Additive Manufacturing Tutorials

Ansys
ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2024 R1
January 2024

Published in the U.S.A.

Table of Contents

What You Need to Know About AM Tutorials

These tutorial problems demonstrate how to use some of the features in the Ansys family of products dedicated to Additive Manufacturing. Each tutorial is a complete step-by-step simulation procedure.

The tutorials in this document were run on a 64-bit Windows 10 system. Your results may vary depending upon your computer hardware and operating system.

Use the link provided under **Tutorial Files** in the problem description of each tutorial to obtain the file(s) required to run the tutorial.

Chapter 1: Calibration - Double Arches Geometry

This example demonstrates the workflow for an additive calibration using an LPBF Thermal-Structural simulation in Mechanical. The Thermal Strain Scaling Factor (TSSF) is obtained from this calibration process for an AM build using Inconel 718 material.

The following features and capabilities are used:

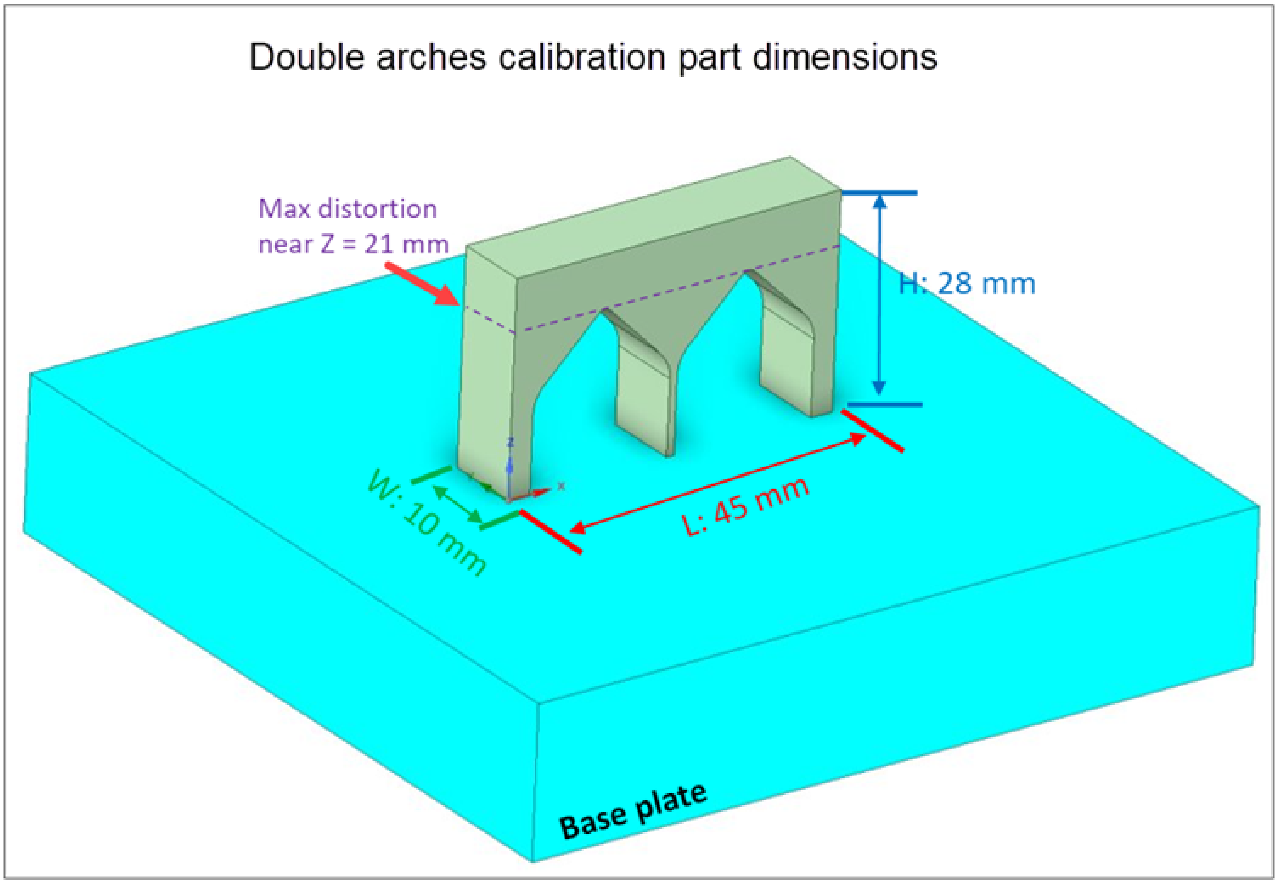
- Thermal-structural simulation
- LPBF Process Add-on
- LPBF Setup Wizard
- Geometry construction of base plate
- Cartesian mesh
- Direct optimization module to perform the calibration iterations

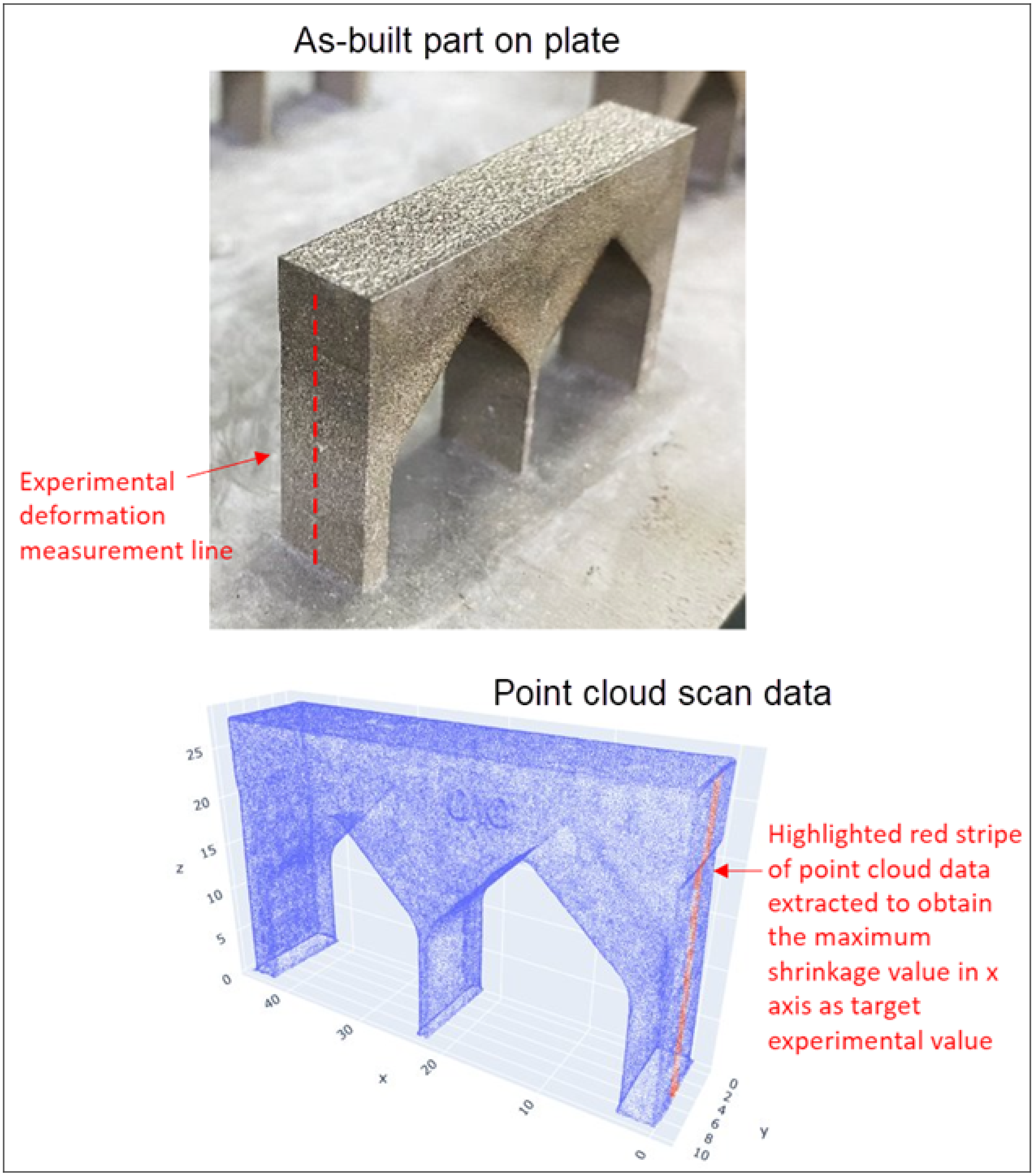
Tutorial steps:

1.1. Problem Description

This example part is used to demonstrate the overall calibration process using the LPBF Setup Wizard in Mechanical. You may want to design/choose the most appropriate calibration geometry for your own scenario by referring to Performing a Calibration in the *LPBF Simulation Guide*.

The double arches geometry is shown in the following schematic. This part's overall dimensions are 45 mm by 10 mm by 28 mm in the X, Y, and Z directions, respectively. The part is designed so that the major distortion can be measured near the region of change in cross-sectional area at Z=21 mm, as indicated by the red arrow. Following fabrication, the as-built on-plate distortion of the part is measured along the center line of the side edge, as highlighted by the red dashed line in the second figure. This measurement line has coordinates X=0 and Y=5 mm. The image depicts an as-built calibration part on the plate after fabrication.





The calibration part was fabricated using a laser powder bed fusion process with the machine process parameter set as shown in the following table. Some of these processing parameters are used as inputs in the simulation. The part was built with Inconel 718 material directly on a build plate with no support structures.

Ti-6Al-4V Processing Parameters	
Layer Thickness (µm)	60
Preheat Temperature (°C)	200

Ti-6Al-4V Processing Parameters	
Inert gas	Ar
Beam Diameter (μm)	85
Laser Power (W)	350
Laser Speed (mm/s)	800
Hatch Distance (mm)	0.12
Stripe Length (mm)	10
Starting Angle (°)	45
Rotation Angle (°)	67
Energy Density (J/mm^3)	61

The maximum as-built on-plate distortion for this build was 0.449 mm in shrinkage near Z = 21 mm located along the measurement line shown in the figures. This distortion value will be used as the target value for the simulation.

Tutorial Files

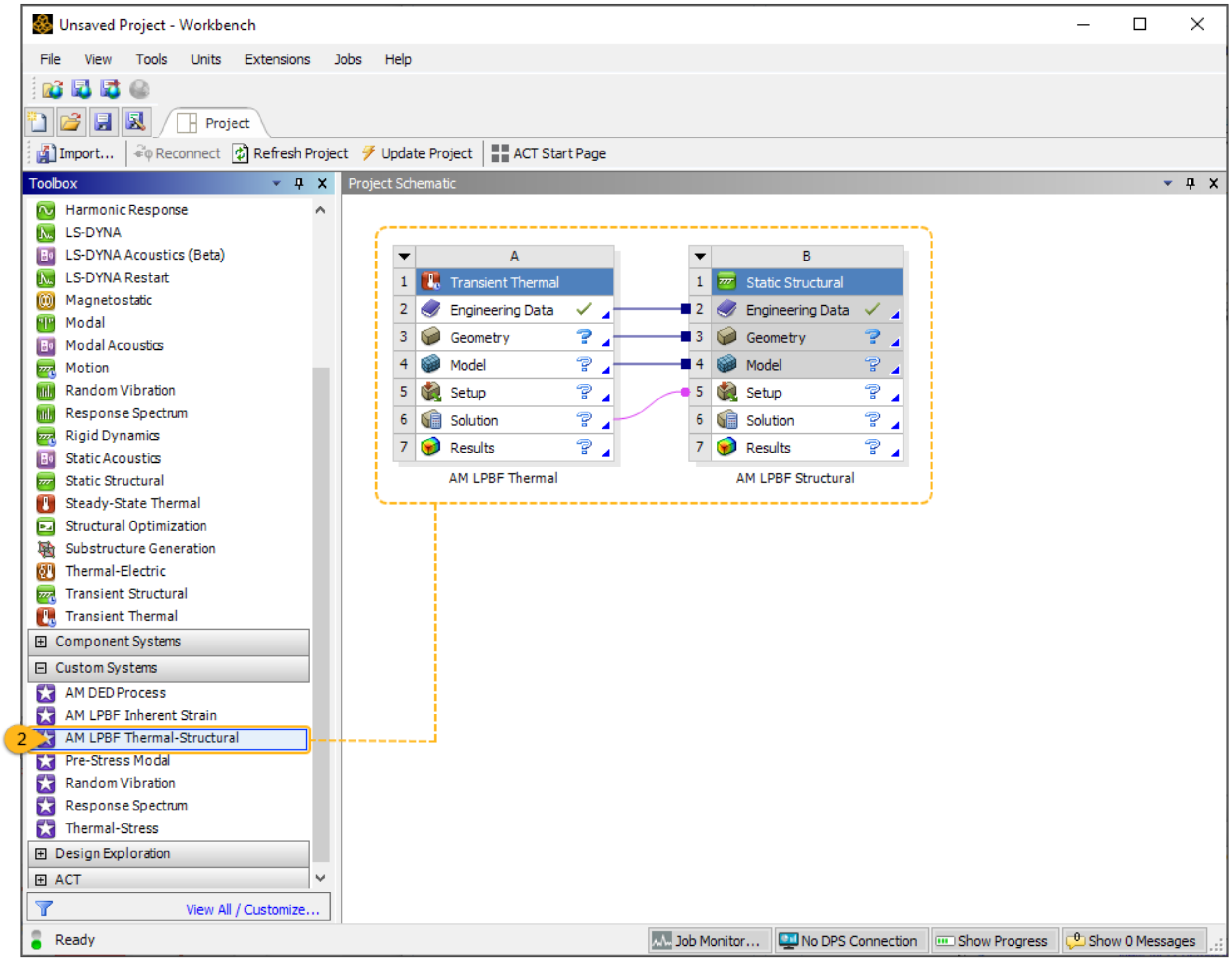
Click here to download:

`Double_arches_p.stl` — Geometry file of the double arches calibration part, saved as an .stl document. There are no supports for this model.

