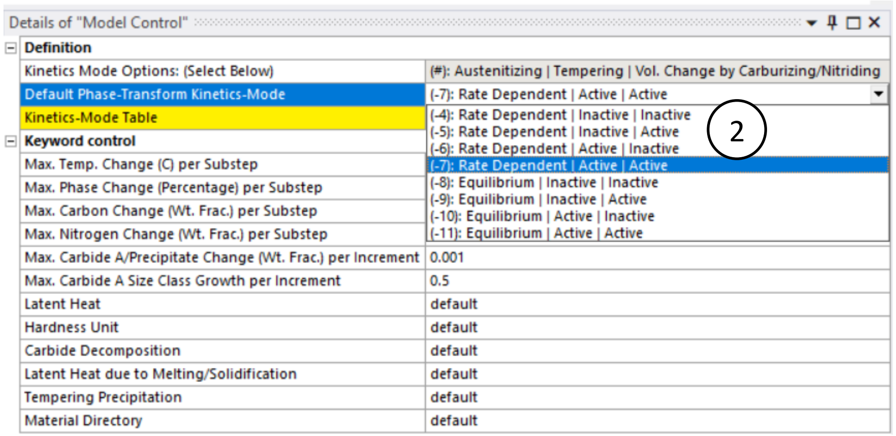
1. Select ***Model Control*** in the ***Dante toolbar***

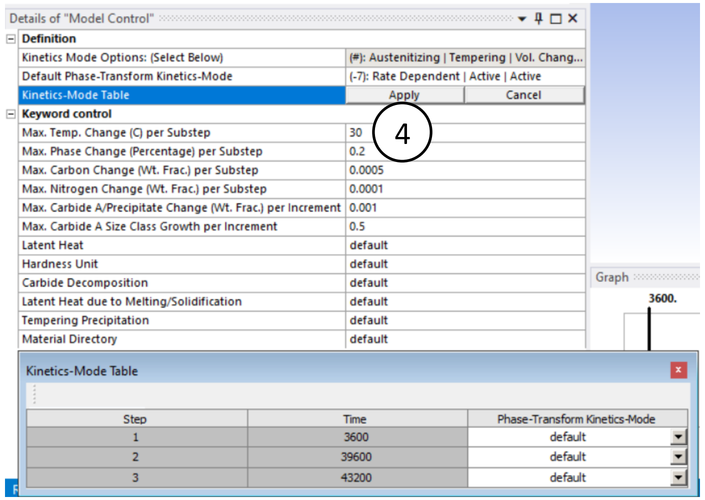
2. Select ***Default Phase-Transform Kinetics-Mode → (-7): Rate Dependent|Active|Active***

3. Click ***Tabular Data → Kinetics-Mode Table***

4. Click ***Apply***

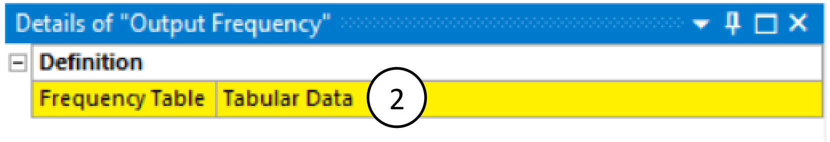


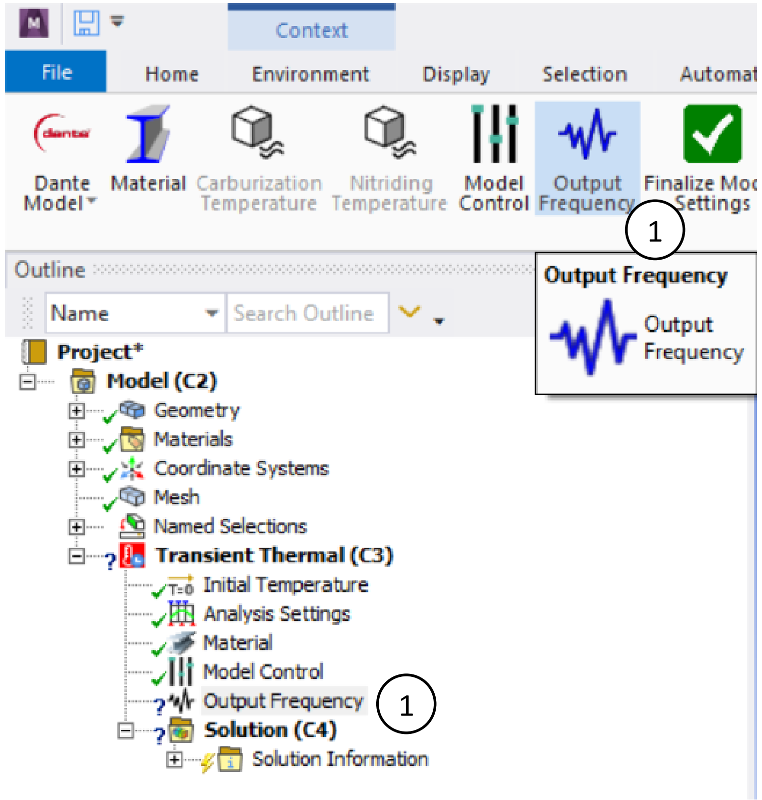


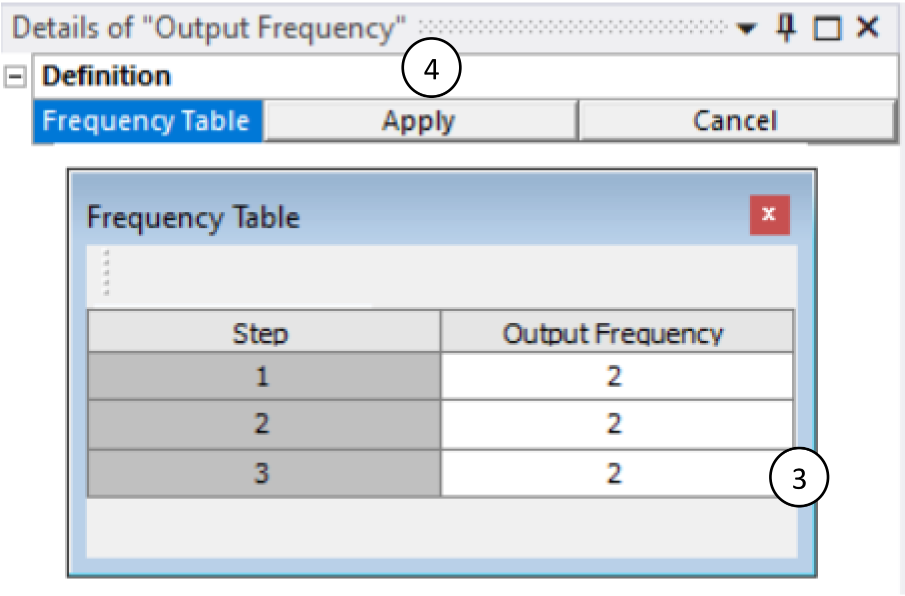


Step 7: Define Output Frequency

1. Select ***Output Frequency*** from the ***Dante toolbar***
2. In ***Details of "Output Frequency"***, click on ***Tabular Data***
3. Leave the default frequency values as 2
4. Click ***Apply***

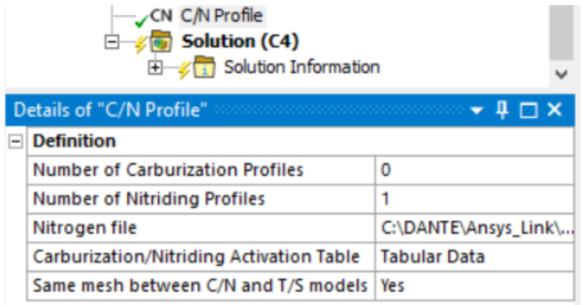


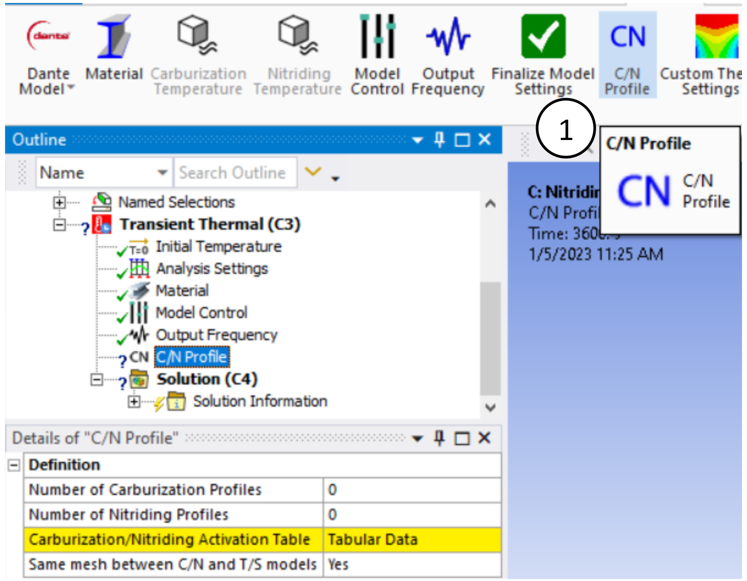


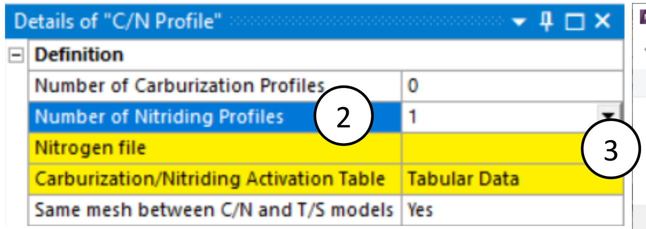


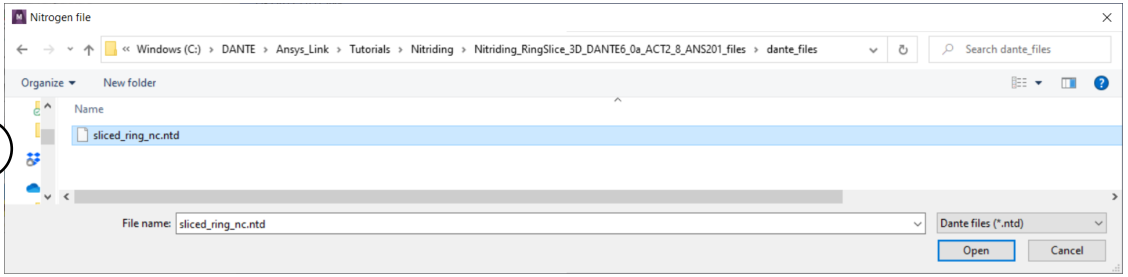
Step 8: Importing Nitrogen Profiles

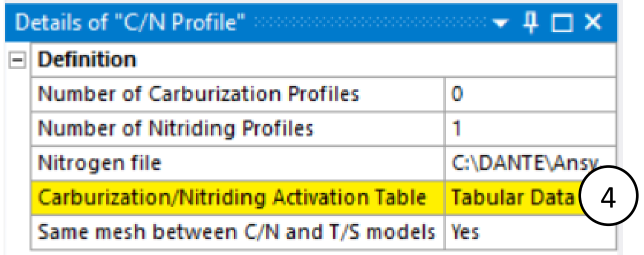
1. Select ***C/N Profile*** from the ***DANTE Toolbar***
2. Set ***Number of Nitriding Profiles → 1***
3. Click ***Nitriding file***, navigate to ***dante_files***, select the nitrogen profile and click ***Open***
4. Under ***Carburization/Nitriding Activation Table***, click ***Tabular Data***
5. Set ***Nitriding → 1*** at ***Step 2*** to activate it and click ***Apply***