1.2. Create the Analysis System

1. Open Ansys Workbench.

2. In the toolbox on the left side of the window, scroll down to Custom Systems, expand the selection, and double-click **AM LPBF Thermal-Structural** to bring up the linked Transient Thermal, Static Structural system in the Project Schematic.

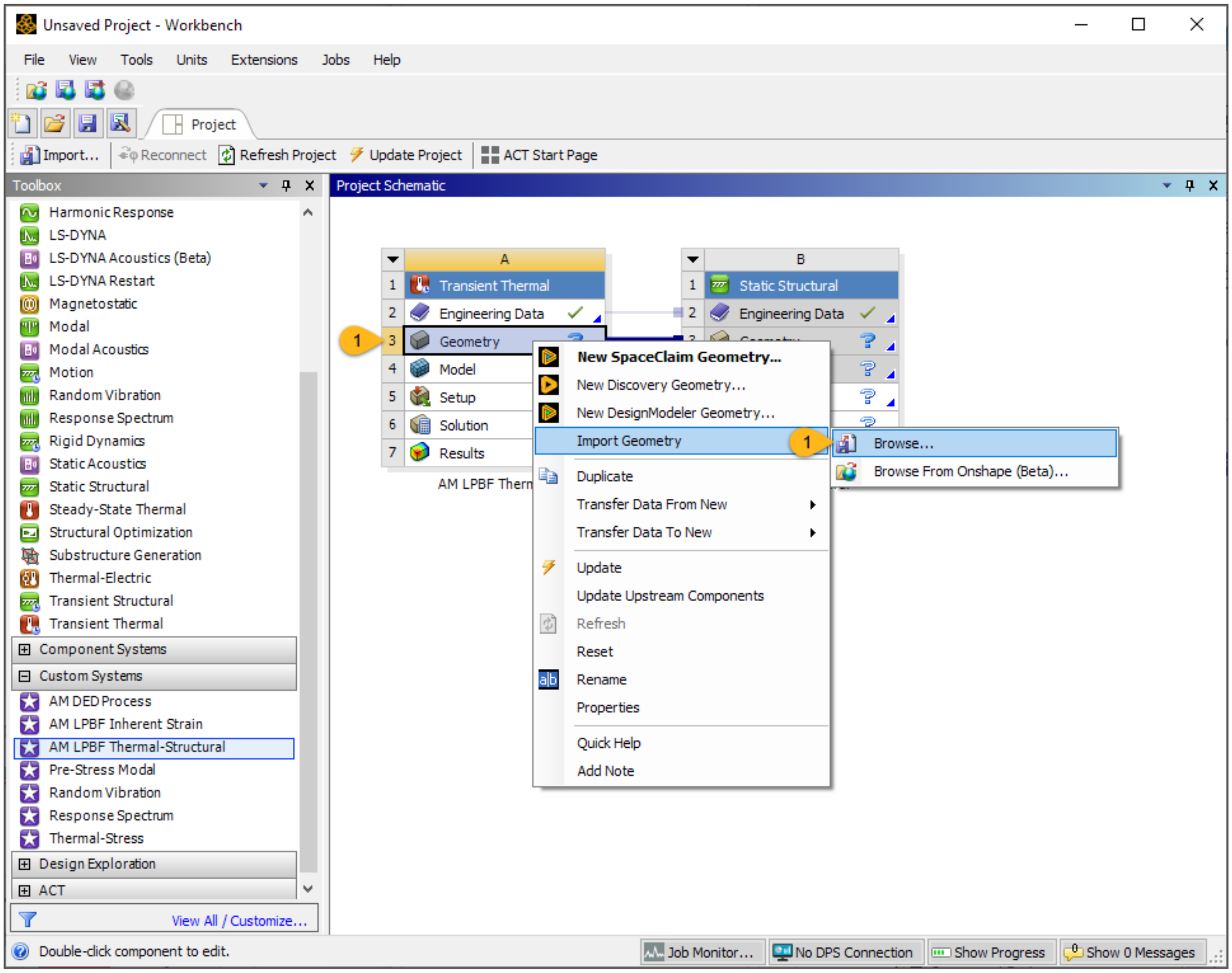


1.3. Import Geometry and Launch Mechanical

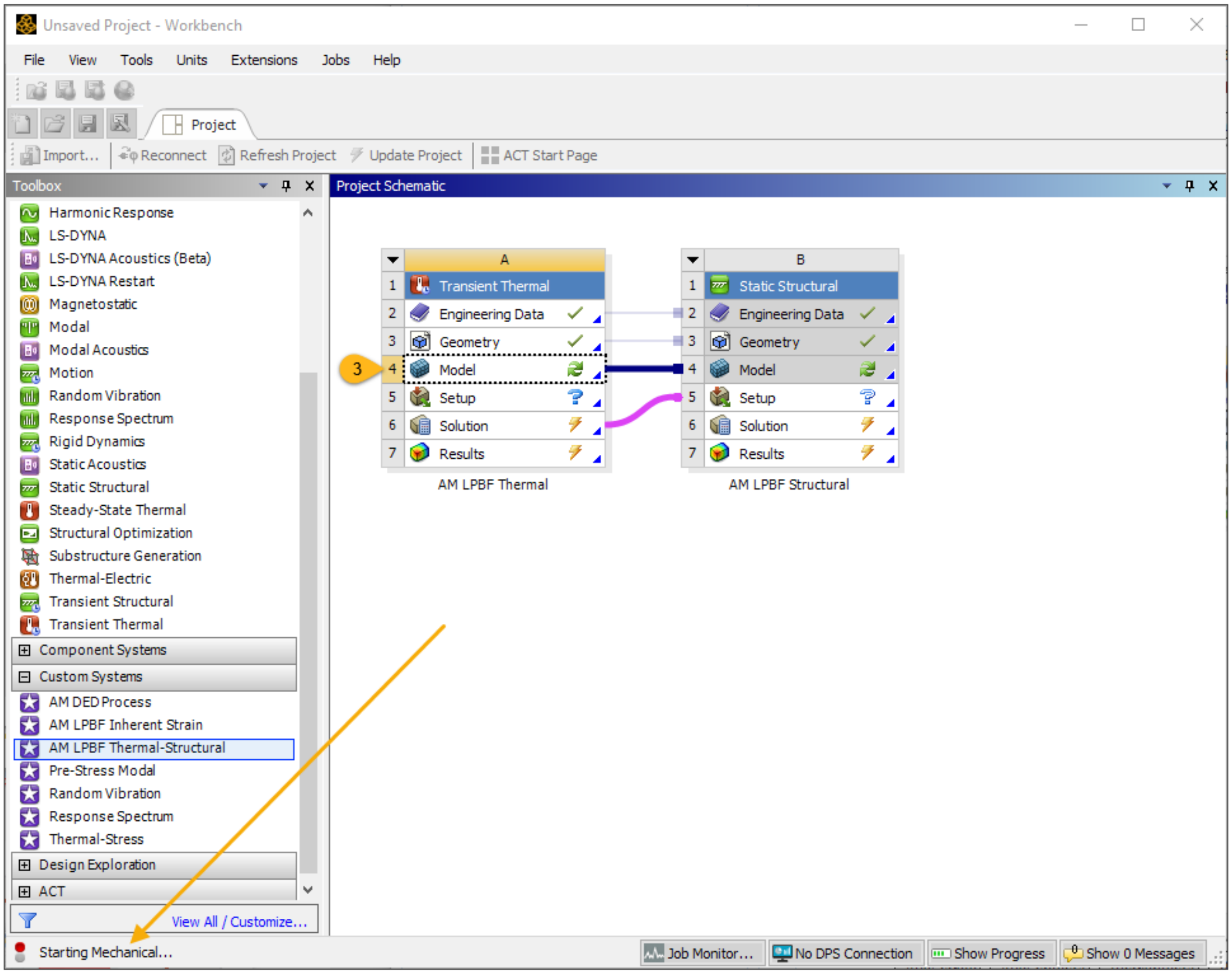
In this step, we import the geometry file in Workbench and then open the Mechanical application.

1. Right-click the **Geometry** cell in the Transient Thermal system and select **Import Geometry** > **Browse**.

2. Navigate to the double arches geometry file, select it, and click **Open** to add it to the analysis. A green checkmark appears next to the Geometry cell in the Project Schematic when the geometry is added.



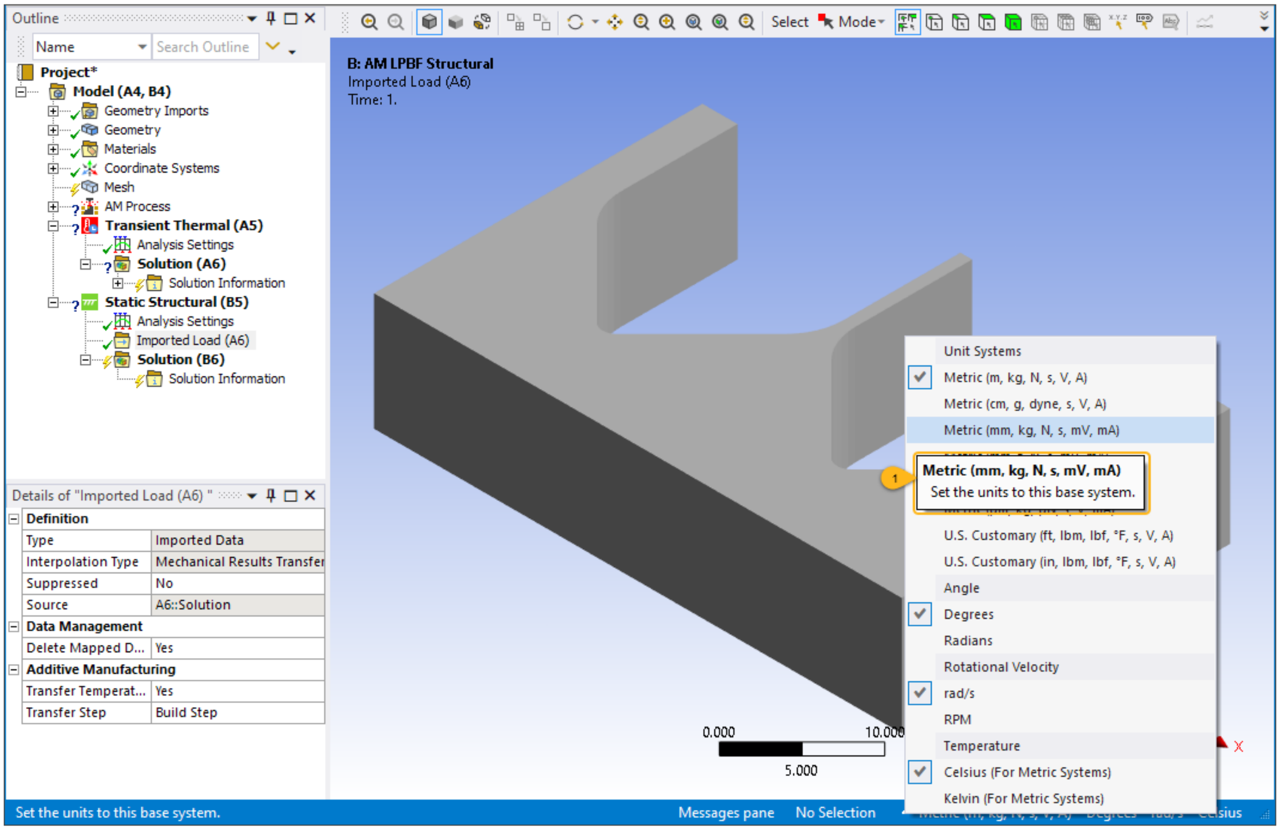
3. Double-click the **Model** cell in the Transient Thermal system to launch the Mechanical application. A message "Starting Mechanical" appears in the status bar in the bottom, left corner. It may take a few moments for the application to open and the geometry to appear.



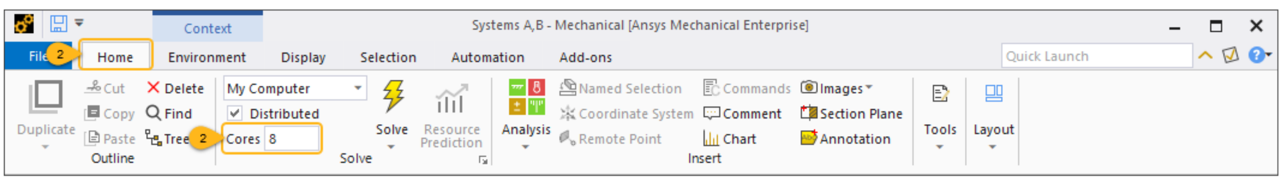
1.4. Set Units and Load the LPBF Process Add-on

In this step, we do a few things to set up for the simulation. Use your mouse buttons to pan, zoom, and rotate the model as you wish.

1. First, set the units to millimeters. In the bottom, right corner of the application UI, select the little up arrow to expand the units options and choose **Metric (mm, kg, N, s, mV, mA)**.

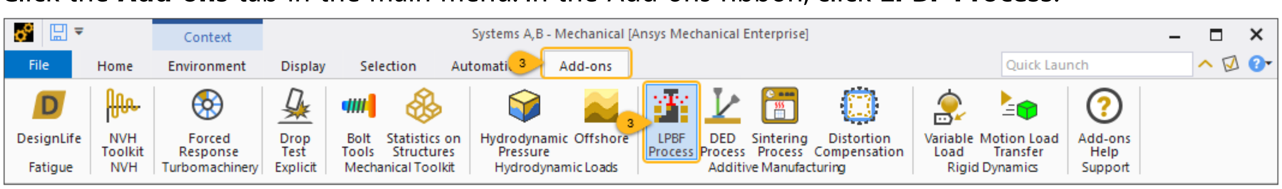


2. Next, adjust the number of processors (cores) to use on your computer. Depending on the complexity of your model, AM Process Simulations may be computer intensive. If you have an Ansys HPC license, access the option from the **Home** tab, under the Solve group and change the **Cores** to something appropriate for your simulation.

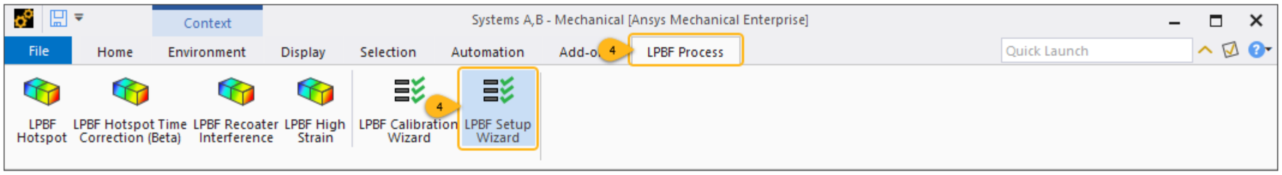


3. Next we'll load the LPBF Process Add-on. This add-on is available only if you have an Ansys Additive Suite software license with Ansys Mechanical Enterprise or one of the multiphysics bundles. If LPBF Process appears grayed out, or does not appear in the Add-ons ribbon as shown below, check your software license.

   Click the **Add-ons** tab in the main menu. In the Add-ons ribbon, click **LPBF Process**.



4. The remainder of the set-up is done using the LPBF Setup Wizard. Click the **LPBF Process** tab and then click **LPBF Setup Wizard** to open the wizard.



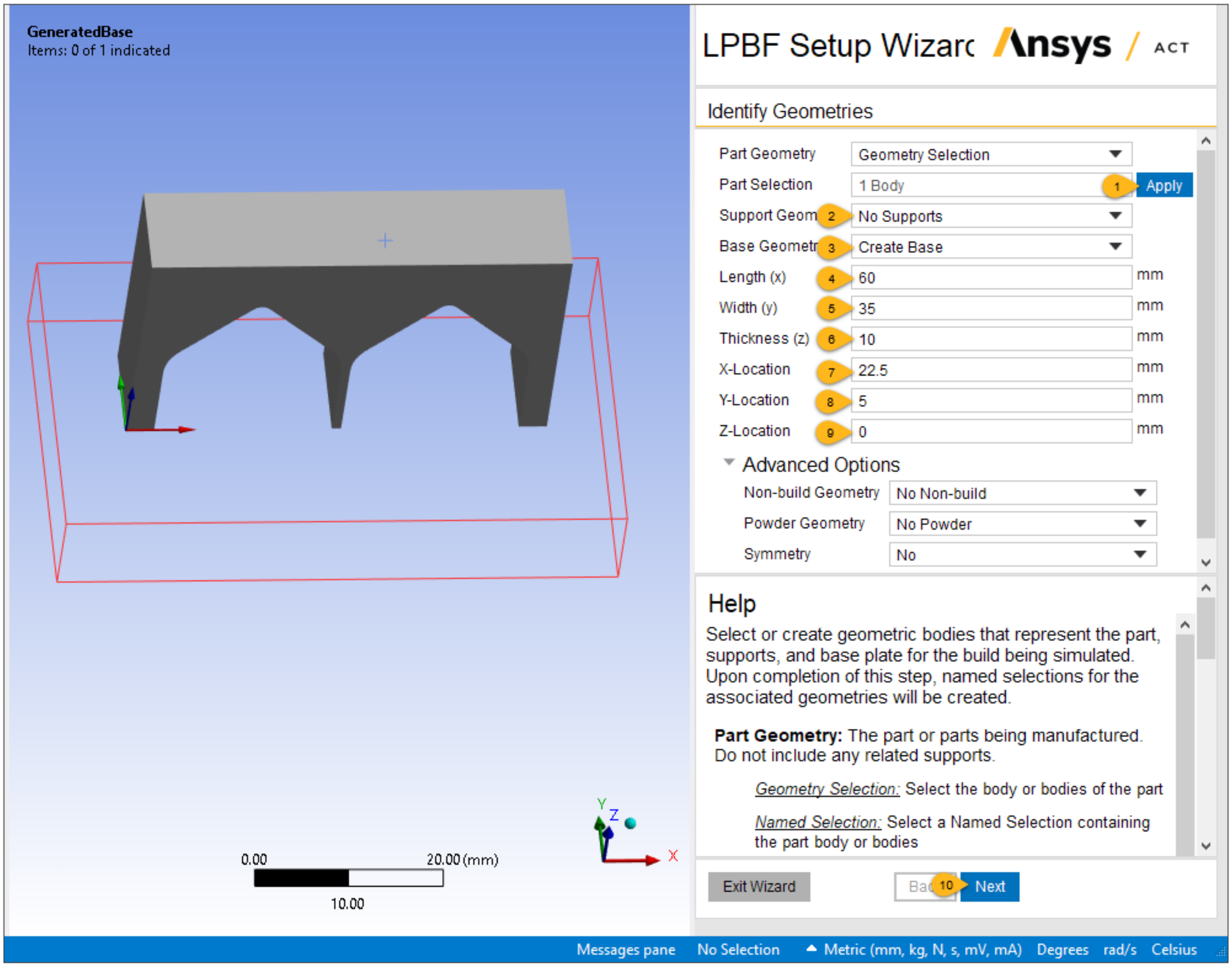
Note:

You may be wondering why we chose the LPBF Setup Wizard rather than the LPBF Calibration Wizard. Use the former when setting up a simulation for the first time, as we are doing in this tutorial. Use the latter when you already have a simulation solved and you simply need to add on a calibration step.

1.5. Wizard Step 1 - Identify Geometries

This step is to identify which body is the part, the base, and the supports. For this tutorial, there are no supports and we will create a base plate using construction geometry.

1. Select the arches body and click **Apply** in the Part Selection field. (Note the Body picker  is active by default.)
2. Under Support Geometry, choose **No Supports** because there are no supports for this model.
3. For the Base Geometry field, choose **Create Base** because there is no base plate included in our geometry file. We will define X, Y, and Z dimensions of the base and the X, Y, and Z location of the *center of the top surface* of the base.
4. Enter **60** for Length (x).
5. Enter **35** for Width (y).
6. Enter **10** for Thickness (z).
7. Enter **22.5** for X-Location.
8. Enter **5** for Y-Location.
9. Enter **0** for Z-Location.
10. Leave the remaining options set to the default of No and click **Next** at the bottom of the wizard page.



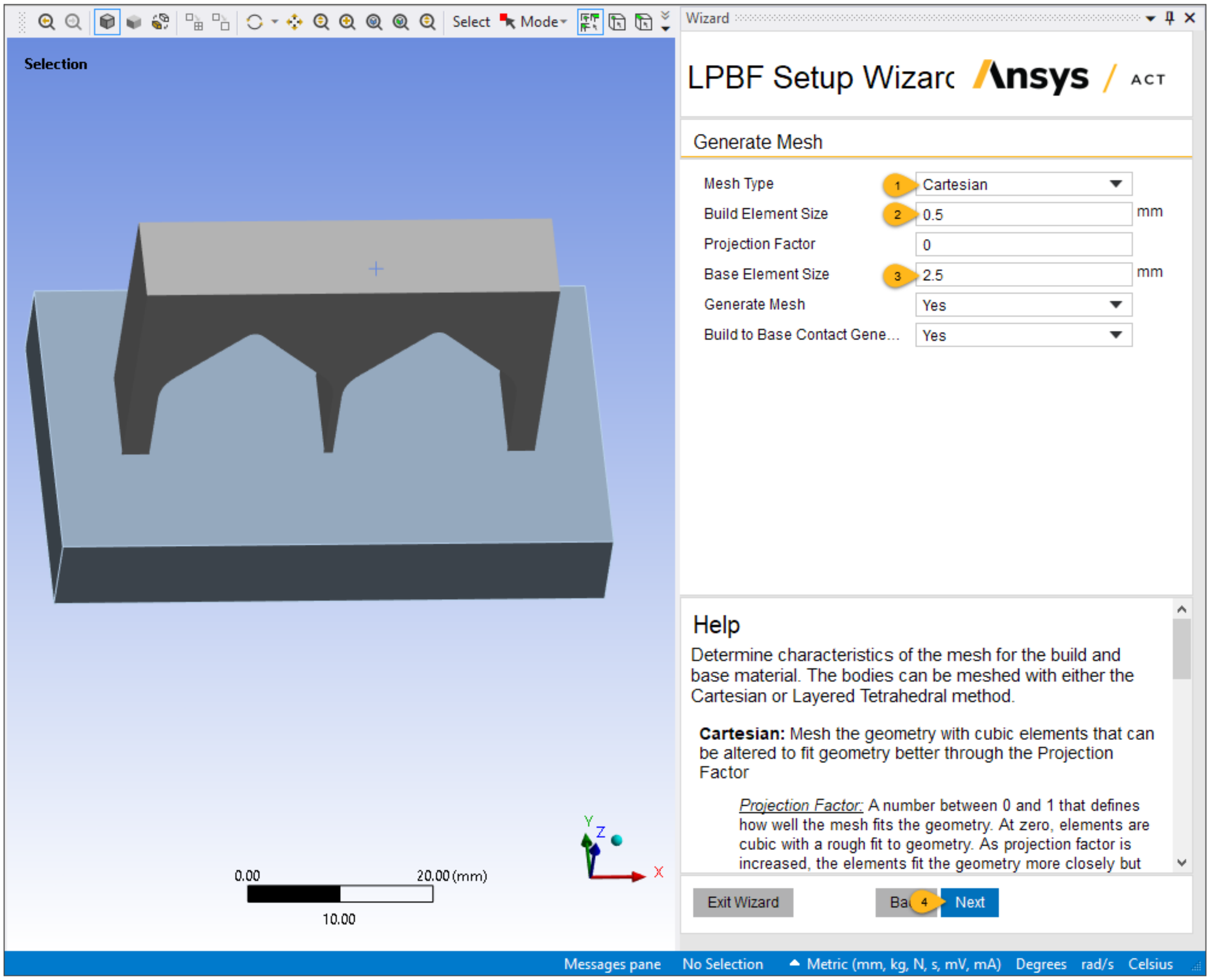
Notice that items are added and/or updated in the project tree after each step in the wizard. It is a good idea to review these items that have changed corresponding to the actions in the wizard as this is a good way to learn about Ansys Mechanical.

1.6. Wizard Step 2 - Generate Mesh

In this step, set mesh controls and generate the mesh. We will use a Cartesian mesh with the recommended element size of 0.5 mm for this part.

1. Under Mesh Type, choose **Cartesian**.
2. For Build Element Size, enter **0.5** to define a mesh size of 0.5 millimeters.

   Keep 0 (default) for Projection Factor.
3. For Base Element Size, enter **2** to allow a much coarser mesh size.
4. Leave the remaining options set to the default of Yes and click **Next** to move to the next step.