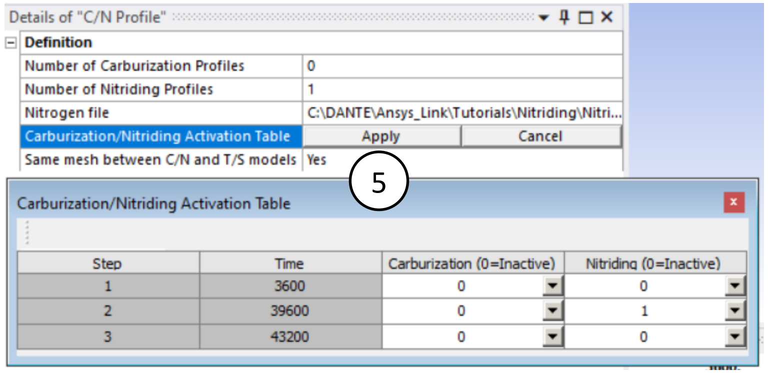






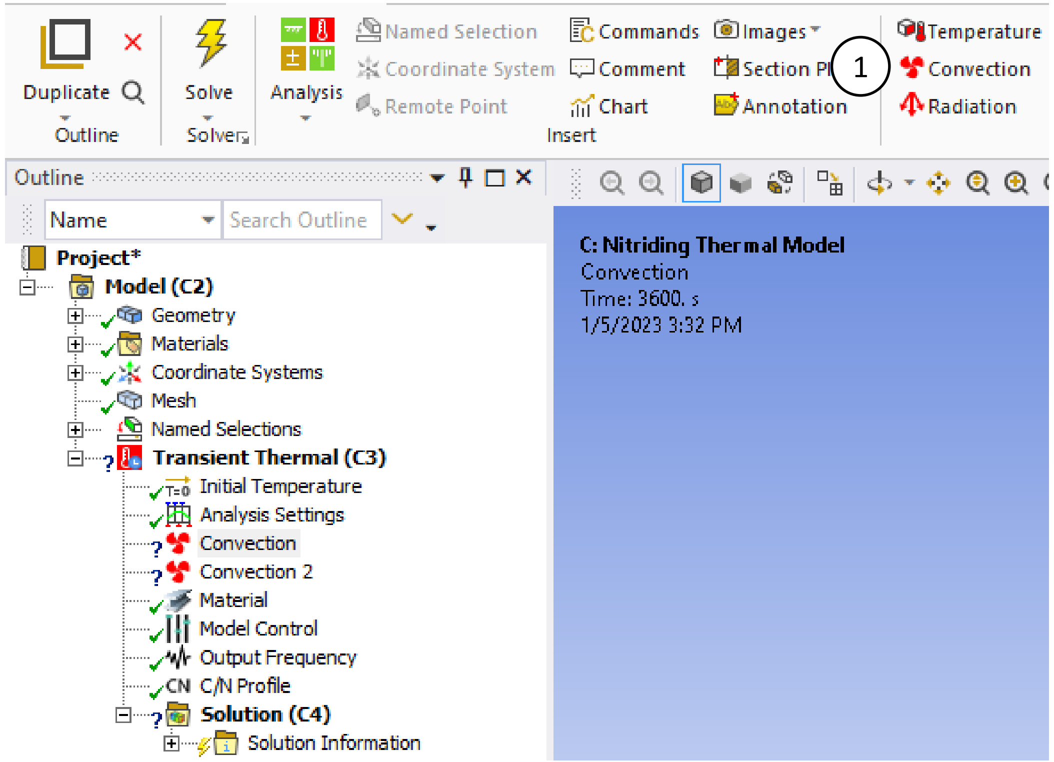


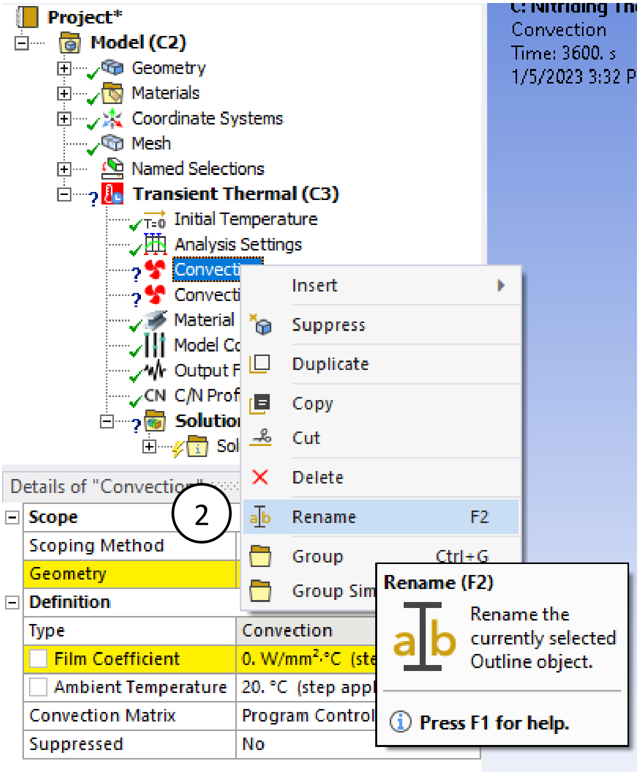


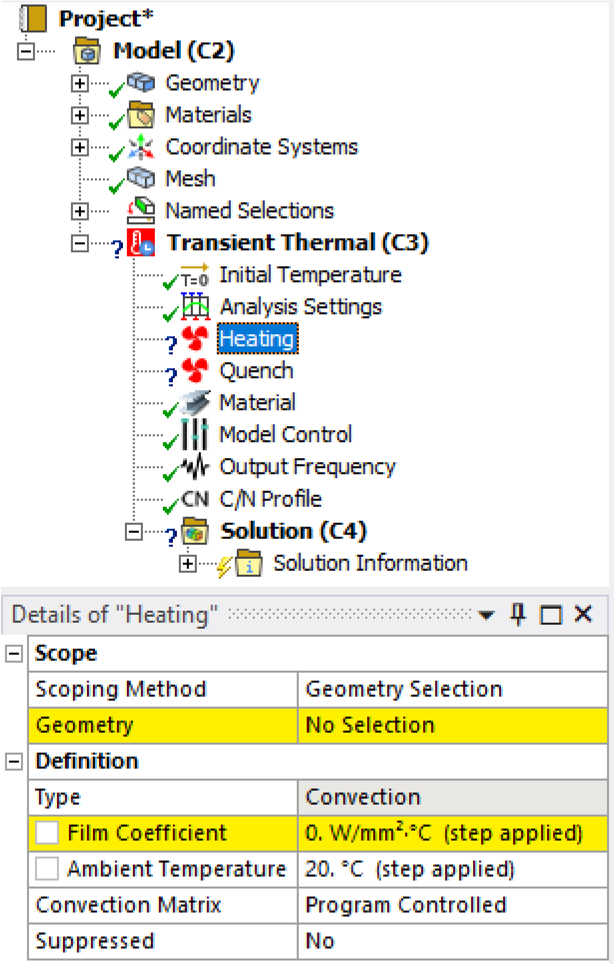


Step 9a: Defined Thermal Boundary Conditions

1. Add 2 ***Convections*** from the ***Environment toolbar*** to the ***Project Tree***
2. Right-click on the ***Convections*** and rename to ***Heating, Quench***

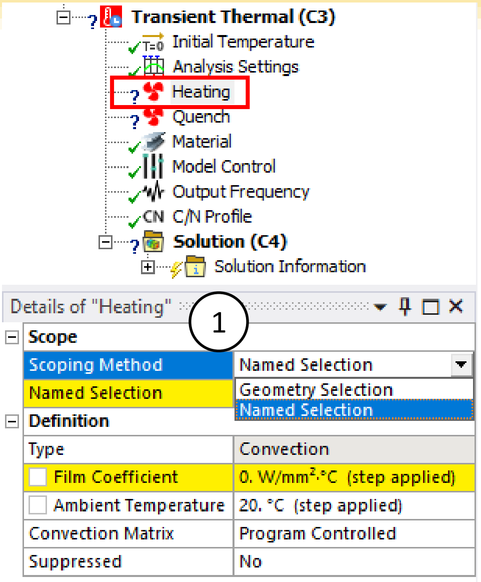


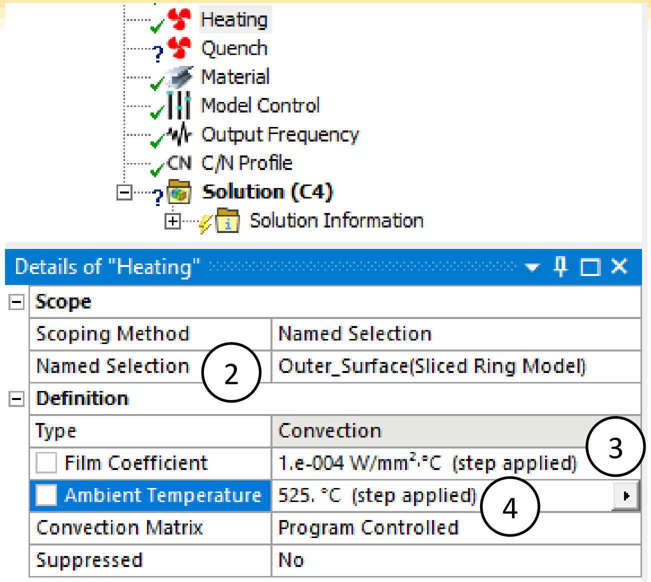


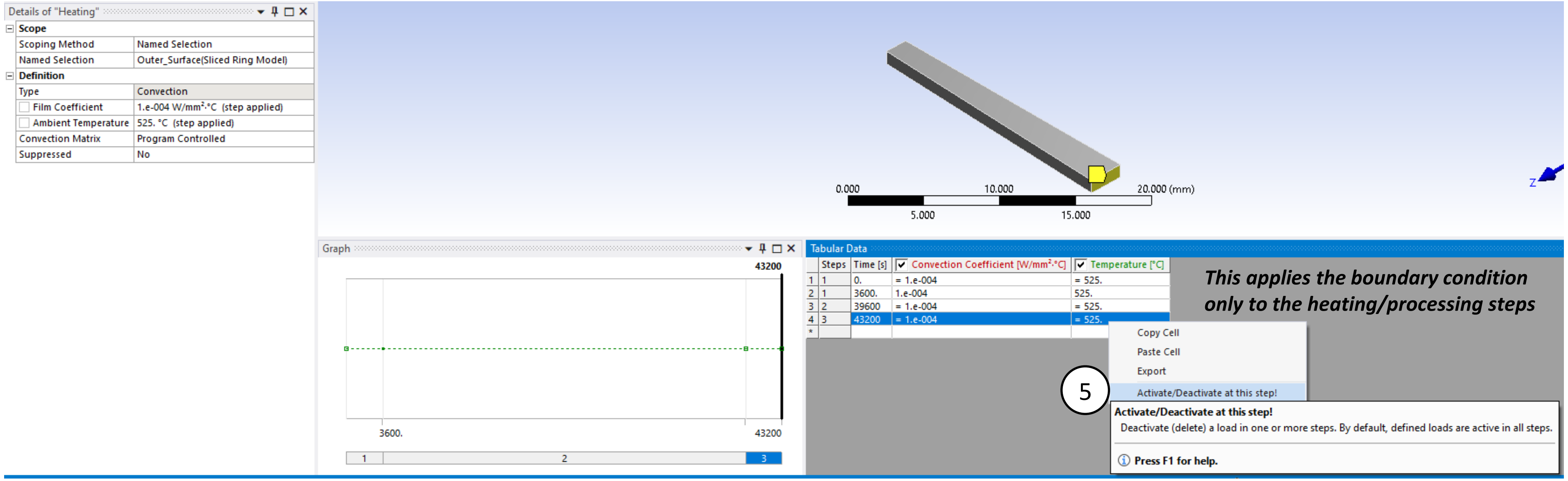


Step 9b: Heating Thermal Boundary Condition

1. Select ***Heating*** and set ***Scoping Method*** to ***Named Selection***
2. Set ***Named Selection → Outer_Surface***
3. Set ***Film Coefficient*** to ***1e-4 (W/mm²C)***
4. Set ***Ambient Temperature*** to ***525ºC***
5. Under ***Details of "Heating" → Tabular Data,*** right-click on row 4 and select ***Activate/Deactivate at this step***