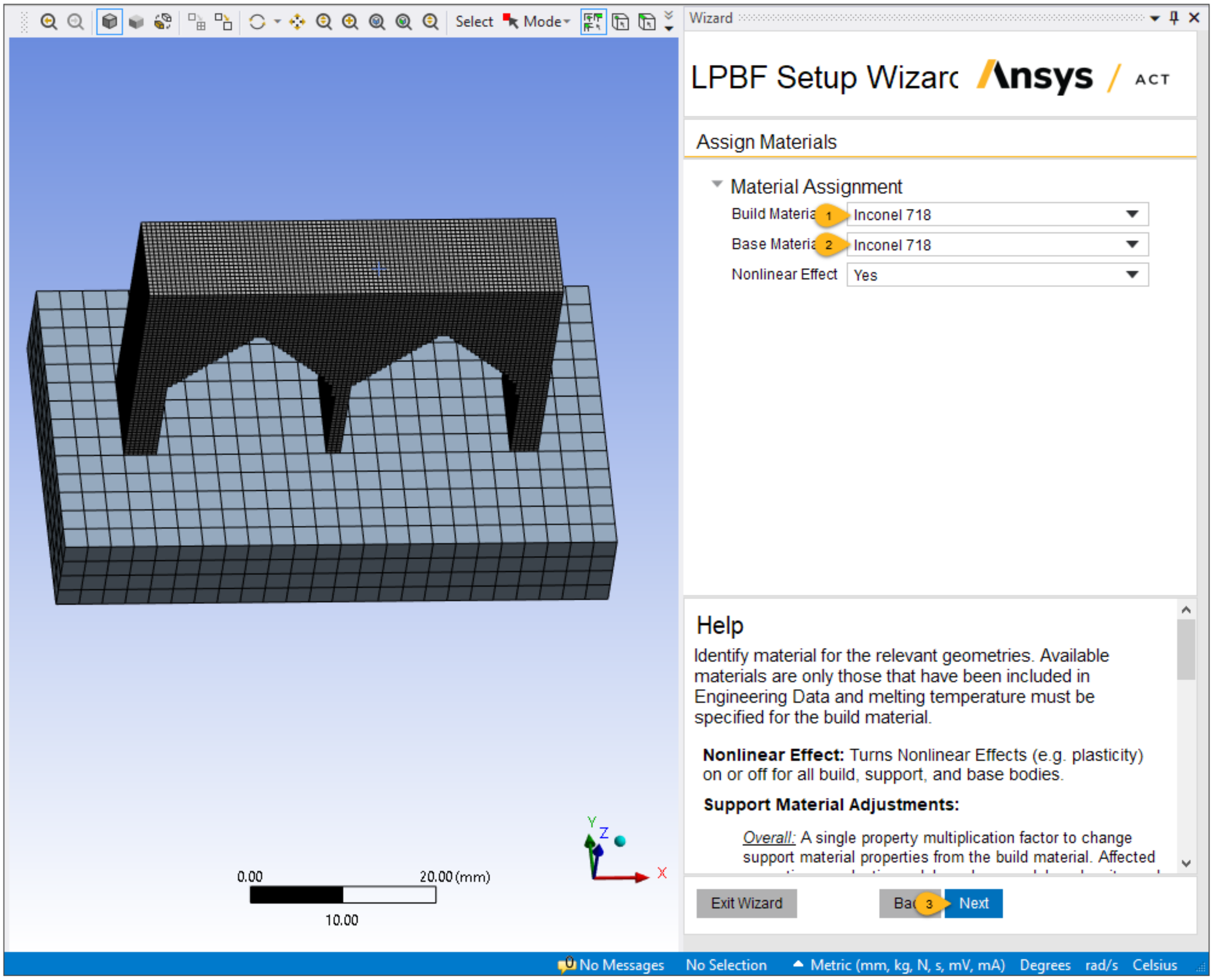
1.7. Wizard Step 3 - Assign Materials

In this step, assign materials to the build and the base.

1. Under Build Material, choose **Inconel 718**.
2. Under Base Material, choose **Inconel 718**.

   By default, bilinear isotropic hardening material properties are applied to the model when a material with those properties is applied. You can confirm this by viewing that Nonlinear Effects is set to Yes for both the part and the base. (In the project tree on the left side of the UI, expand the Geometry object so you see the two bodies. Select each body and review the Details panel for each at the bottom of the tree.)

3. Click **Next**.



1.8. Wizard Step 4 - Define Build Settings

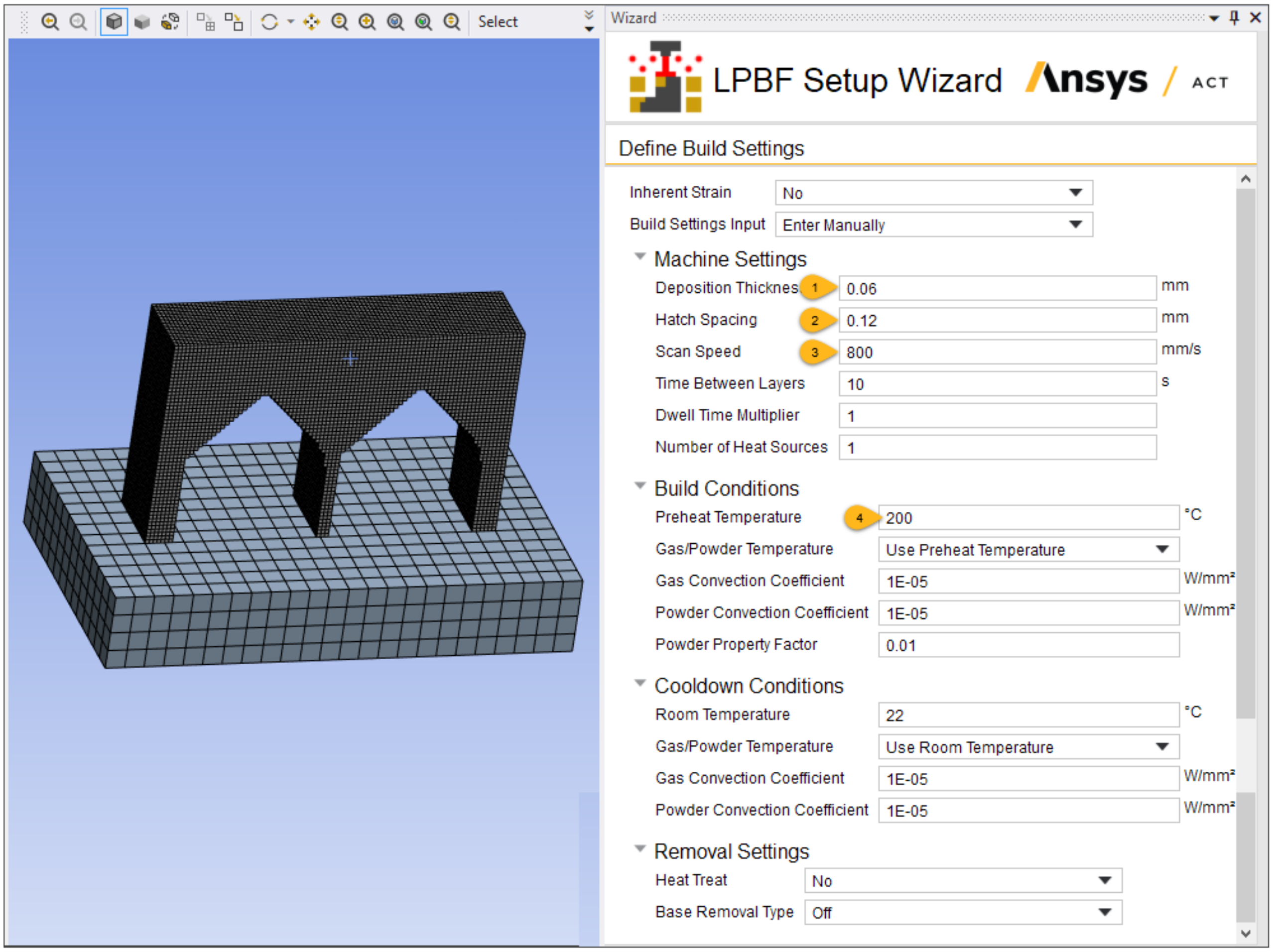
In this step, define machine settings, build conditions, and cooldown conditions.

Under **Machine Settings**:

1. Set Deposition Thickness to **0.06** mm. (This corresponds to the Layer Thickness of 60 μm.)
2. Set Hatch Spacing to **0.12** mm.
3. Set Laser Speed to **800** mm/s.

Under **Build Conditions**:

4. Set Preheat Temperature to **200** °C.
5. Leave the remaining settings to their defaults. Note that this includes removal settings. (Scroll down in the wizard panel to see all the options.) We will not simulate cut-off of the part from the base, as distortion measurements were taken on-plate in the experimental portion of the calibration. Click **Next**.



1.9. Wizard Step 5 - Define Boundary Conditions and Constraints

In this step, apply boundary conditions to the base. This includes a constant preheat temperature during the build, an ambient cooldown temperature during cooldown, and a fixed condition throughout the simulation because we assume the base is rigid with no distortion.

Under **Base Thermal Boundary Conditions**:

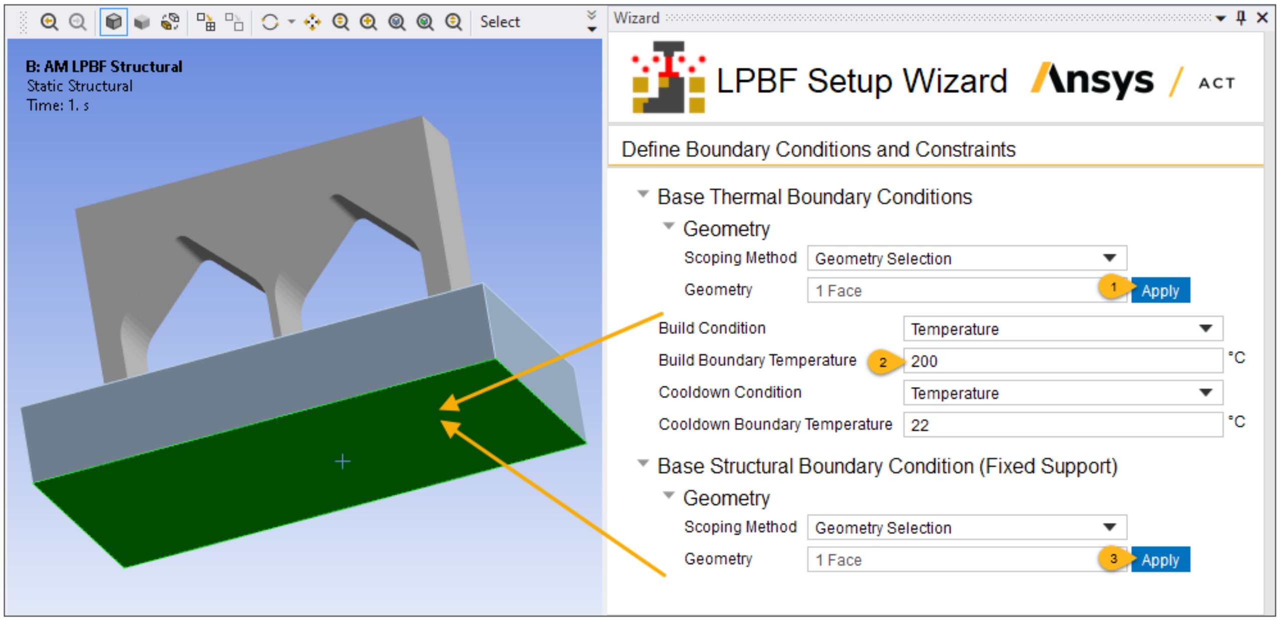
1. Select the bottom face of the base. Click **Apply** next to the Geometry input field.

2. Keeping Temperature as the Build Condition, enter **200** °C for Build Boundary Temperature. Leave the default Cooldown condition as a 22 °C temperature.

Under **Base Structural Boundary Condition**:

3. Select the bottom face of the base again and click **Apply** as the geometry for the fixed support.

4. Click **Next**.



1.10. Wizard Step 6 - Set Up Calibration

In this step, we establish a line of nodes for distortion measurements (similar to the red dashed line in the experiments), enter our distortion target, and finish setting up for the calibration iterations. The measurement line is along the Z axis located at X=0 and Y=5 mm.

1. First, we need to define a line of nodes and create a named selection for it. This is done outside of the LPBF Setup Wizard but keep the wizard open. In the project tree, right-click **Named Selection** and choose **Insert** > **Named Selection**. **Rename** the new Named Selection object to **MeasuredLine**.

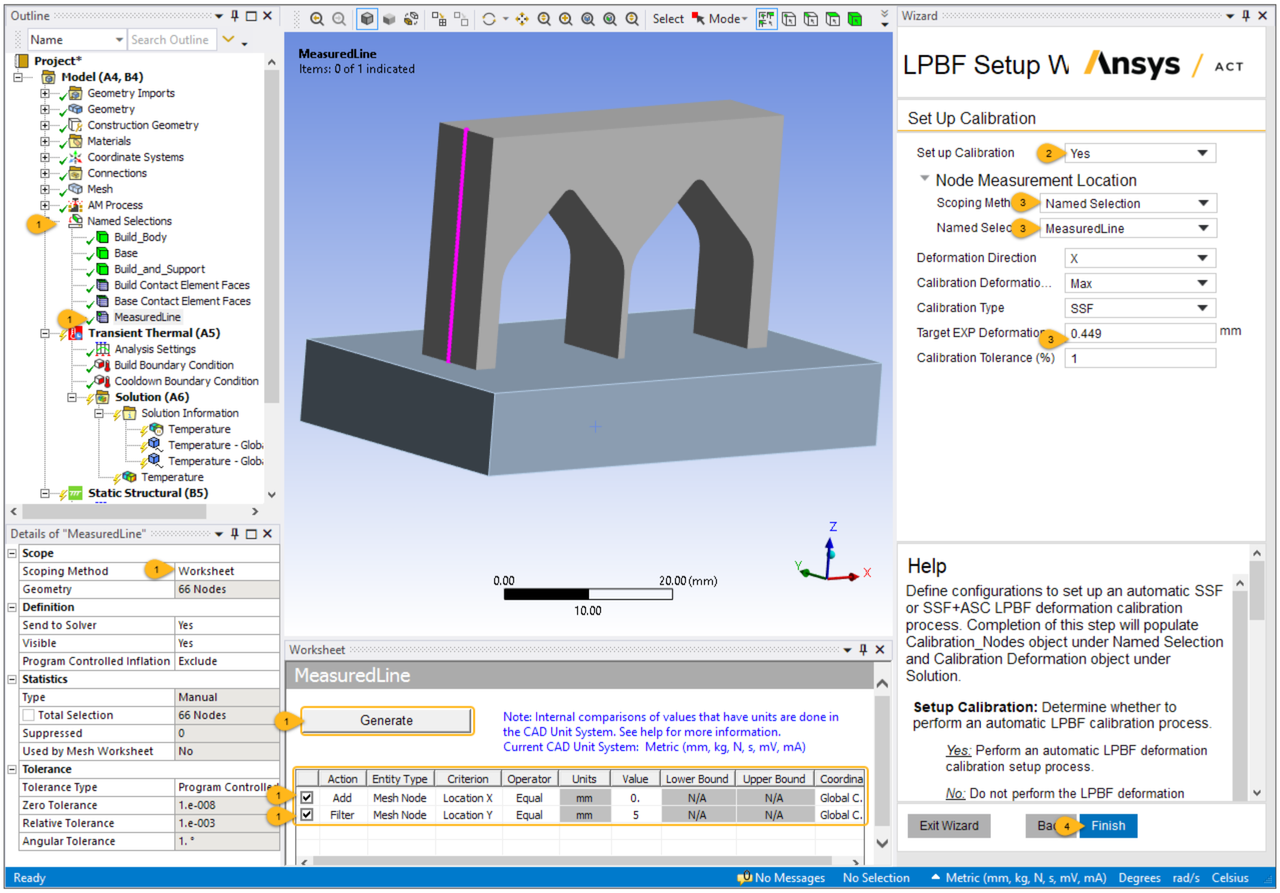
   In the Details panel of MeasuredLine, change the Scoping Method to **Worksheet**.

   Right-click in the Worksheet area and choose **Add** to add a new row. Set up the fields in the new row as shown in the following figure. Then add a new **Filter** row and set up the fields as shown. Click **Generate**. A line of highlighted nodes is shown on the vertical wall of the pillar.

2. Back in the wizard, choose **Yes** for Setup Calibration.

3. Under Measurement Location, change the Scoping Method to **Named Selection**. Choose the **MeasuredLine** named selection. Choose **X** for Distortion Direction and enter **0.449** mm for Target Distortion, which is our maximum distortion according to the experimental measurements.

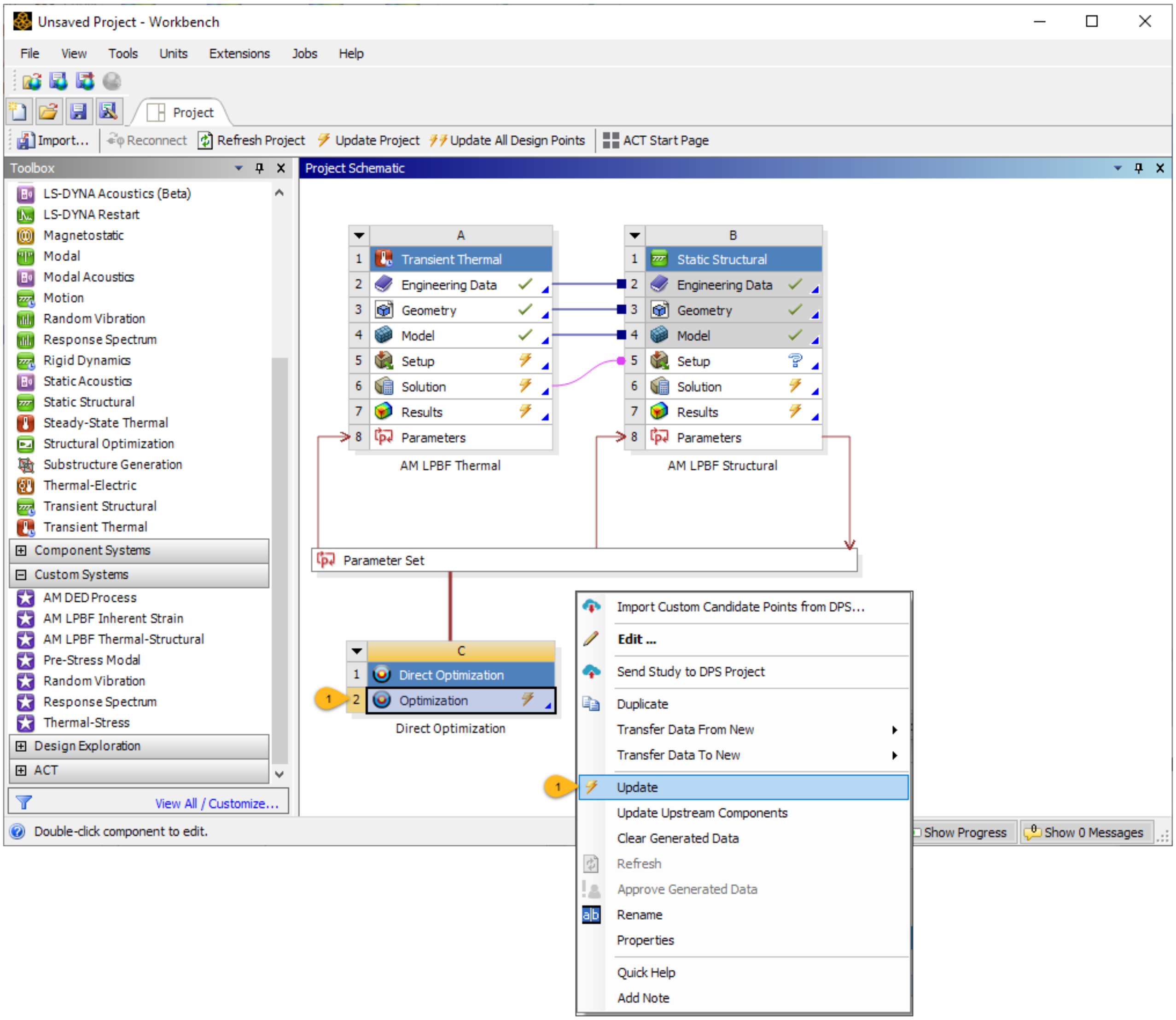
4. Click **Finish**. Finally, click the **x** in the upper, right corner to exit the wizard panel. The calibration simulation has now been set up successfully.



1.11. Run Calibration Iterations

In this step, we run the calibration iterations using the automated Direct Optimization system in Workbench.

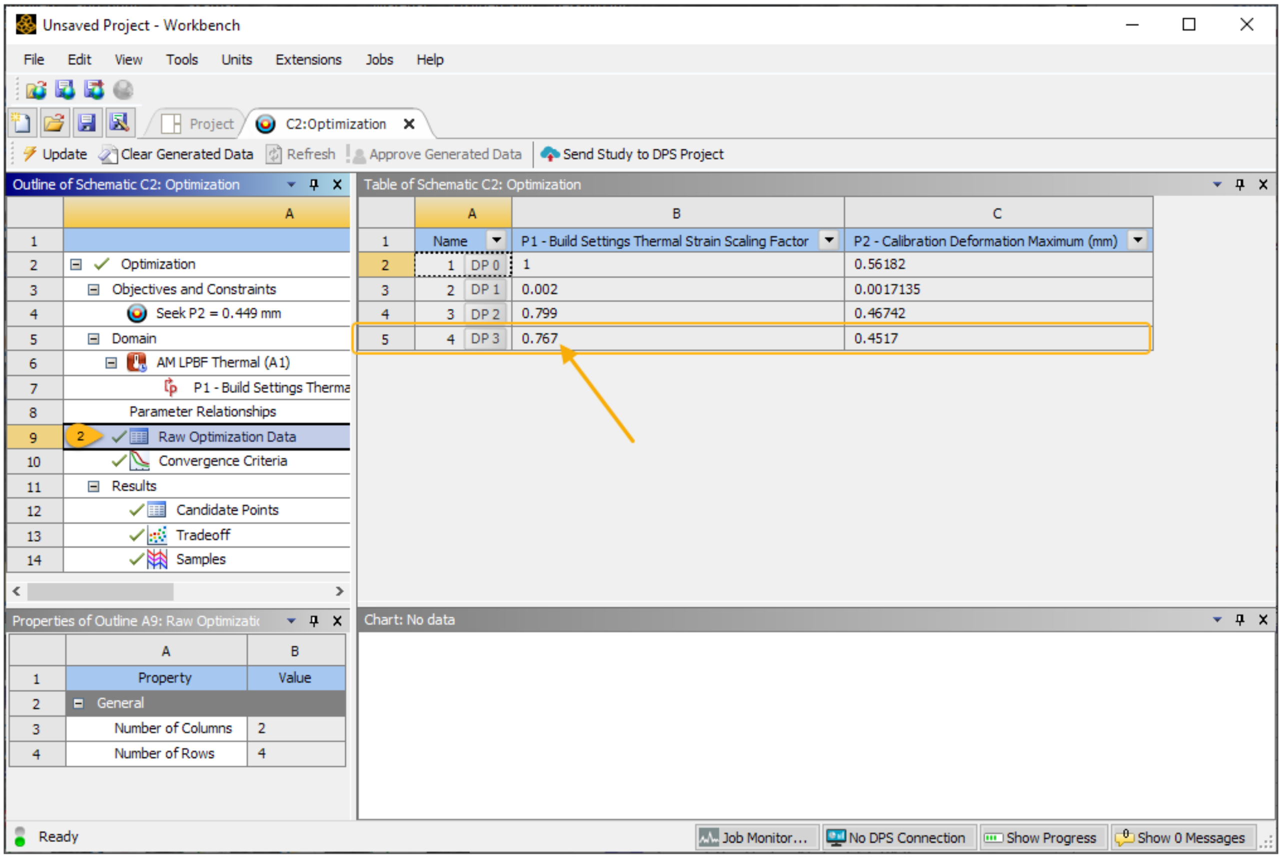
1. Go back to the Workbench UI. After completion of the calibration setup in the LPBF Setup Wizard, notice that the Direct Optimization system is automatically connected to the AM Thermal-Structural Analysis systems. In the Direct Optimization system, right-click **Optimization** > **Update** to start the calibration iterations. The optimization process will run the thermal-structural simulations iteratively by automatically applying different TSSF values starting from the default of 1 in order to search for the maximum distortion value along the defined MeasuredLine that best matches the experimental distortion value. The simulations may take a few hours to complete depending on the number of cores specified and the machine used to run this model.



1.12. Obtain the Optimum Calibration TSSF

1. Double-click the **Optimization** cell to open the Direct Optimization module.

2. Green check marks are shown under Results on the left hand side of the menu when the optimization is done. The last row shows the optimized calibration TSSF. In our example, the calibrated TSSF is 0.767, which results in a distortion of 0.4517 mm. This is within our 1% tolerance.



Congratulations! You have completed the tutorial.

Chapter 2: Additive Print Calibration - 4 Pillars Geometry

This example demonstrates the workflow of performing an Assumed Strain, Linear Elastic calibration simulation in Ansys Additive. The strain scaling factor (SSF) is obtained from this calibration process for an AM build using Ti-6Al-4V alloy.

The following features and capabilities are used:

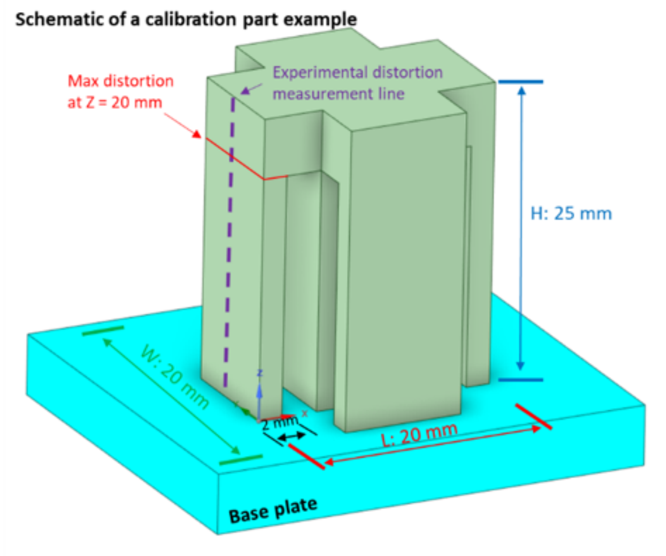
- Assumed Strain simulation type
- Ti64 material, Linear Elastic stress mode
- Calibration spreadsheet

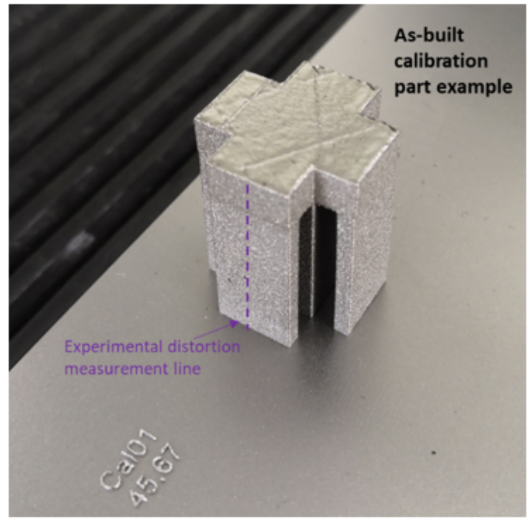
Tutorial steps:

2.1. Problem Description

This example part is used to demonstrate the overall calibration process in Ansys Additive. You may want to design/choose the most appropriate calibration geometry for your own scenario by referring to Determine Your Calibration Part in the *Additive Print and Science Calibration Guide*.

The 4-pillars geometry is a simple, symmetrical geometry with four 2-mm thick rectangular pillars attached to a center cross beam. The overall dimensions of this part are 20 mm by 20 mm by 25 mm along the X, Y, and Z directions respectively. The part is designed in such a way that a major distortion can be measured near the overhang interface at Z=20 mm, as shown at the red arrow in the following schematic. After fabrication, the on-plate distortion of the part in the as-built condition is measured along the center line of the left pillar as highlighted by the purple dashed line. This measurement line is located at X=0 and Y=5 mm. The picture shows the actual as-built calibration part example on plate after fabrication.