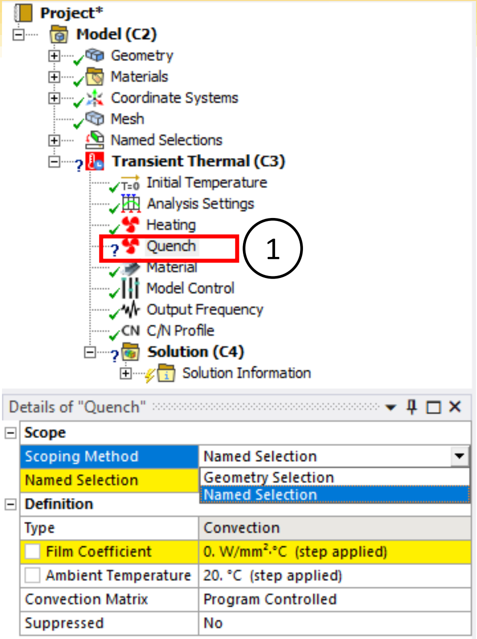


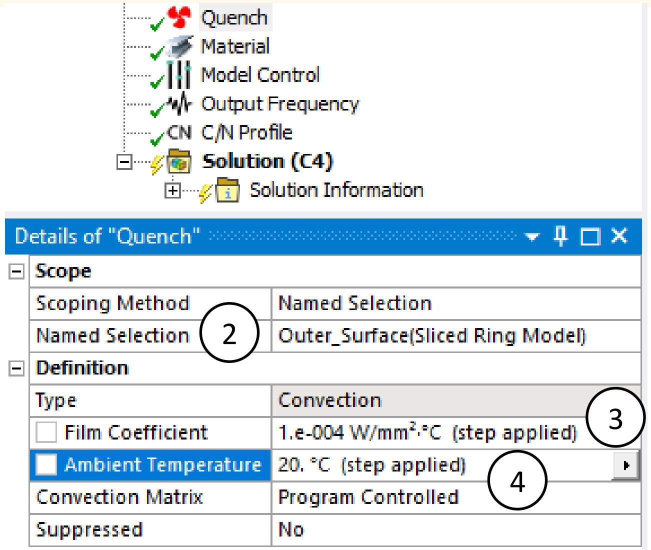


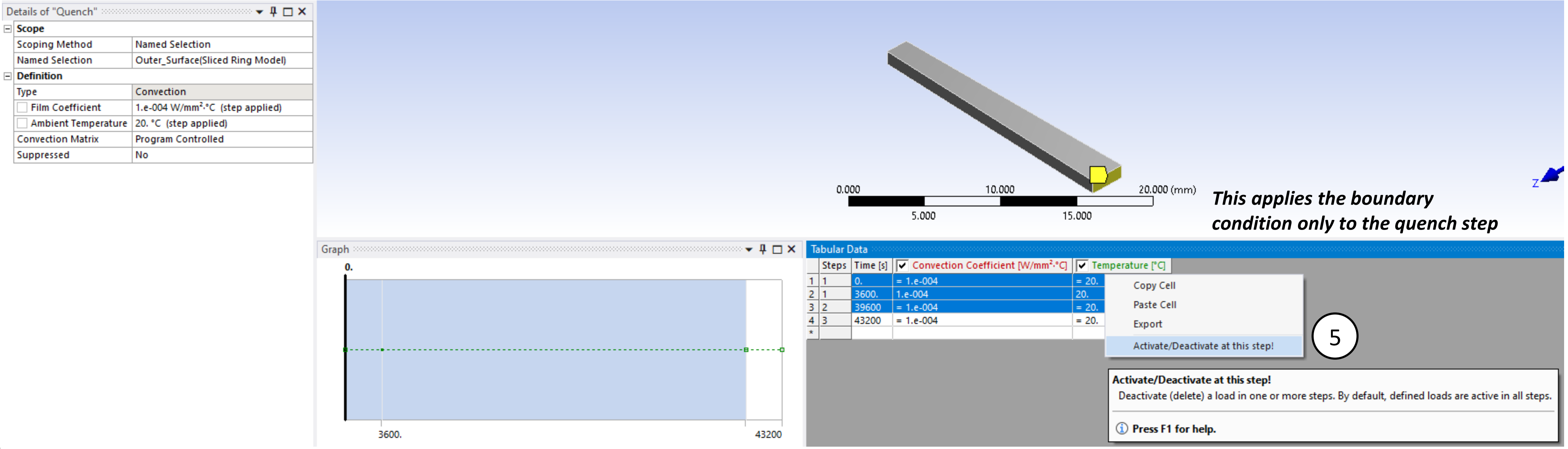
	Steps	Time [s]	Convection Coefficient [W/mm².°C]	Temperature [°C]
1	1	0.	= 1.e-004	= 525.
2	1	3600.	1.e-004	525.
3	2	39600	= 1.e-004	= 525.
4	3	43200	= 1.e-004	= 525.

Step 10: Quench Thermal Boundary Condition

1. Select ***Quench*** and set ***Scoping Method*** to ***Named Selection***
2. Set ***Named Selection → Outer_Surface***
3. Set ***Film Coefficient*** to ***1e-4 (W/mm²C)***
4. Set ***Ambient Temperature*** to ***20°C***
5. Under ***Details of "Quench" → Tabular Data,*** right-click and highlight all rows except row 4 and select ***Activate/Deactivate at this step***



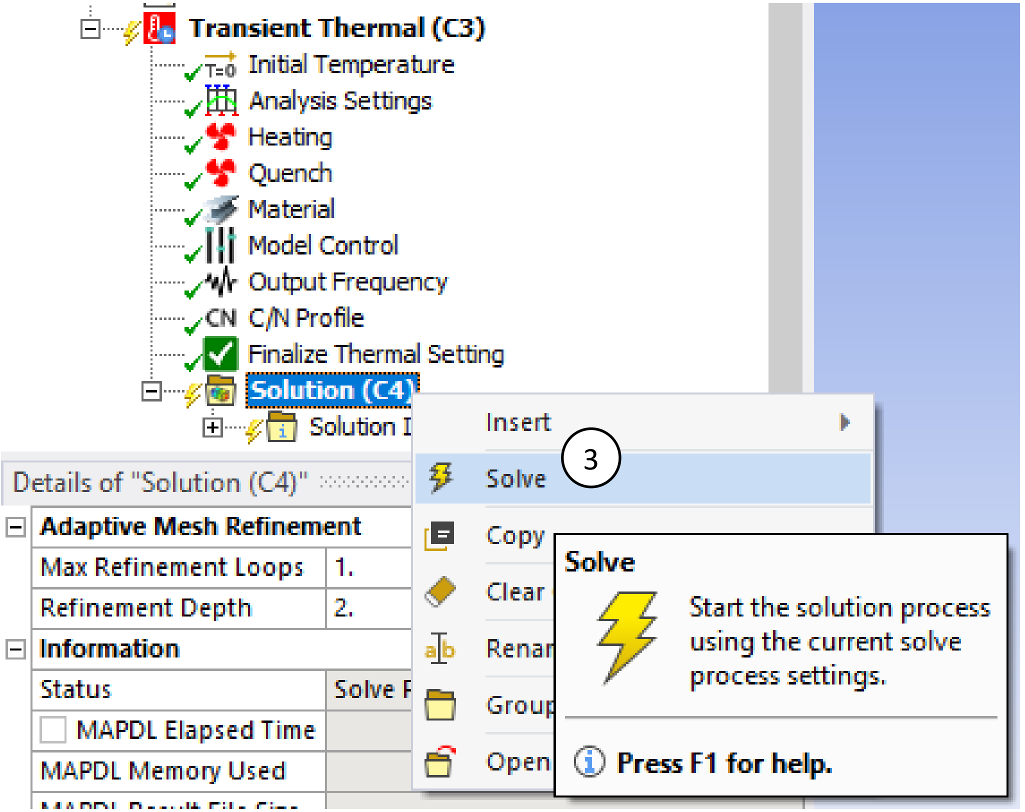


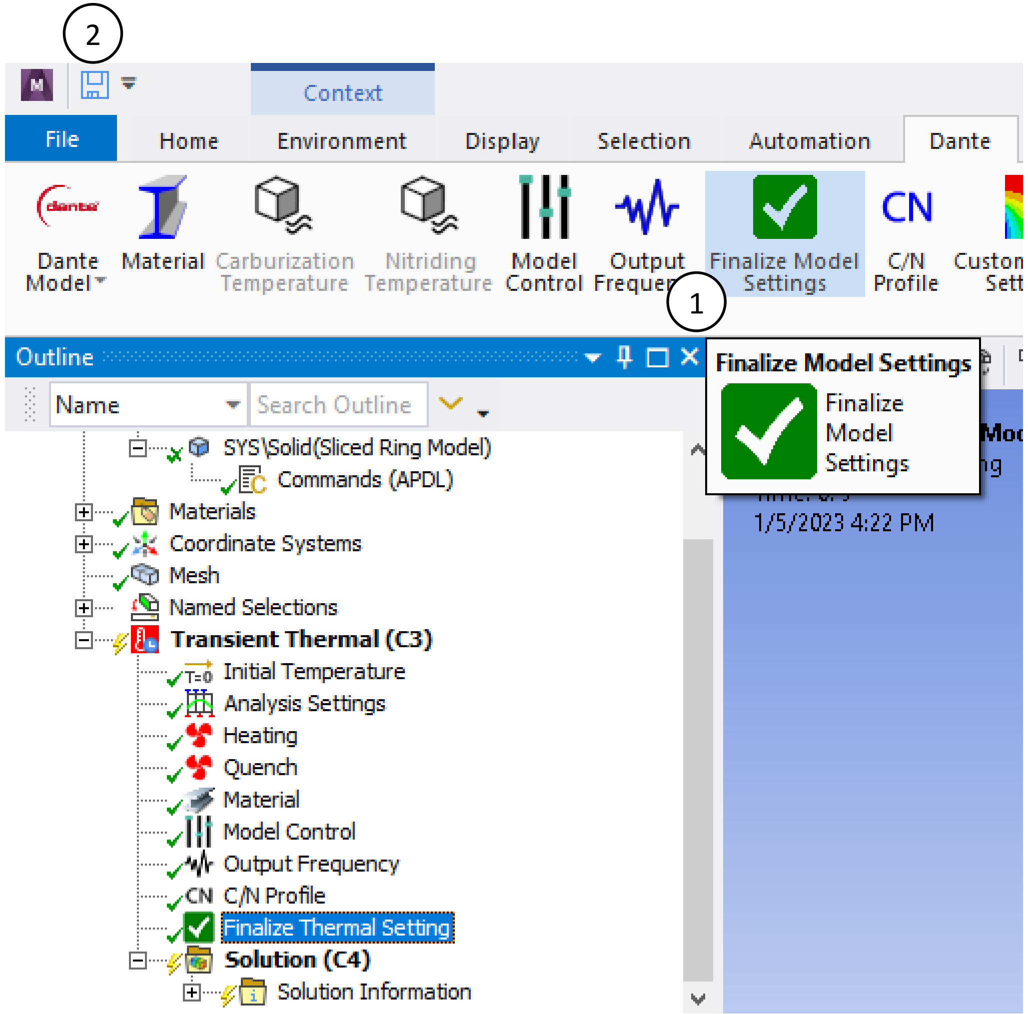


This applies the boundary condition only to the quench step

Step 15: Running the Thermal Model

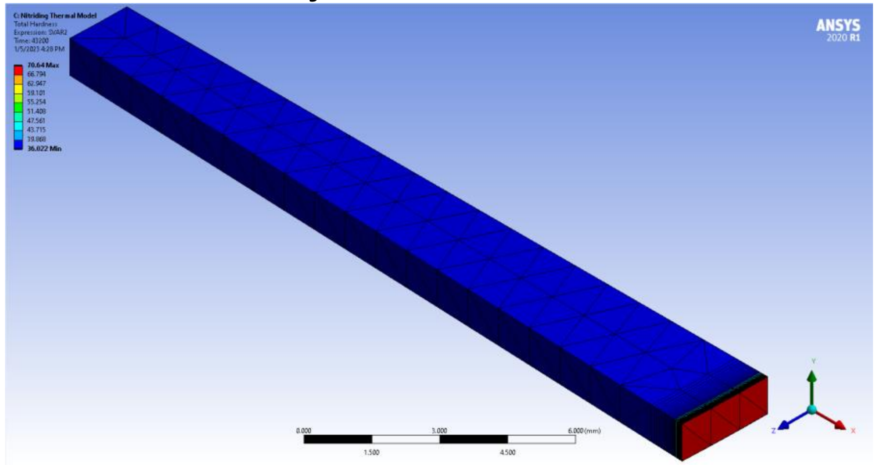
1. Select ***Finalize Model Settings*** from the ***Dante toolbar***
2. ***Save*** the model
3. Right-click ***Solution* → Solve**

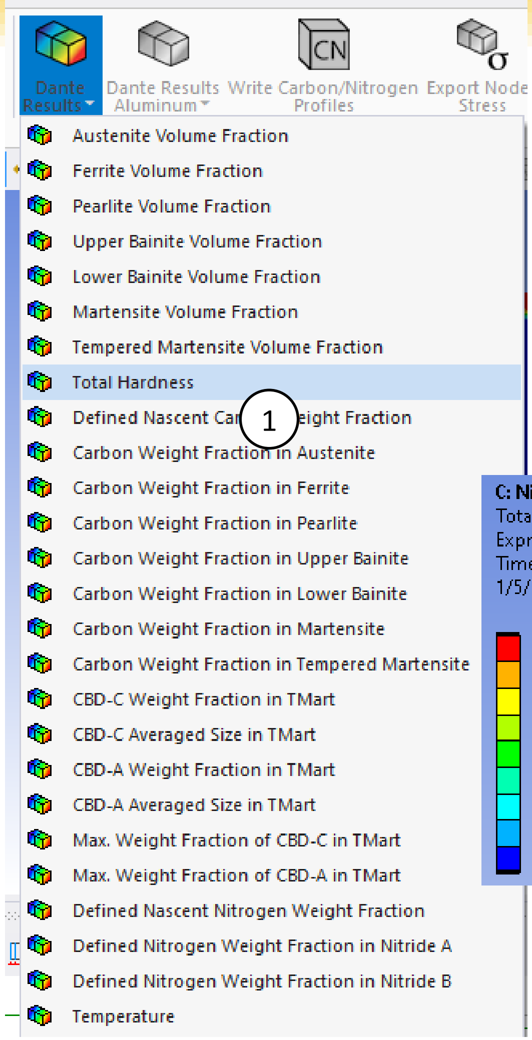


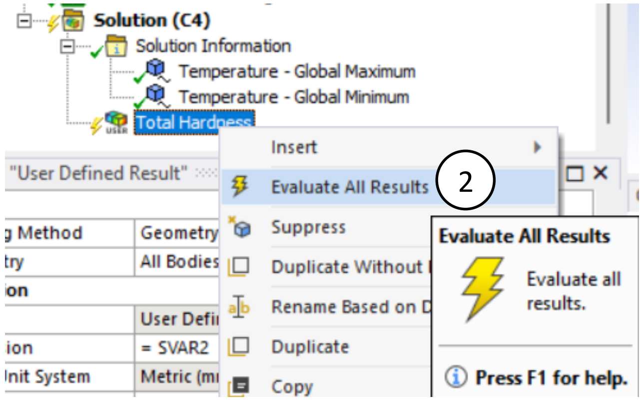


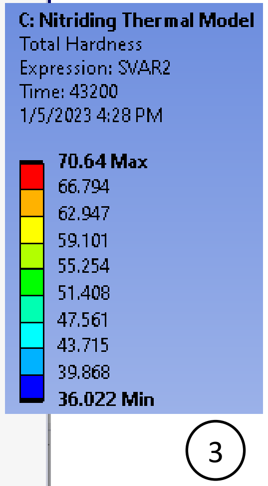
Step 16: Post-Processing Thermal History

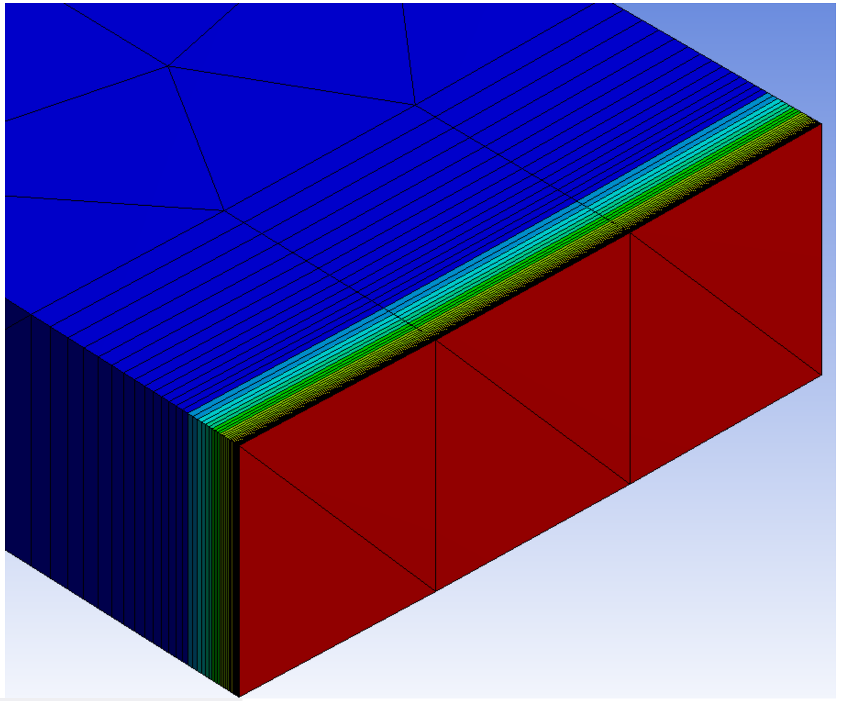
1. Under ***Dante Results*** select ***Total Hardness***
2. Right click ***Total Hardness → Evaluate All Results***
3. Check results
4. Save the Project and close ***Mechanical***



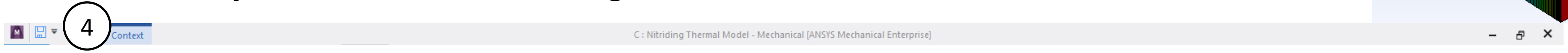








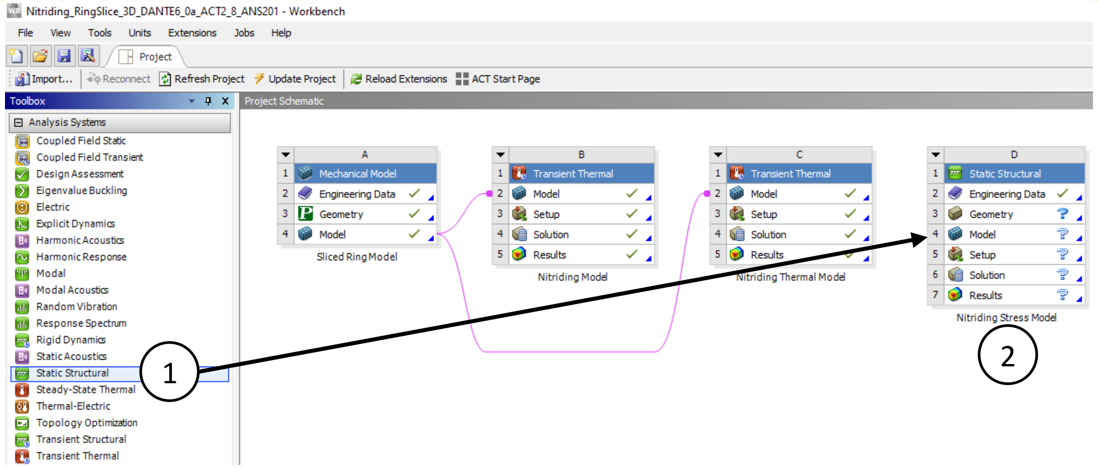
NOTE: These DANTE results can be added before running the model, when solving the model, they will be evaluated along with it

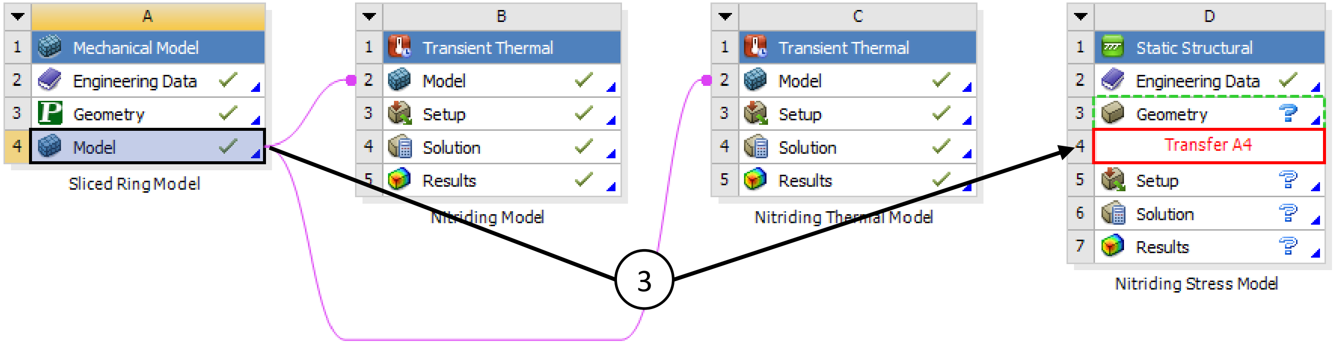


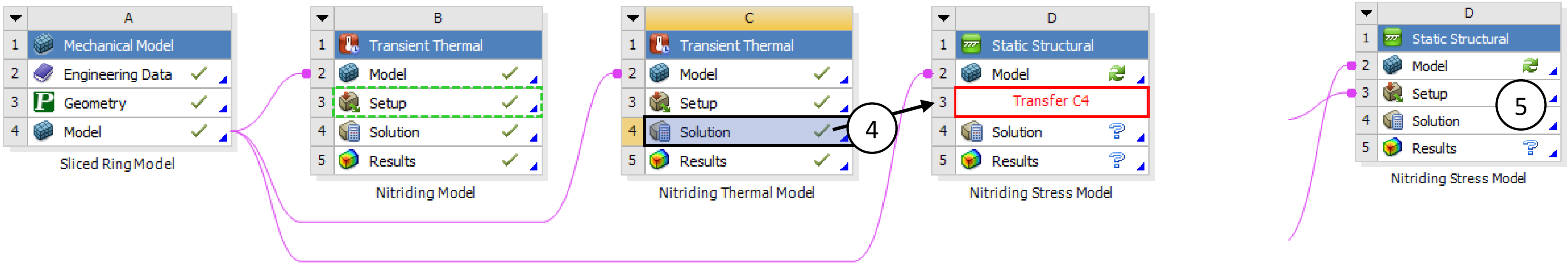
Quench Hardening Stress Model

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

1. Drag and drop a ***Static Structural Analysis Systems*** into the ***Project Schematic***
2. Rename it as "***Nitriding Stress Model***"
3. Drag and drop the ***Model*** from ***Sliced Ring Model*** to ***Nitriding Stress Model***
4. Drag and drop the ***Solution*** from ***Nitriding Thermal Model*** to the ***Setup*** of ***Nitriding Stress Model***
5. Double click ***Setup*** in the ***Nitriding Stress Model***



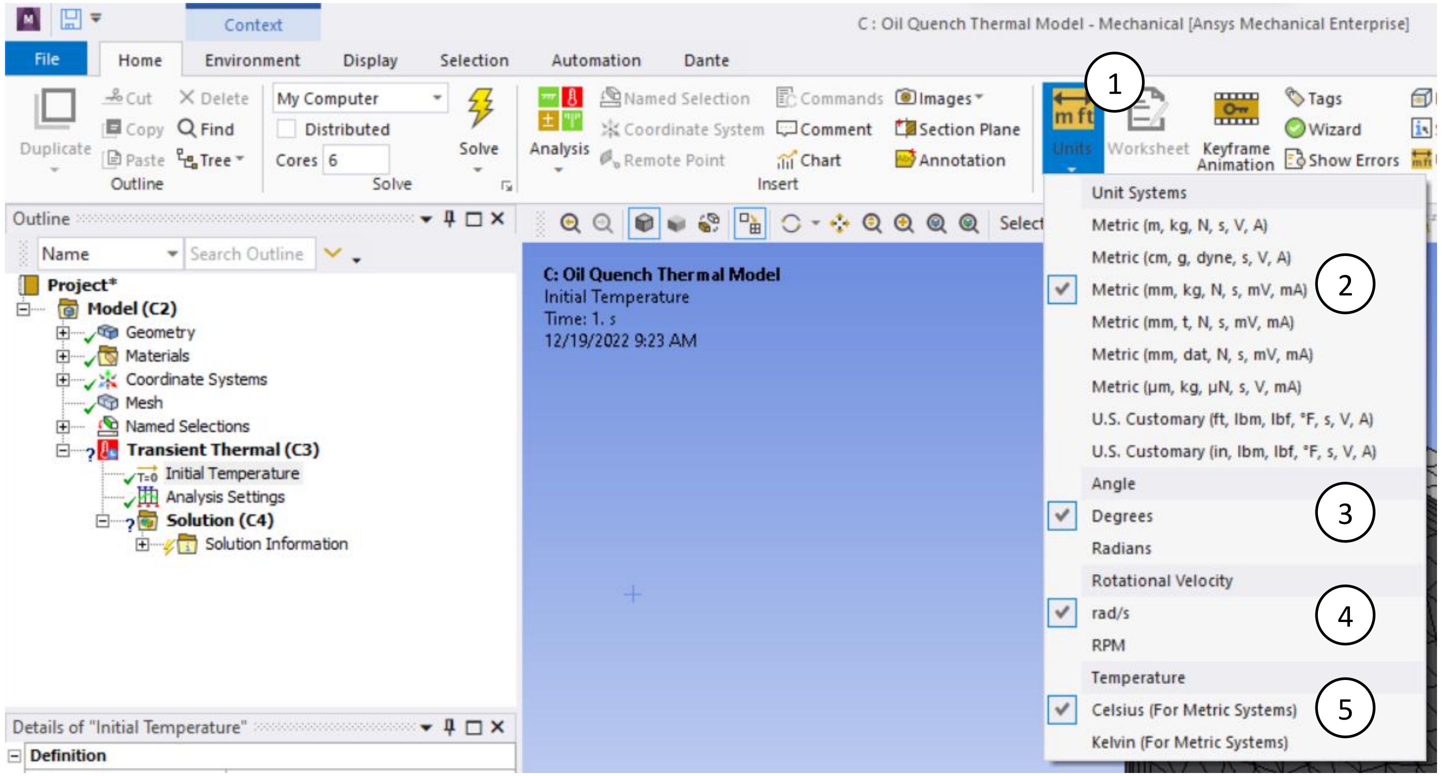




Step 2: Check Units

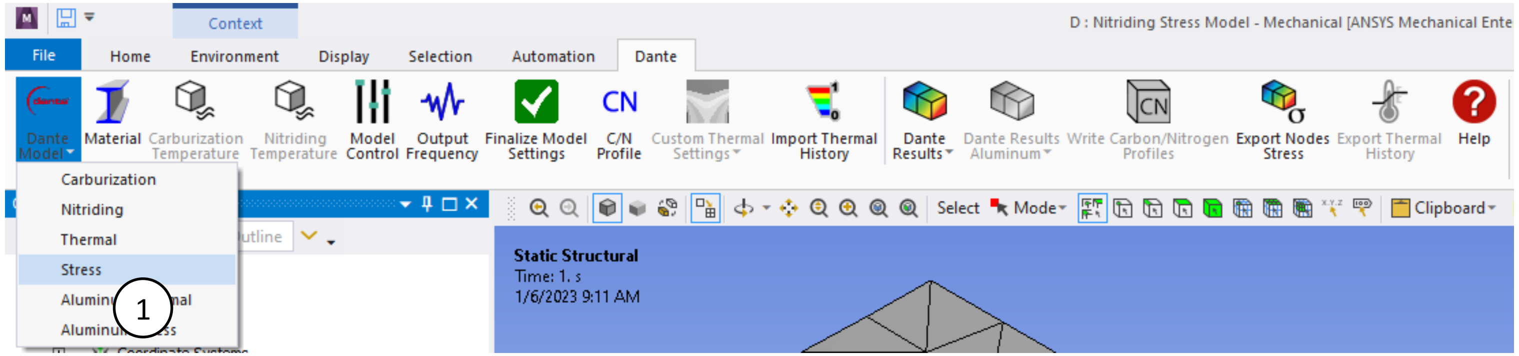
It is critical that the units be properly defined