The calibration part was fabricated using a laser powder bed fusion process with the machine process parameter set as shown in the following table. Some of these processing parameters are used as inputs in the simulation. The part was built with Ti-6Al-4V material directly on a build plate with no support structures.

Ti-6Al-4V Processing Parameters	
Layer Thickness (μm)	60
Preheat Temperature (°C)	200
Inert gas	Ar
Beam Diameter (μm)	85
Laser Power (W)	350
Laser Speed (mm/s)	1100
Hatch Distance (mm)	0.12
Stripe Length (mm)	10
Starting Angle (°)	45
Rotation Angle (°)	67
Energy Density (J/mm^3)	44

The maximum as-built on-plate distortion for this build was 0.242 mm in shrinkage near Z = 20 mm located along the measurement line shown in the schematic, at coordinates (0,10,20). This distortion value will be used as the target value for the simulation.

Tutorial Files

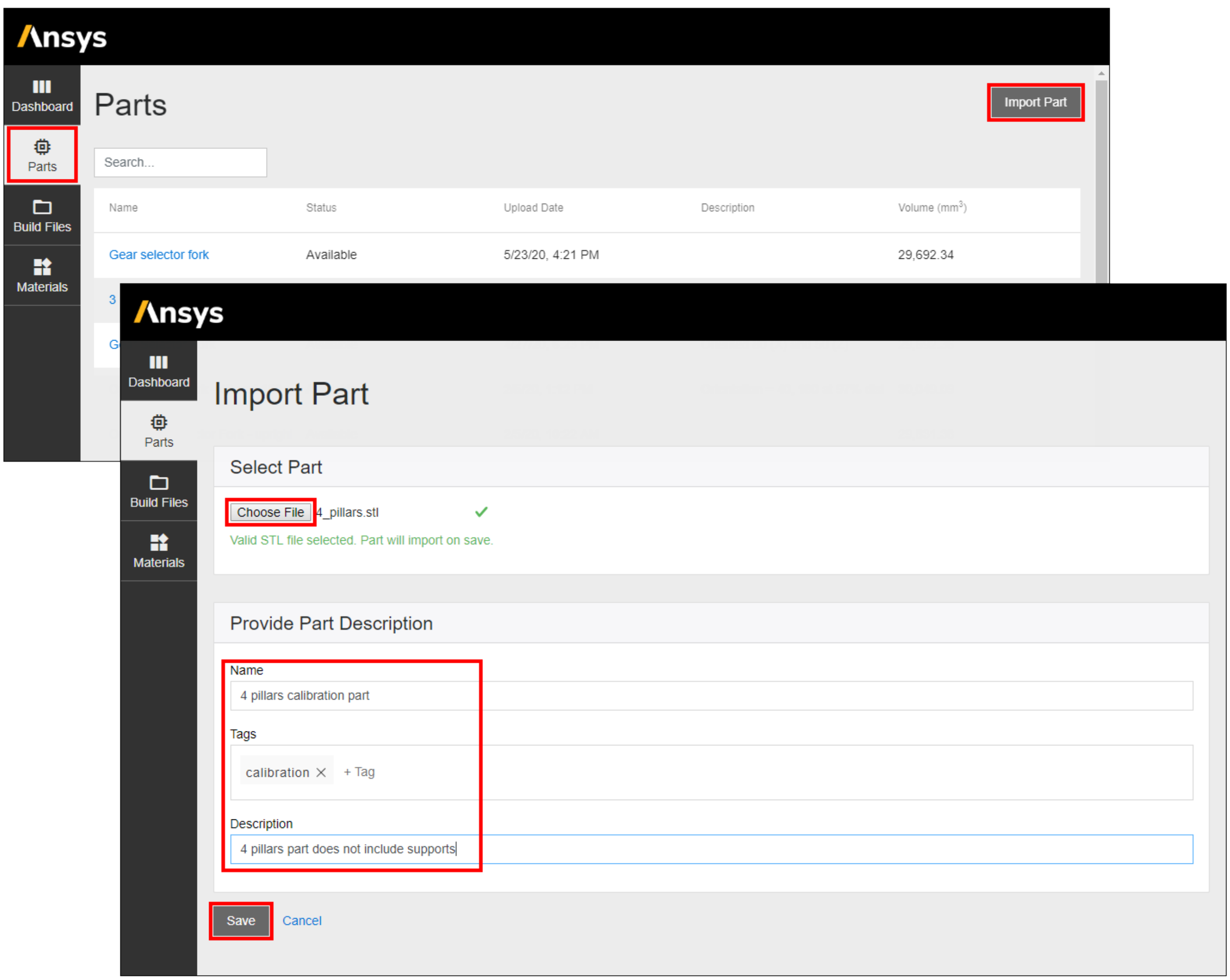
Click here to download the following:

- `4_pillars.stl` — Geometry file of the 4-pillars calibration part. There are no supports for this model and a base plate is not needed in Ansys Additive.

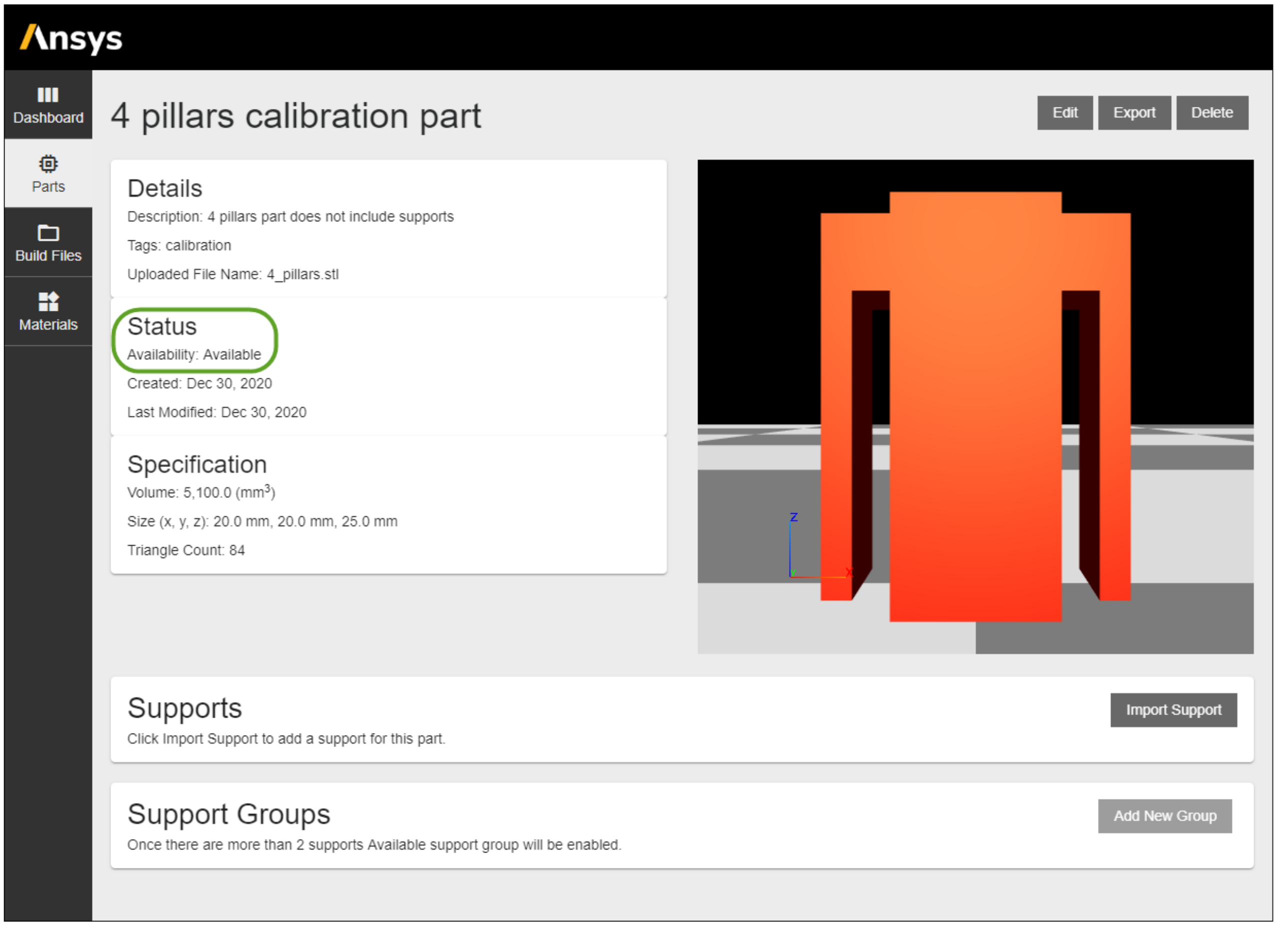
- `Additive_Print_Calibration_Spreadsheet.xlxs` — Custom spreadsheet with embedded calibration calculations.

2.2. Import Geometry

1. Open Ansys Additive.
2. Click **Parts** to bring up the Parts Library.
3. When the Parts Library is loaded, click the **Import Part** button in the upper, right corner.
4. In the Import Part page, click **Choose File** and navigate to the `4_pillars.stl` geometry, select it, and click **Open**. Parts must be in .stl format and dimensions must be in units of millimeters. A green check mark appears on the page indicating a valid part has been chosen.
5. Enter a **Name** for the part and, optionally, tags and a description.
6. Click **Save**.

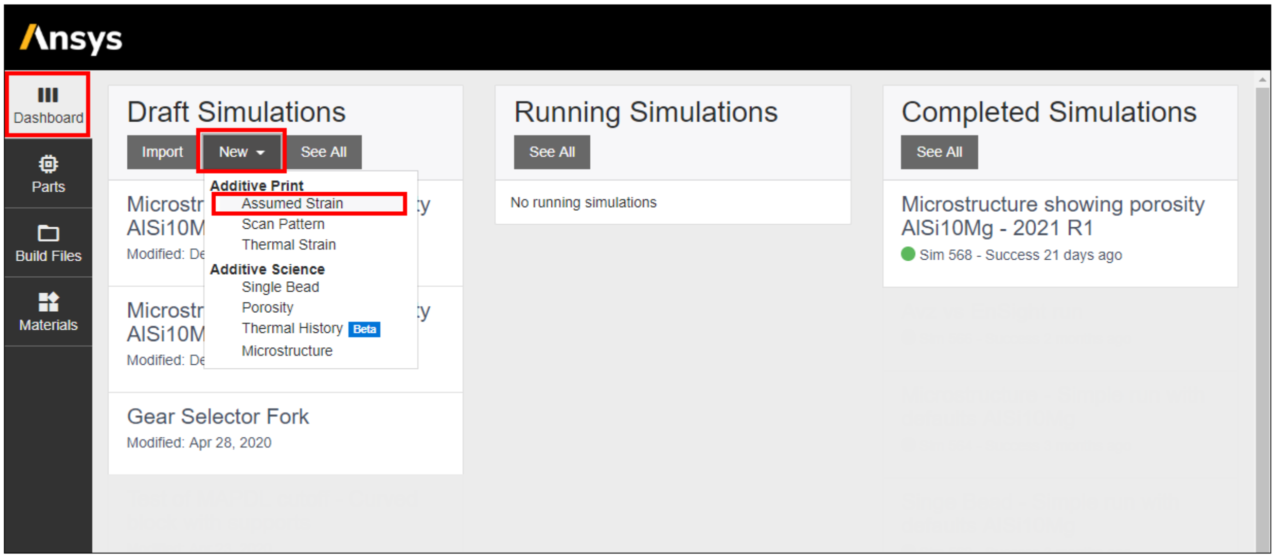


The import operation occurs when we hit Save. It may take several minutes to import, depending on the size of the part. The part is available to be used in a simulation when the availability status says "Available" on the part's details page.



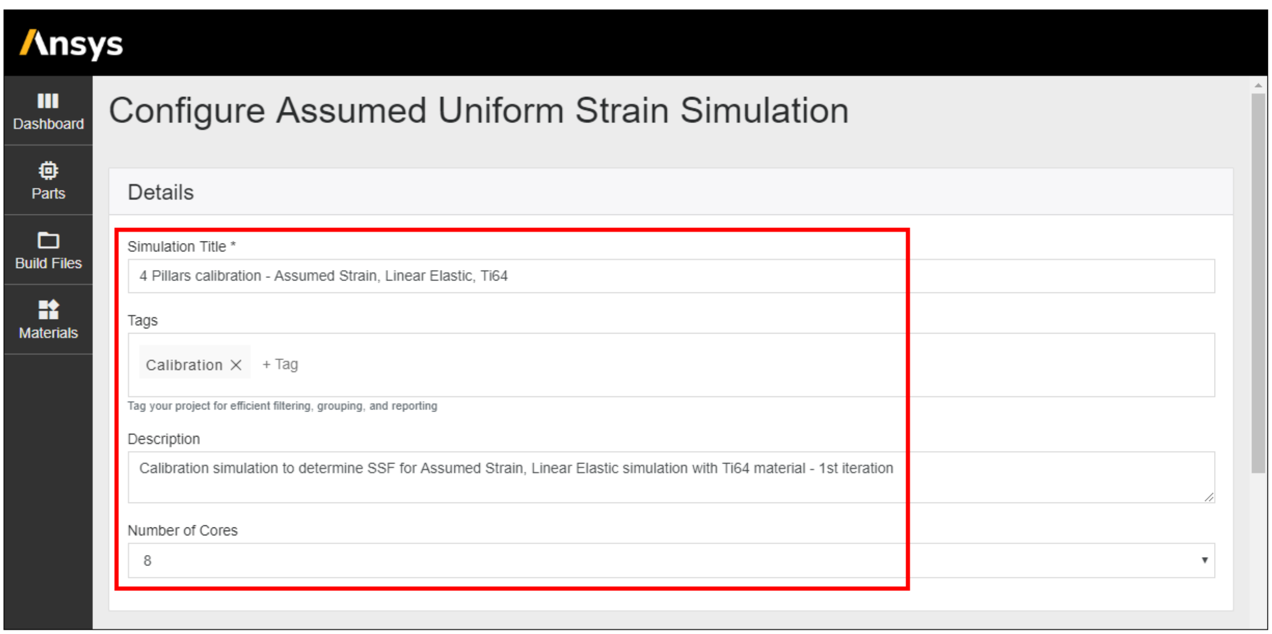
2.3. Set Up Simulation

1. Click **Dashboard**, and under **New**, choose **Assumed Strain**. This will bring up a new Assumed Strain simulation form.



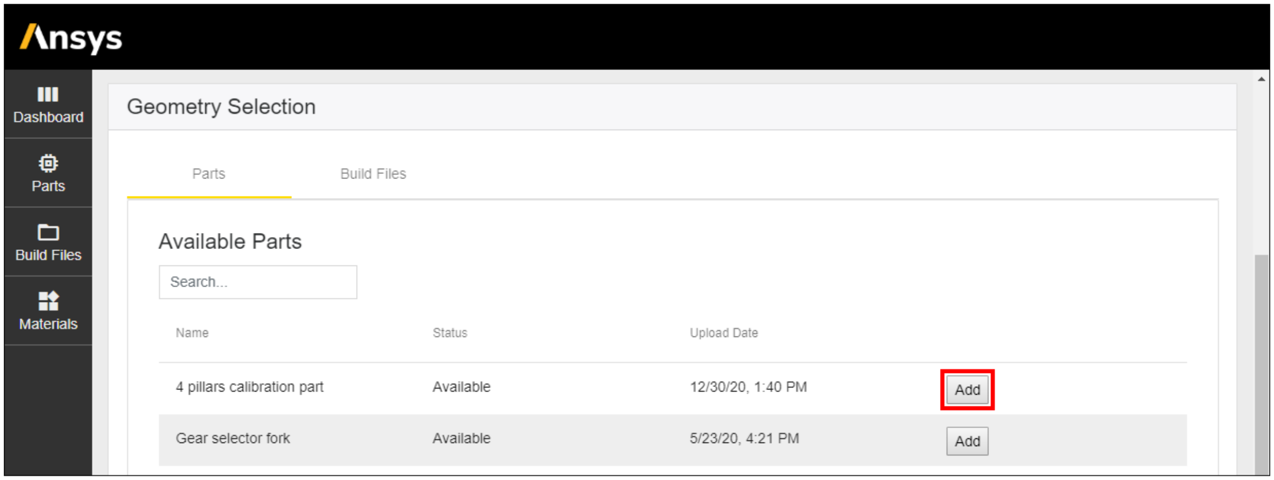
2. Enter a **Simulation Title** and, optionally, tags and a description for the simulation. We have found it is good practice to use a detailed description for record-keeping purposes. In this example we added "1st iteration" to the description because we anticipate additional iterations when performing a calibration.

3. Depending on your Additive license, change the **Number of Cores**, as desired.



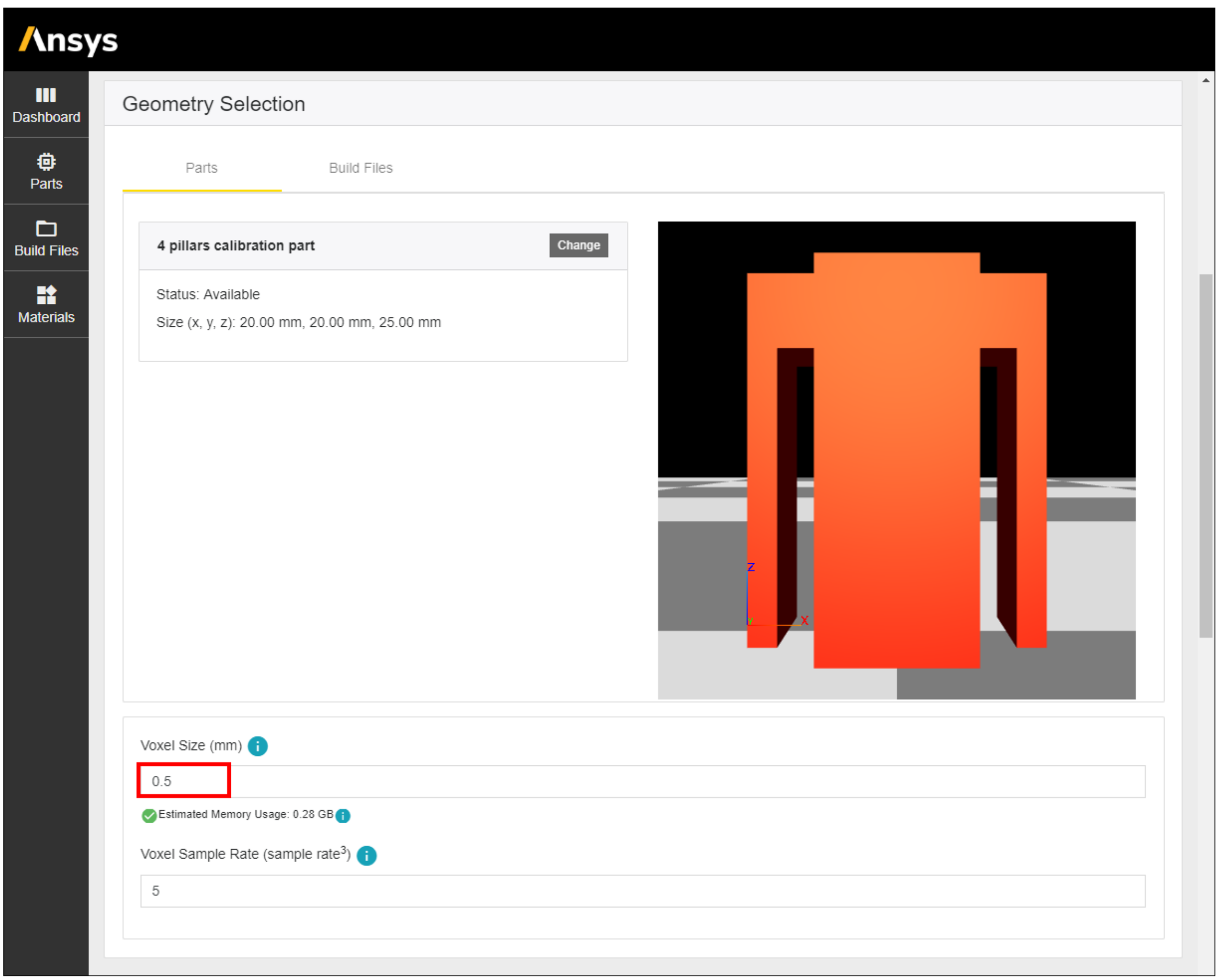
2.4. Select Geometry and Supports

1. Scroll to the Geometry Selection section and choose the 4-pillars calibration part we just imported and click **Add**.

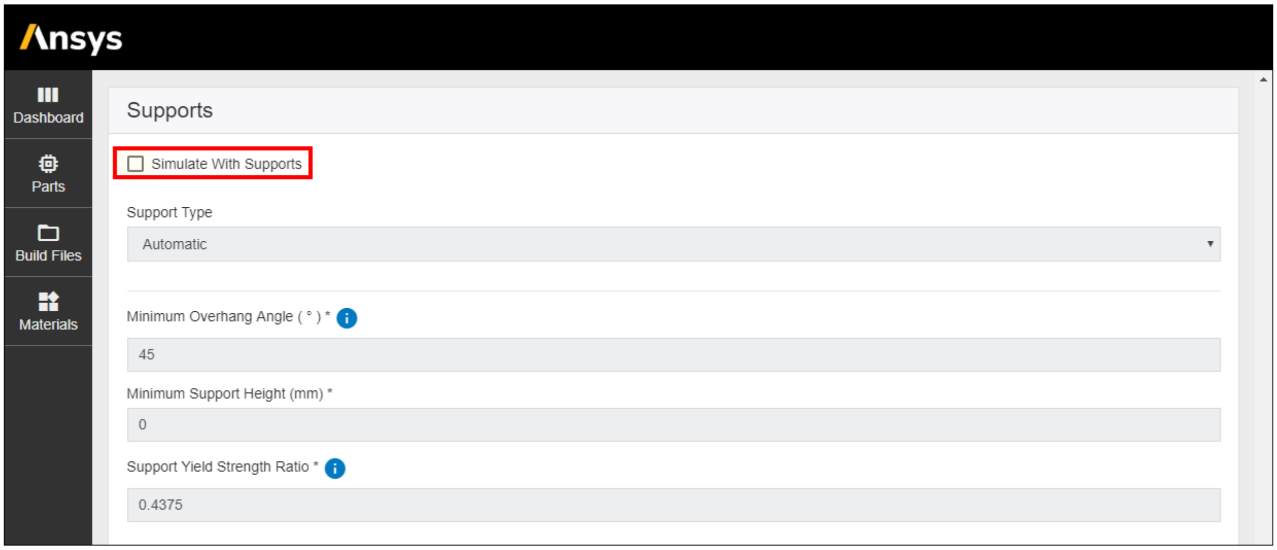


2. For Voxel Size, enter **0.5** to define a mesh size of 0.5 millimeters.

   A voxel size of 0.5 mm for this geometry is larger than we recommend but is used here to speed up the tutorial. A much smaller voxel size of 0.2 mm is more realistic for this geometry.



3. Scroll to the Supports section and **uncheck Simulate with Supports** since there are no support structures for this model.

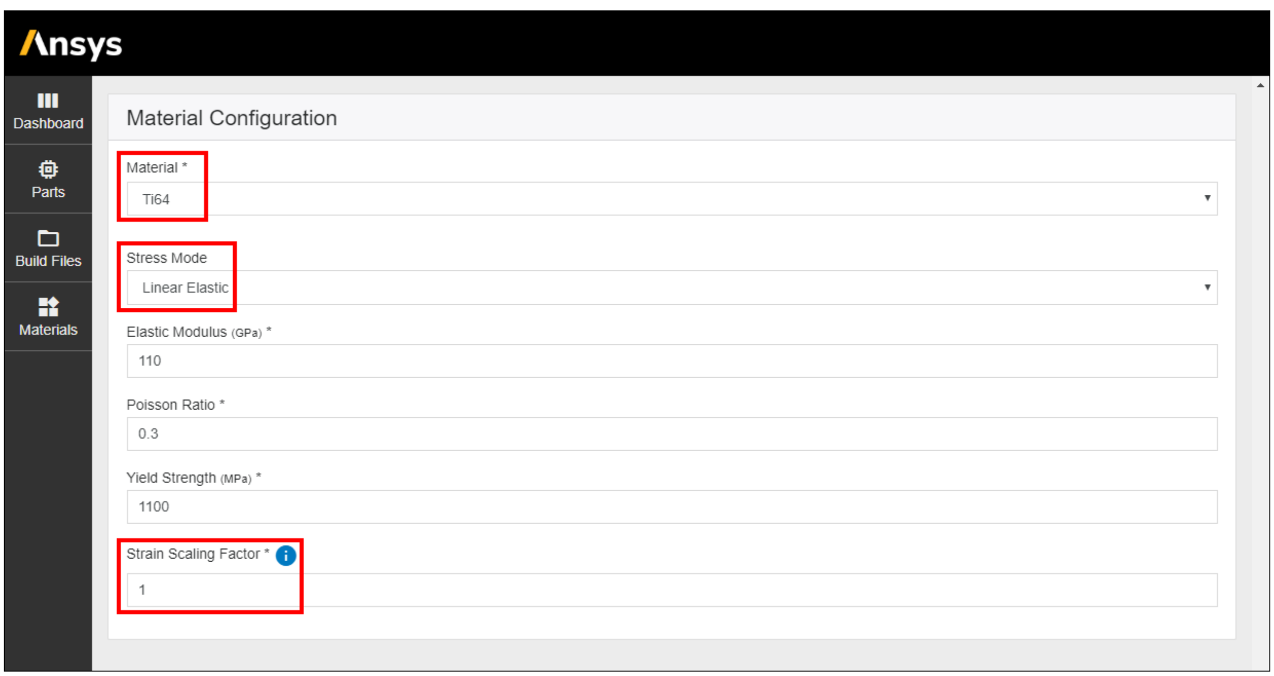


2.5. Configure Materials

1. Scroll to the Material Configuration section and select **Ti64** from the drop-down list.

2. Under Stress Mode, choose **Linear Elastic**.

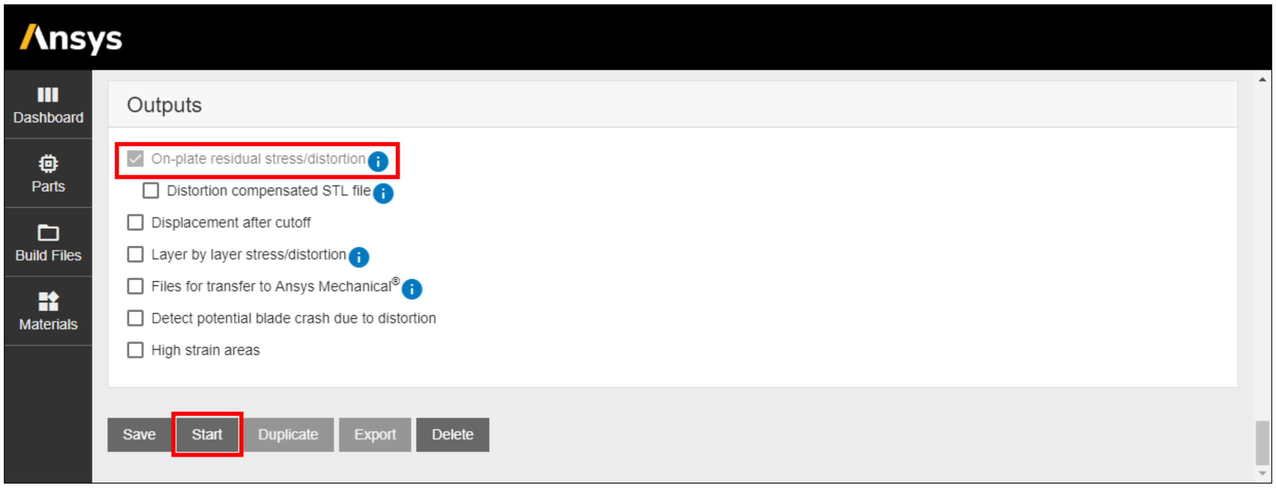
3. Use the default Strain Scaling Factor of **1** for this first calibration iteration. We will be changing this value for subsequent iterations.