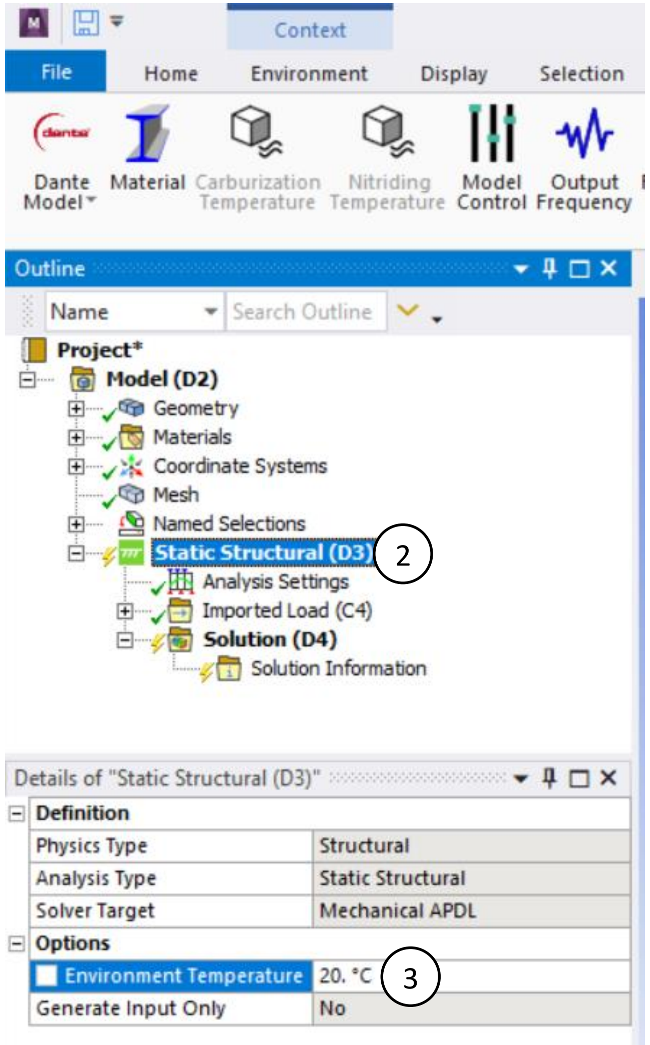
1. **Click on Units in the Home toolbar**
2. **Select Metric (mm, kg, N, s, mV, mA)**
3. **Select Degrees**
4. **Select rad/s (This isn't critical as there is no motion defined in the heat treatment models)**
5. **Select Celsius (For Metric Systems)**



Step 3: Define DANTE Model & Initial Temperature

1. Click on ***Dante Model*** in the ***Dante Toolbar*** and select ***Stress***
2. Click on ***Static Structural*** in the ***Project Tree***
3. In the ***Details of "Static Structural",*** change **Environment Temperature** to ***20 °C***



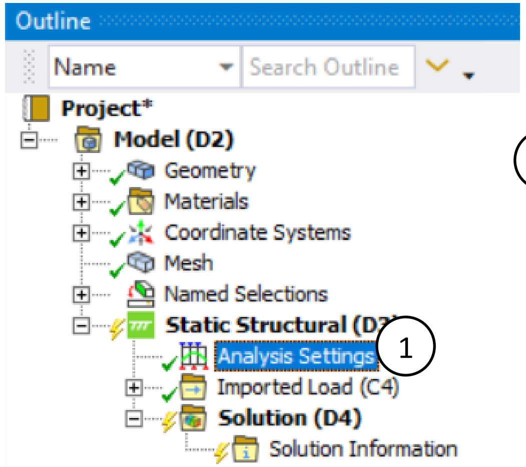


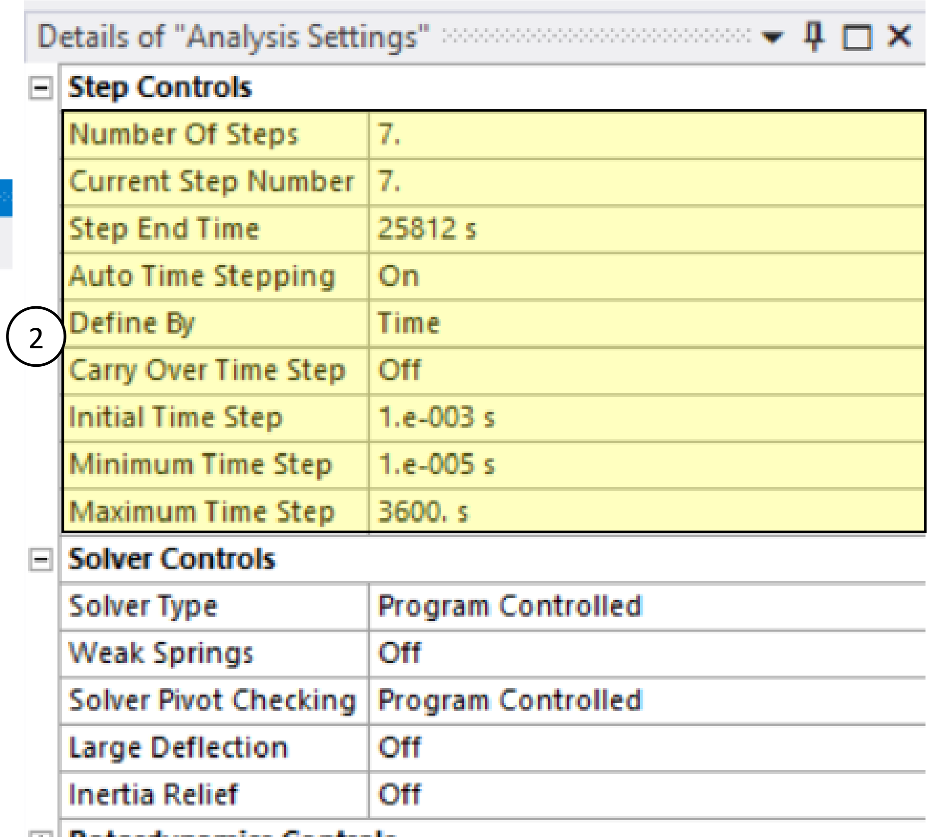
Step 4: Define Processing Steps

1. Select **Analysis Settings**
2. Modify the ***Details of "Analysis Settings"***

   NOTE: The step numbers MUST be entered from highest to lowest; i.e., start with Current Step Number 7

3. Repeat for ***Current Step Numbers*** 6 through 1



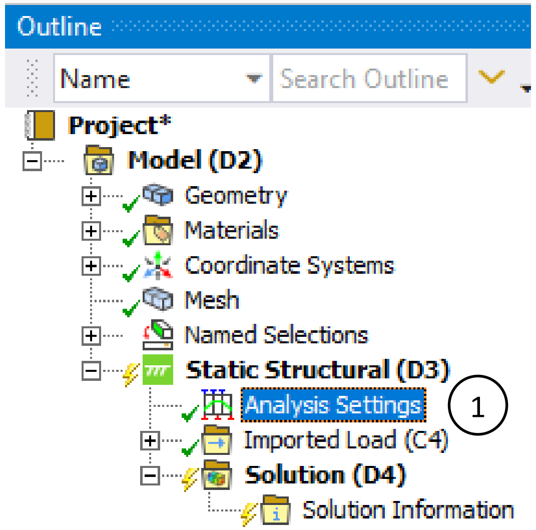


3

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	25812						3600
6	18612						3600
5	11412						3600
4	4212	On	Time	Off	1.00E-02	1.00E-05	600
3	3612						10
2	3601						1
1	3600			N/A			3600

Step 4: Define Processing Steps

1. Select **Analysis Settings**
2. Enter ***3*** for ***Number of Steps***
3. Set ***Current Step Number*** to ***3*** and enter values
4. Set ***Current Step Number*** to ***2*** and enter values
5. Set ***Current Step Number*** to ***1*** and enter values



3

Details of "Analysis Settings"

Step Controls	
Number Of Steps	3.
Current Step Number	3.
Step End Time	43200 s
Auto Time Stepping	On
Define By	Time
Carry Over Time Step	Off
Initial Time Step	1.e-002 s
Minimum Time Step	1.e-005 s
Maximum Time Step	3600. s
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	Off
Inertia Relief	Off

2

4

Details of "Analysis Settings"

Step Controls	
Number Of Steps	3.
Current Step Number	2.
Step End Time	39600 s
Auto Time Stepping	On
Define By	Time
Carry Over Time Step	Off
Initial Time Step	1.e-002 s
Minimum Time Step	1.e-005 s
Maximum Time Step	36000 s
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	Off
Inertia Relief	Off

5

Details of "Analysis Settings"

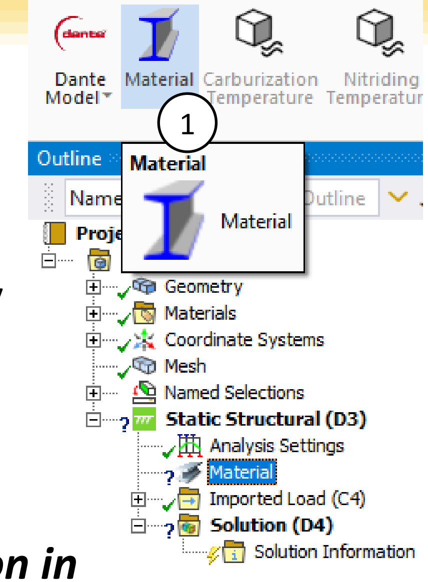
Step Controls	
Number Of Steps	3.
Current Step Number	1.
Step End Time	3600. s
Auto Time Stepping	On
Define By	Time
Initial Time Step	1.e-002 s
Minimum Time Step	1.e-005 s
Maximum Time Step	3600. s
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	Off
Inertia Relief	Off

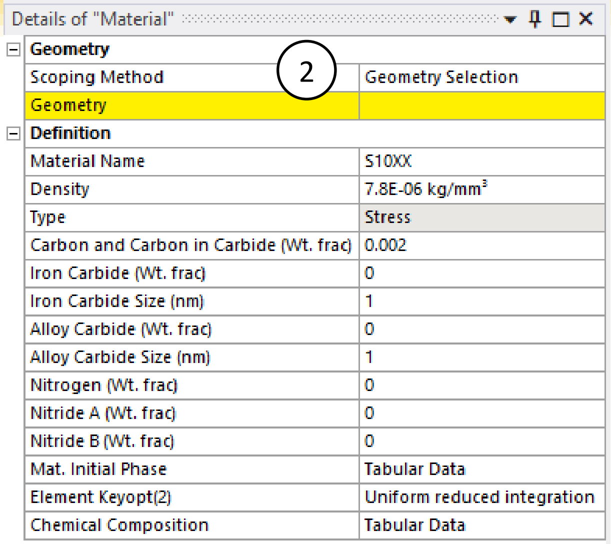
<table>
<tr><th>Current Step Number</th><th>Step End Time</th><th>Auto Time Stepping</th><th>Define By</th><th>Carry Over Time Step</th><th>Initial Time Step</th><th>Minimum Time Step</th><th>Maximum Time Step</th></tr>
<tr><td>3</td><td>43200</td><td rowspan="3">On</td><td rowspan="3">Time</td><td rowspan="2">Off</td><td rowspan="3">1.00E-02</td><td rowspan="3">1.00E-05</td><td>3600</td></tr>
<tr><td>2</td><td>39600</td><td>36000</td></tr>
<tr><td>1</td><td>3600</td><td>N/A</td><td>3600</td></tr>
</table>

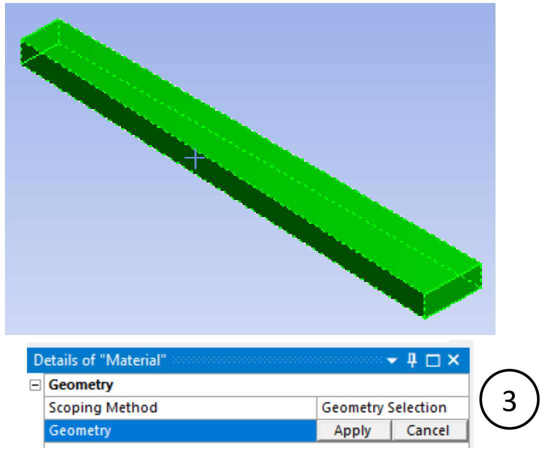
The part heat up, nitriding, and cooling down steps are analyzed in the stress model

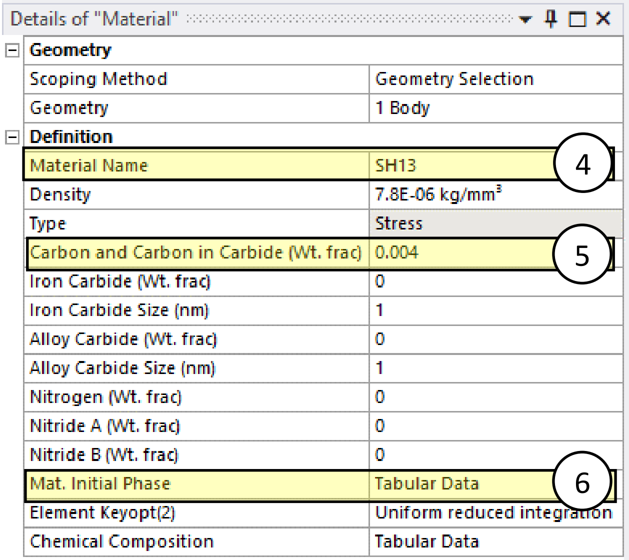
Step 5: Assign Material

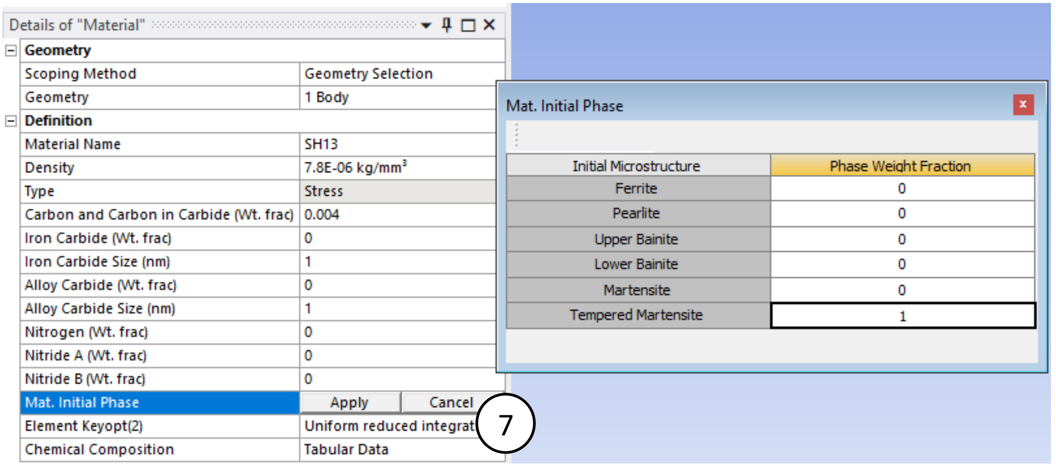
1. Select ***Material*** from the ***Dante Toolbar***
2. Click on the yellow box next to ***Geometry*** → Select the part body
3. Select ***Apply*** for ***Geometry***
4. Select ***SH13*** for ***Material Name***
5. Enter ***0.004*** for ***Carbon and Carbon in Carbide***
6. Click ***Tabular Data*** for ***Mat. Initial Phase*** and set values as shown
    - Initial microstructure composed fully of tempered martensite, set to ***1*** with other phases ***0***
7. Click ***Apply***





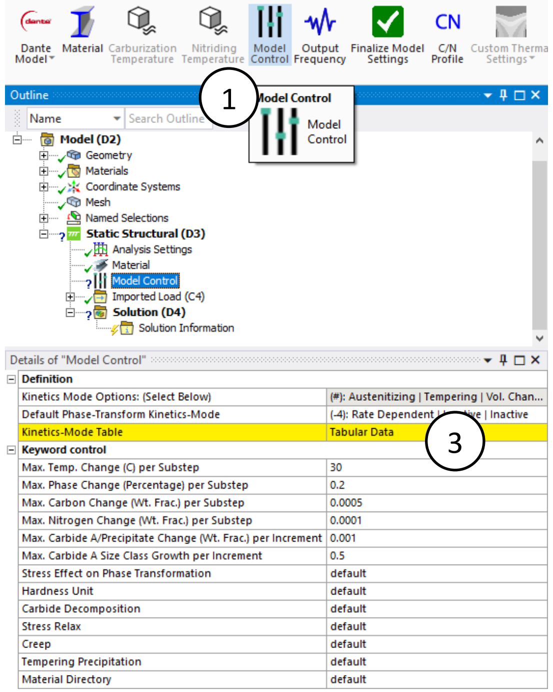


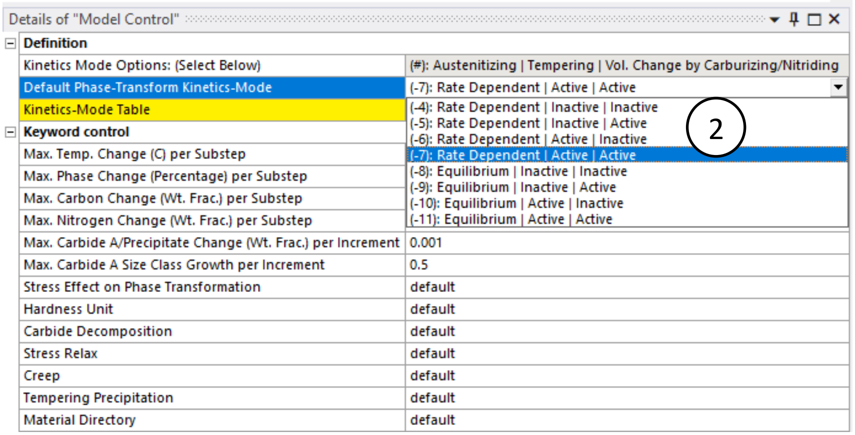


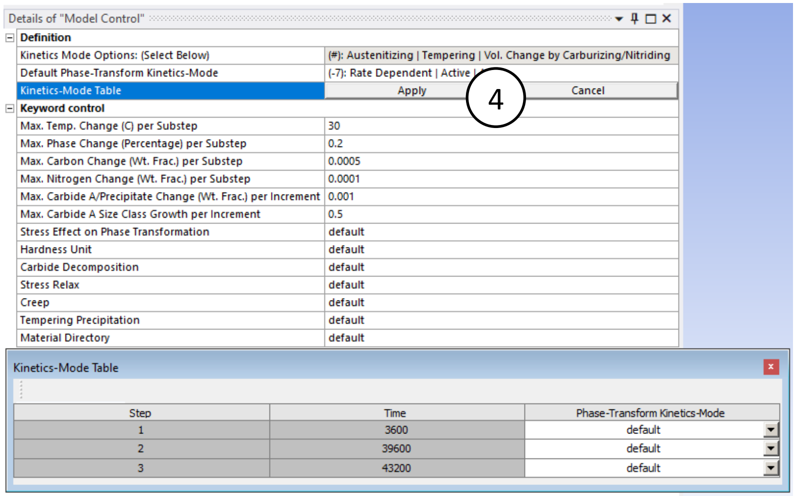


Step 6: Define Control File

1. Select ***Model Control*** in the ***Dante toolbar***
2. Select ***Default Phase-Transform Kinetics-Mode → (-7): Rate Dependent|Active|Active***
3. Click ***Tabular Data → Kinetics-Mode Table***
4. Click ***Apply***

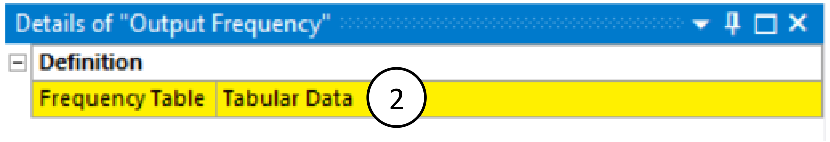


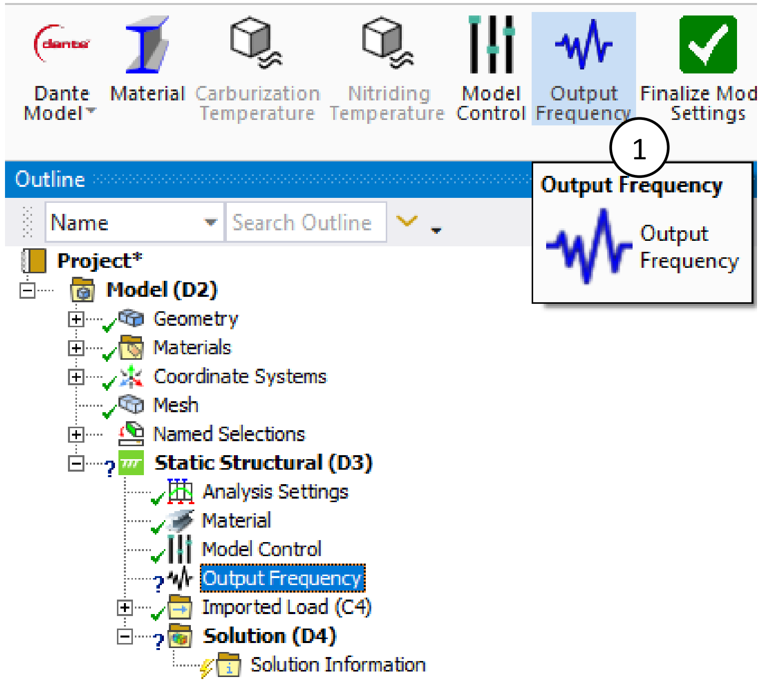


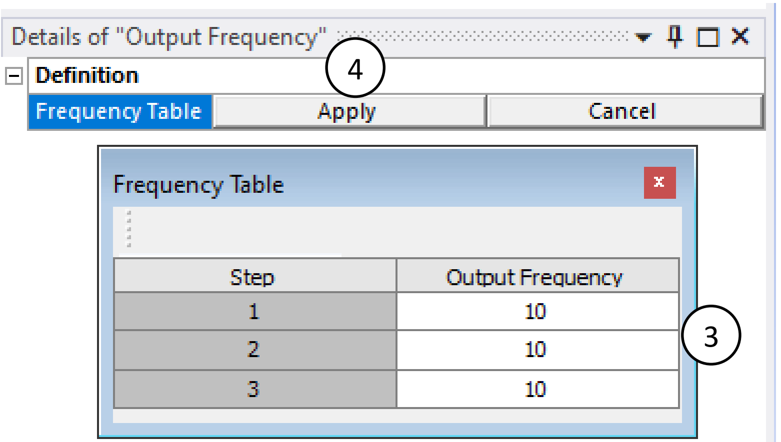


Step 7: Define Output Frequency

1. Select ***Output Frequency*** from the ***Dante toolbar***
2. In ***Details of "Output Frequency"***, click on ***Tabular Data***
3. Leave the default frequency values as 10
4. Click **Apply**

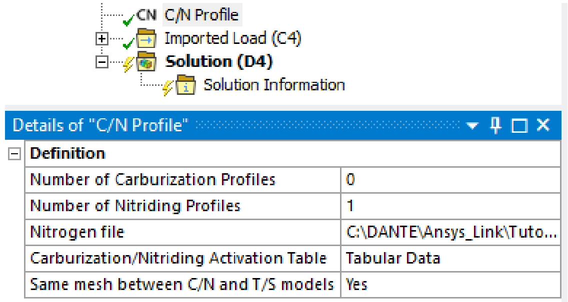


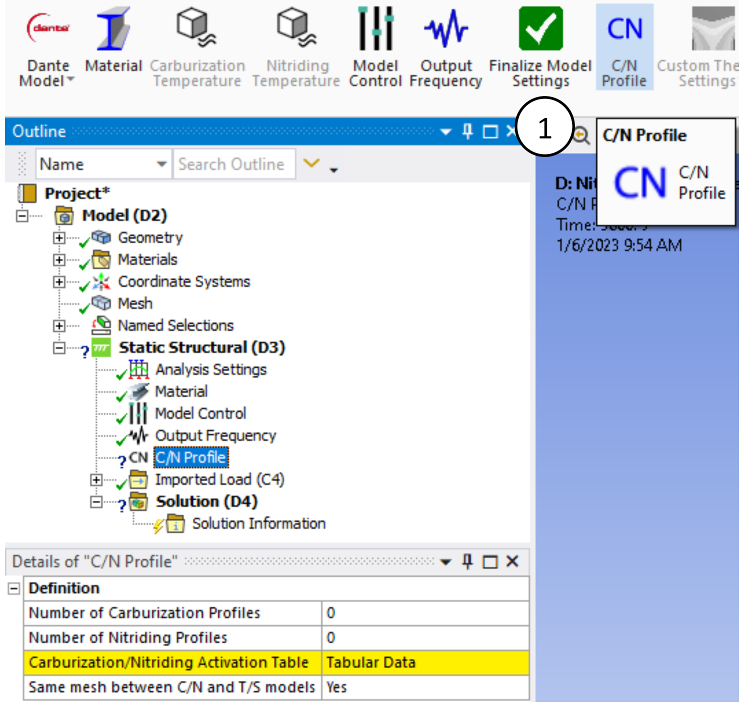


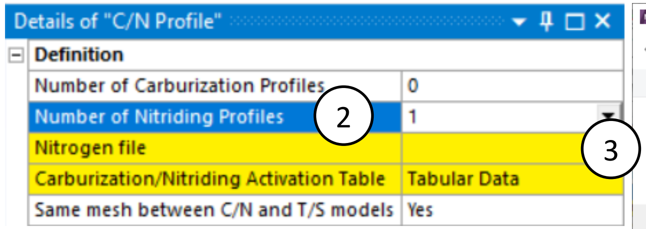


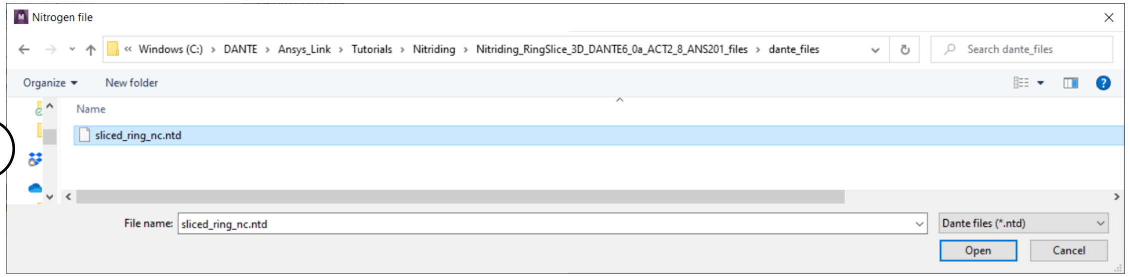
Step 8: Importing Nitrogen Profiles

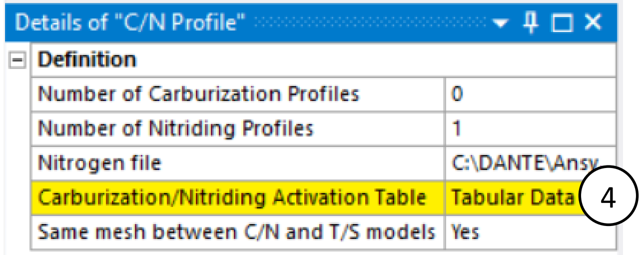
1. Select ***C/N Profile*** from the ***DANTE Toolbar***
2. Set ***Number of Nitriding Profiles → 1***
3. Click ***Nitriding file***, navigate to ***dante_files***, select the nitrogen profile and click ***Open***
4. Under ***Carburization/Nitriding Activation Table***, click ***Tabular Data***
5. Set ***Nitriding → 1*** at ***Step 2*** to activate it and click ***Apply***

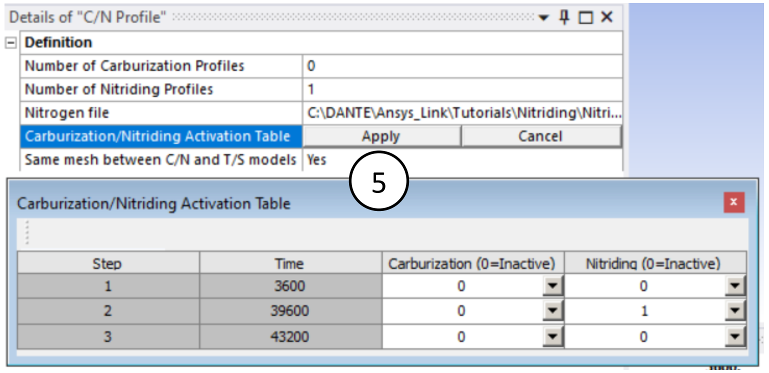






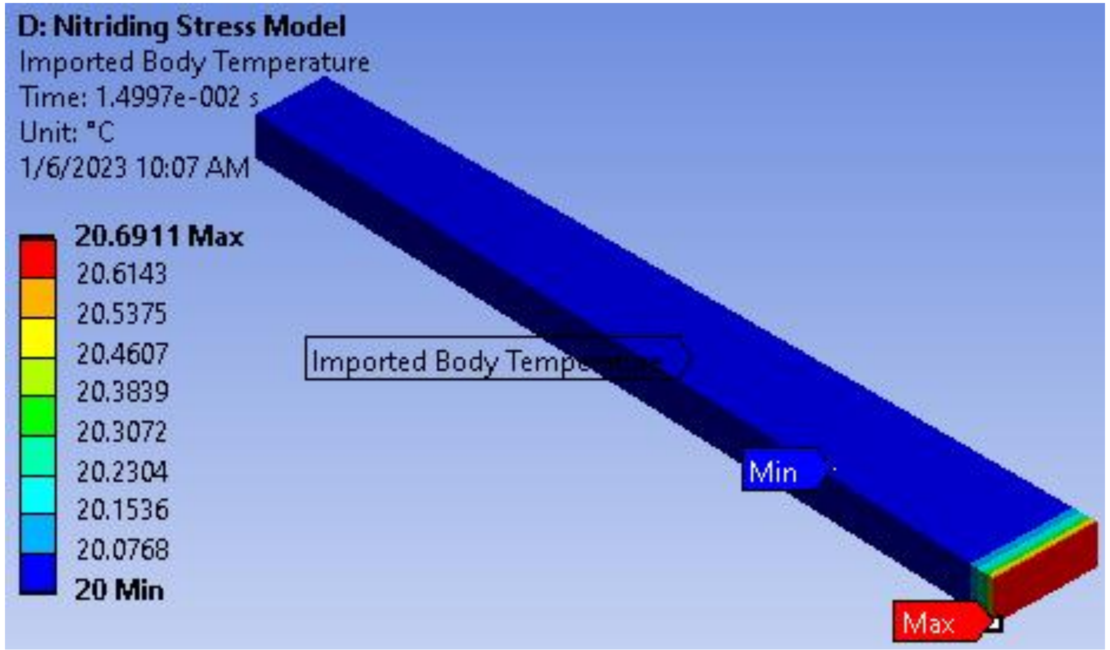


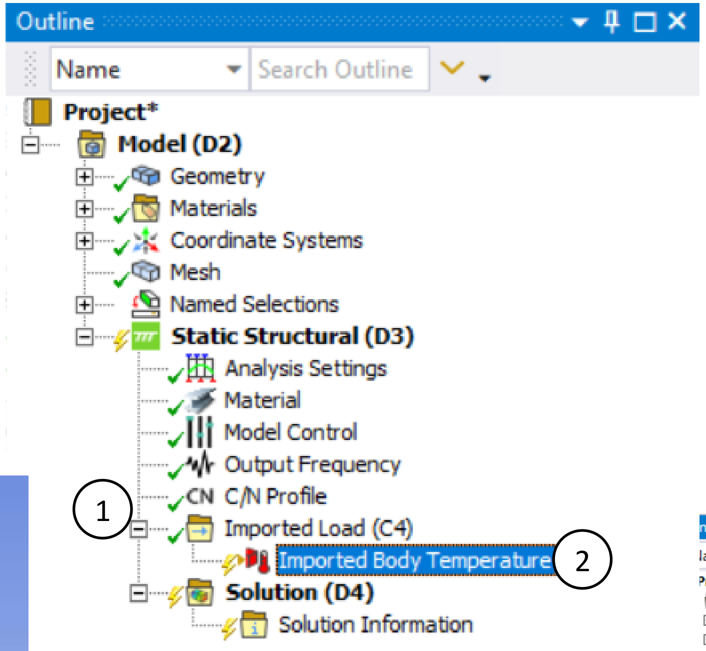


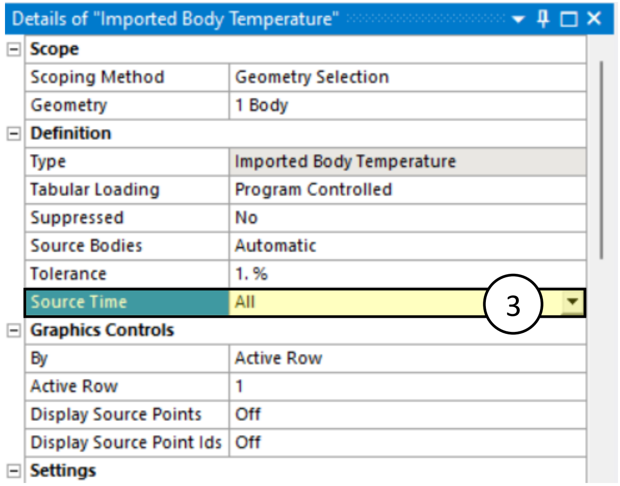


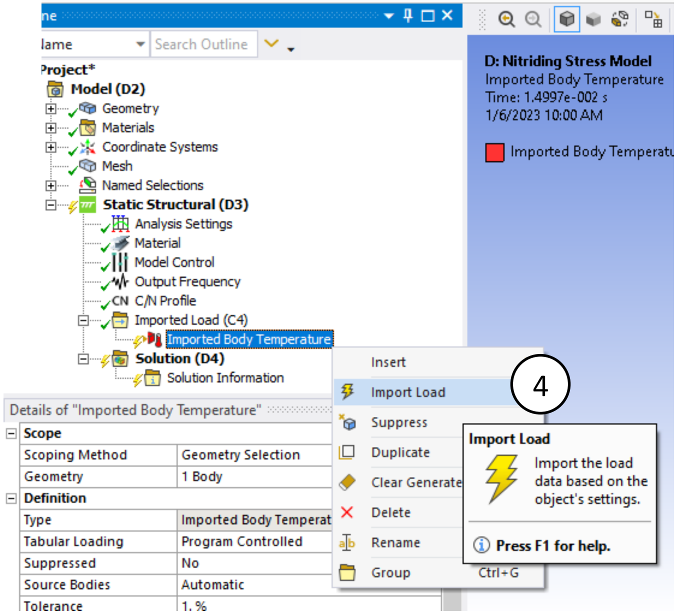
Step 9: Import Thermal History

1. Expand ***Imported Load (C4)*** in the ***Project Tree***
2. Select ***Imported Body Temperature***
3. In ***Details of Imported Body Temperature***, set the ***Source Time*** to ***All***
4. Right click on ***Imported Body Temperature*** in the ***Project Tree*** and select ***Import Load***



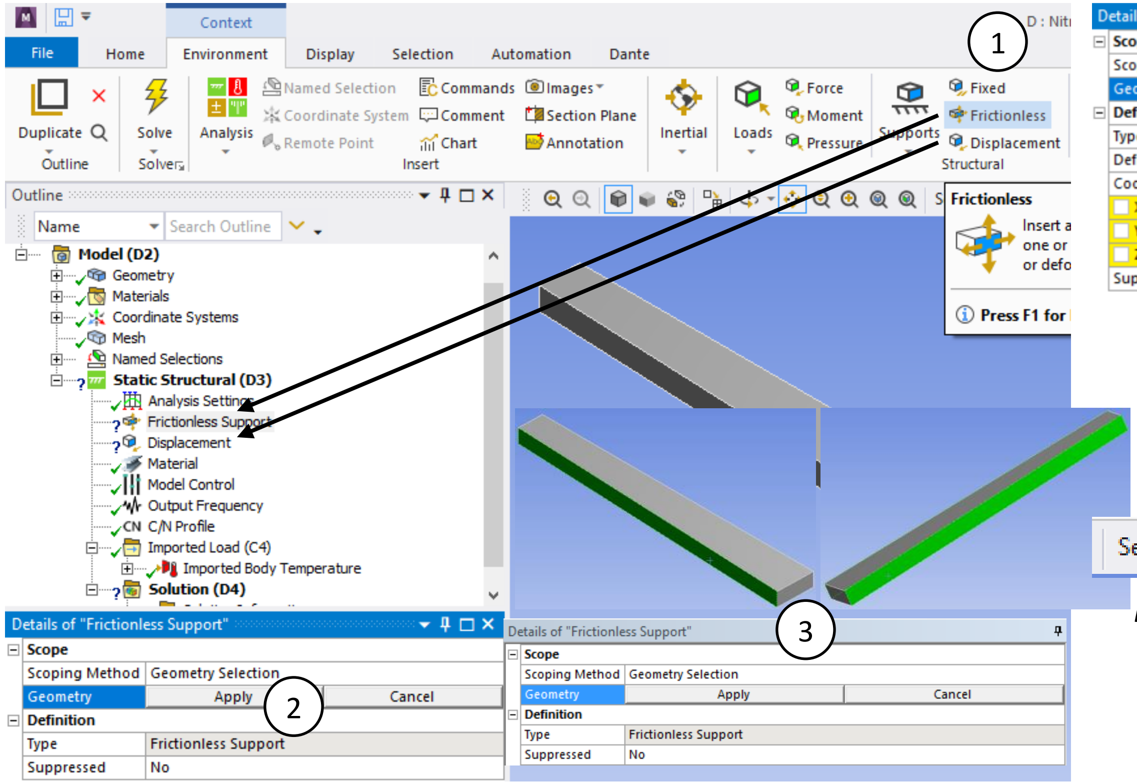


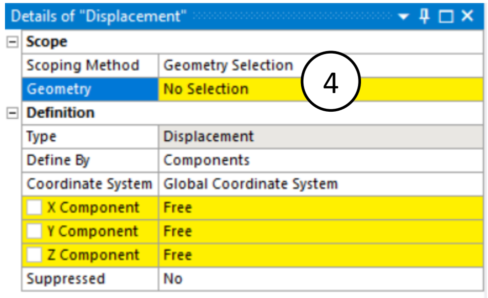




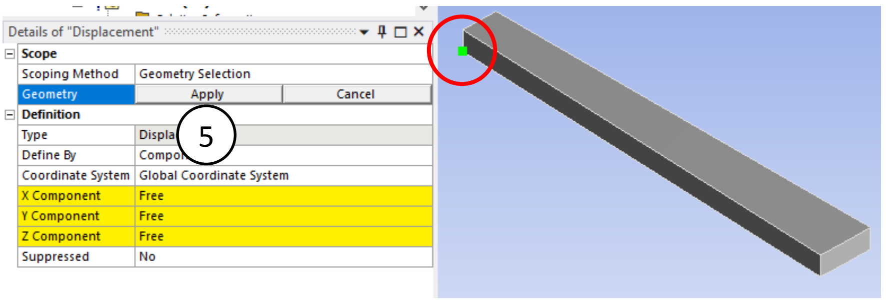
Step 10: Define Mechanical Boundary Conditions

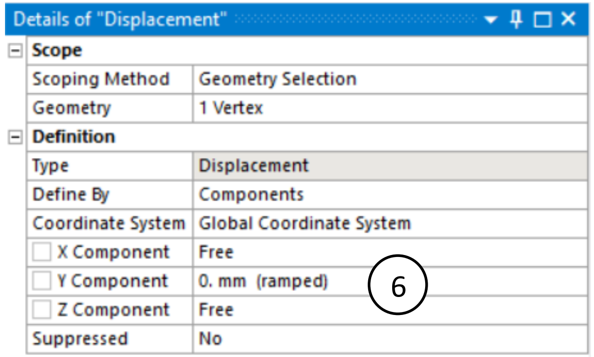
1. From the ***Environment toolbar,*** add a ***Frictionless Support*** and ***Displacement*** to the ***Project Tree***
   - ➢ Click on ***Static Structural*** to add structural boundary conditions
2. In the ***Details of "Frictionless Support",*** click the yellow box next to ***Geometry***
3. Select the two internal faces of the ring slice holding the ***Ctrl*** key and click ***Apply***
4. In the ***Details of "Displacement",*** click the yellow box next to ***Geometry***
5. Select the bottom inner vertex and click ***Apply***
6. In the ***Details of "Displacement",*** set the ***Y Component*** to ***0***





Note: Use the selection modes to select vertices and faces



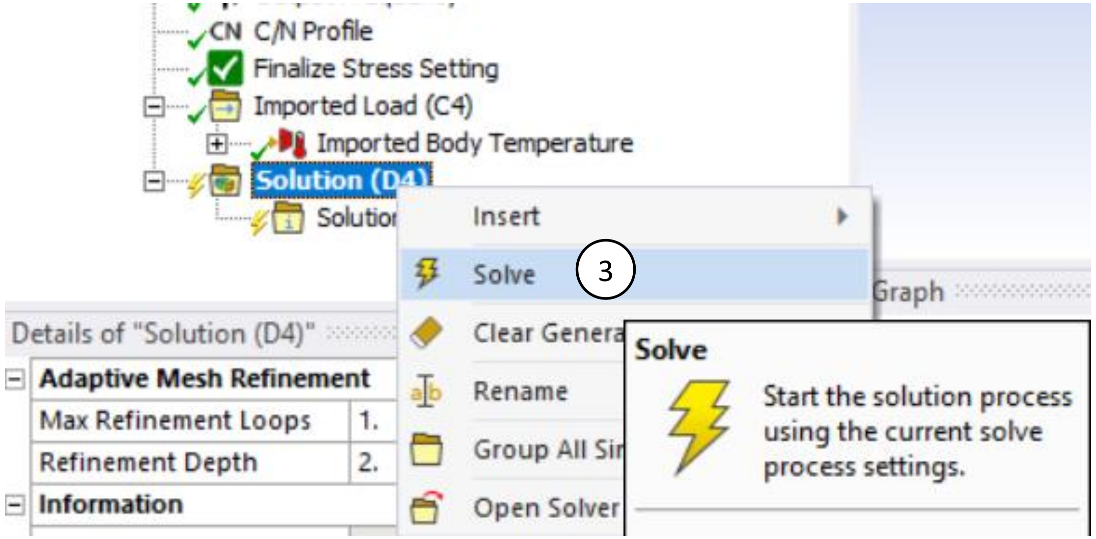


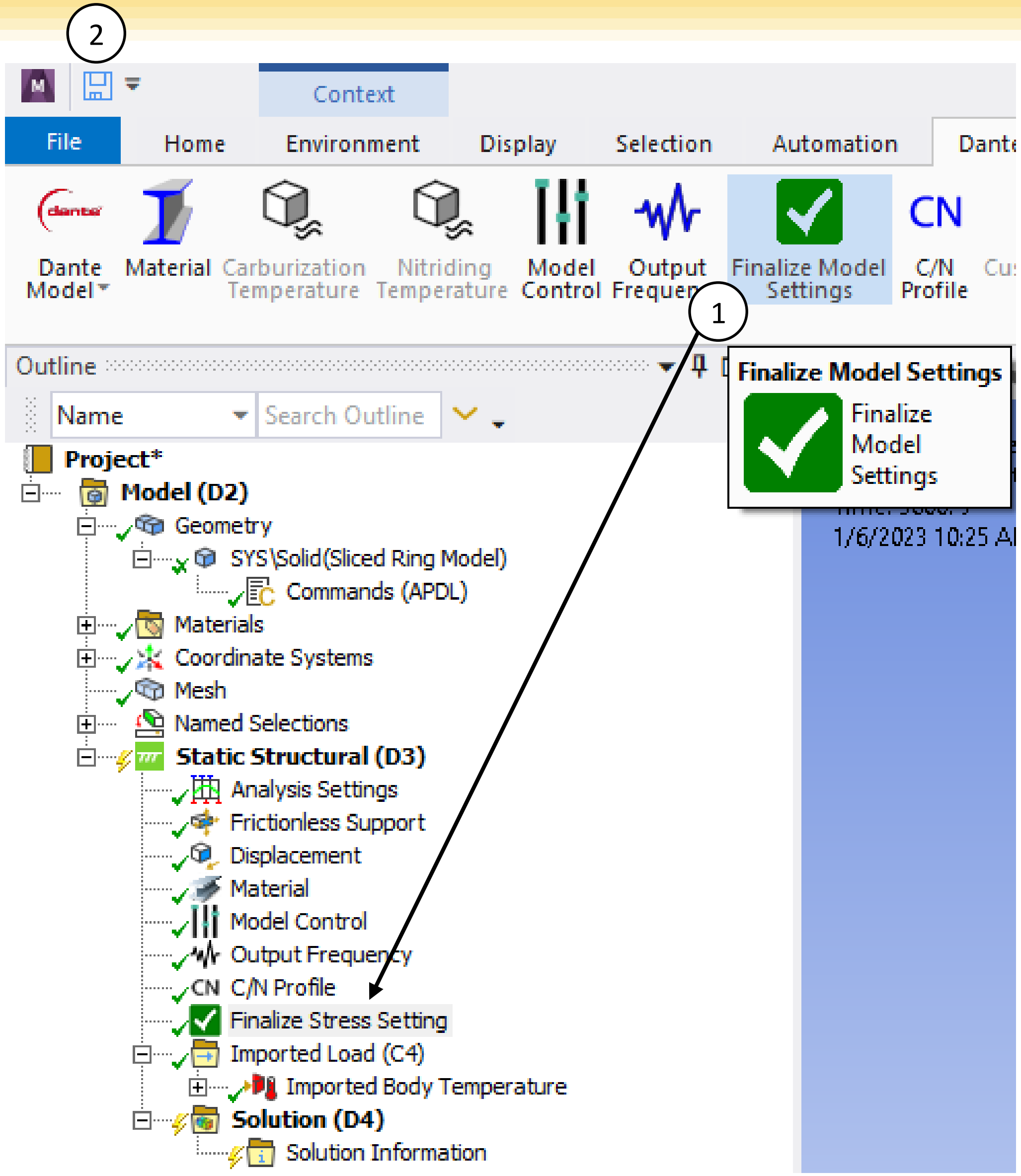
Step 11: Running the Stress Model

1. Select ***Finalize Model Settings*** from the ***Dante toolbar***

2. ***Save*** the model

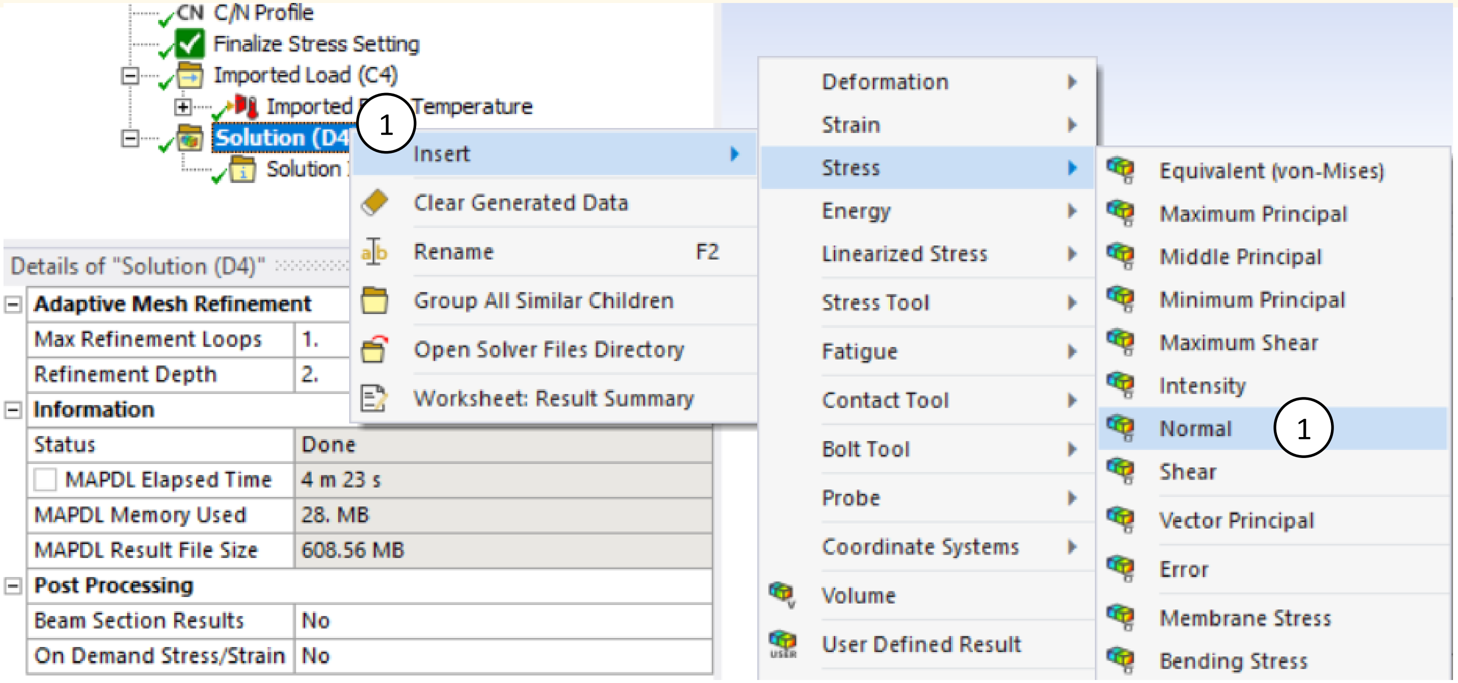
3. Right-click ***Solution*** → **Solve**

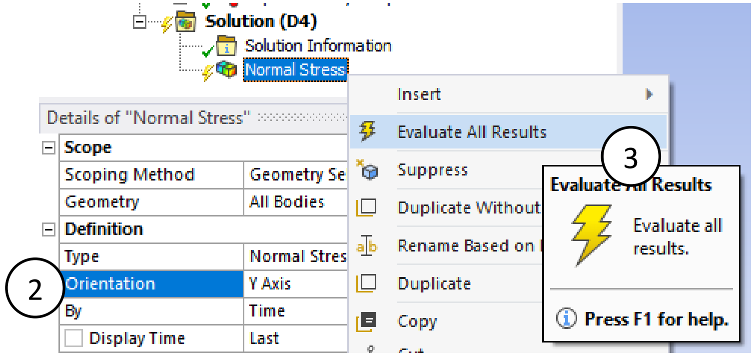


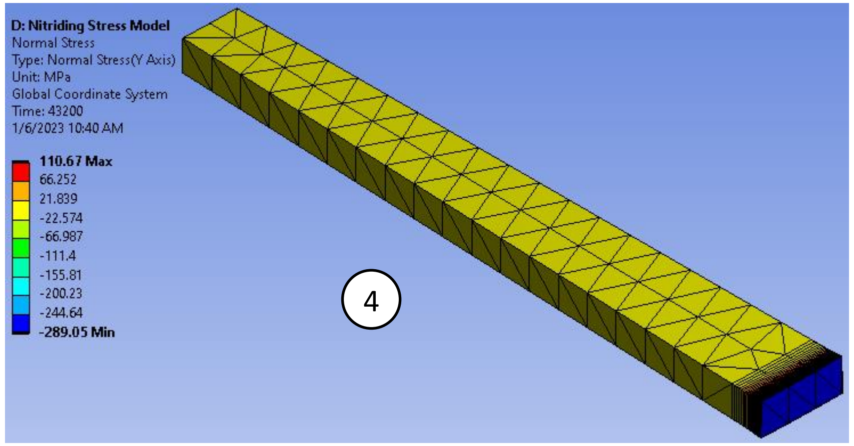


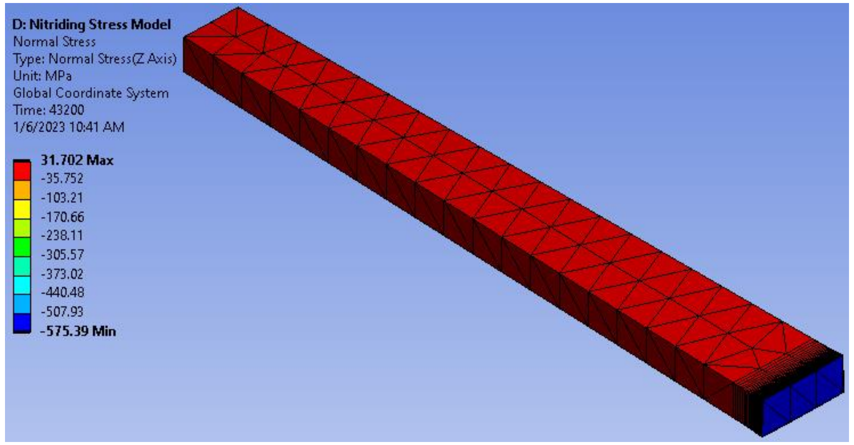
Step 12: Stress Model Post-Processing, Stress Results

1. Right-click ***Solution*** in the ***Project Tree*** and select ***Insert → Stress → Normal***
2. In ***Details of "Normal Stress",*** select the ***Y Axis***, signifying axial stress
3. Right click on ***Normal Stress*** and select ***Evaluate All Results***









The part surface is in high compression in the axial and circumferential direction

NOTE: This concludes the tutorial, save and close Ansys Mechanical