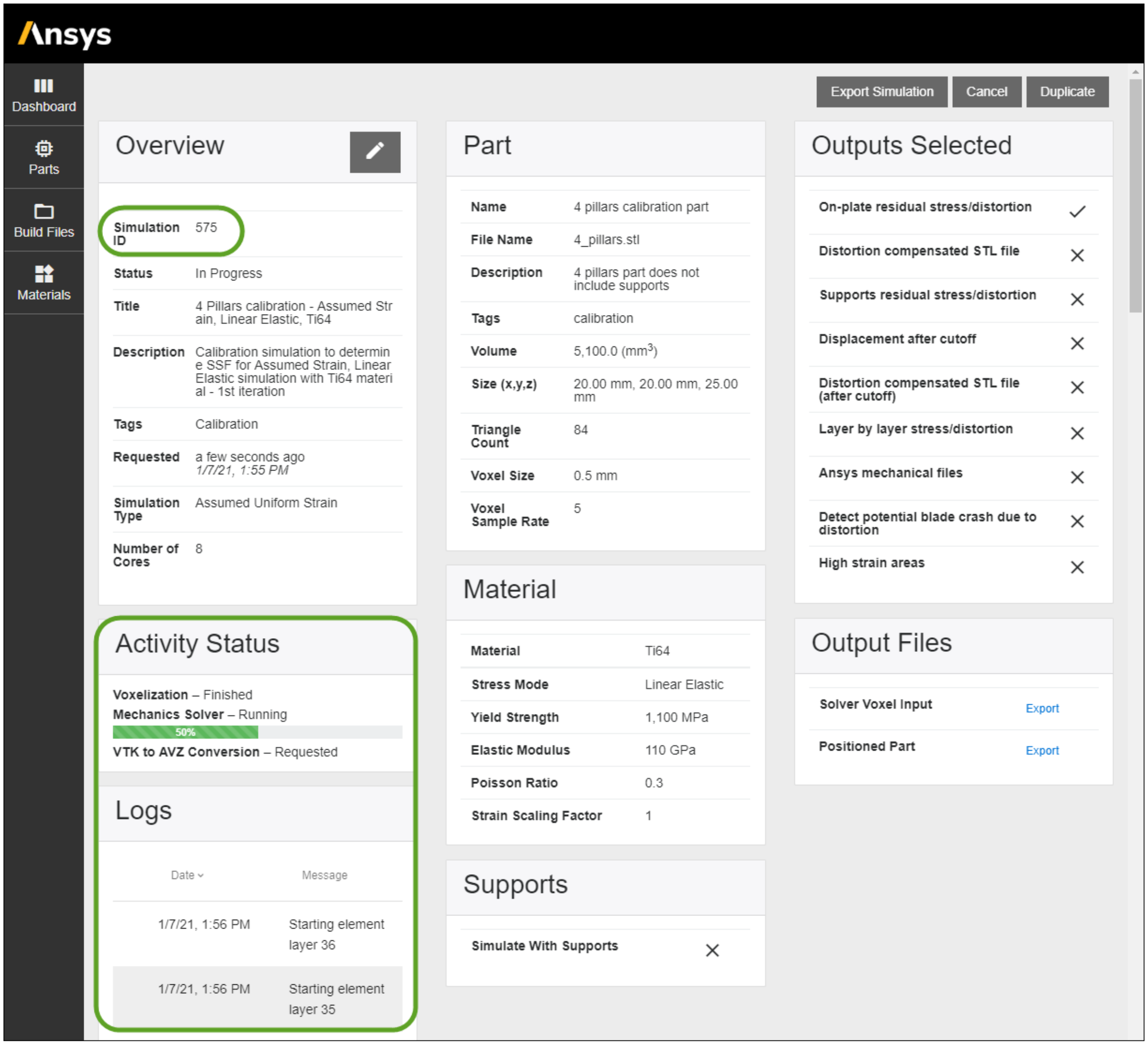
2.6. Select Outputs and Solve

1. We do not need to add any extra output items so we will simply use the default on-plate residual stress/distortion results.

2. Click **Start** to begin the first calibration iteration.



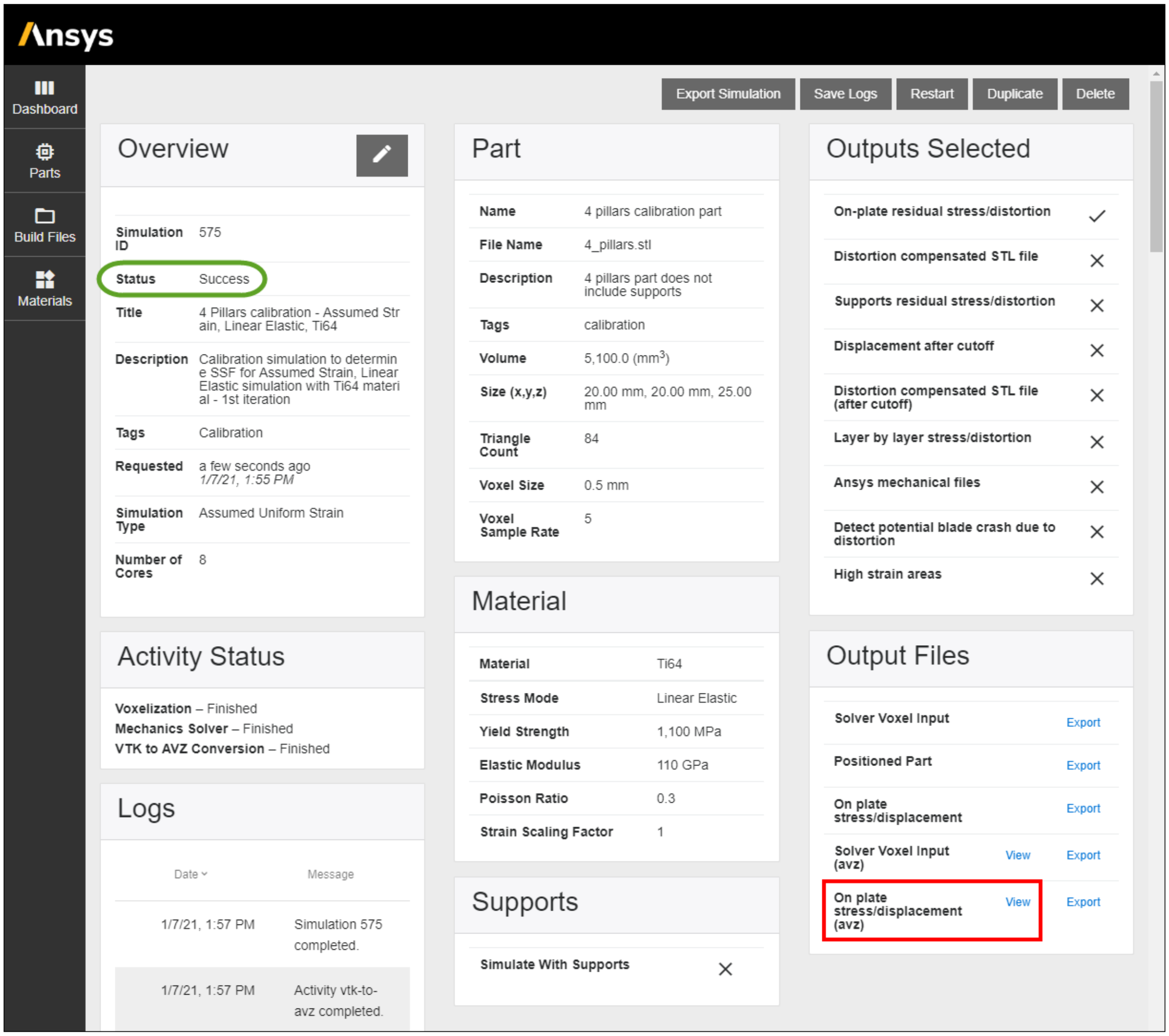
While the simulation is running, review the activity status and logs for helpful information. The Simulation ID is a unique identification number given to this one simulation. Output files will include this ID in their file names.



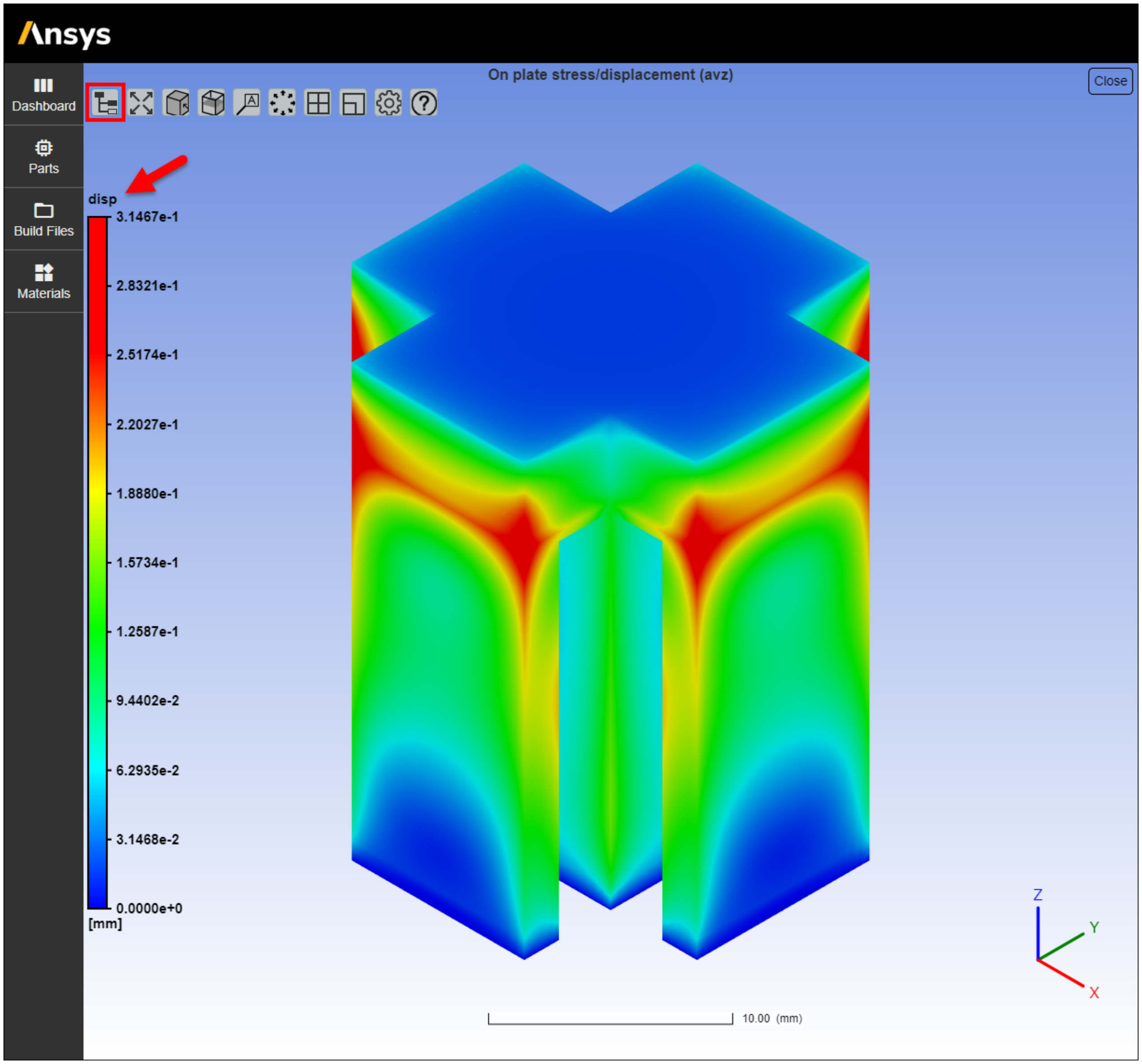
2.7. Obtain the Distortion

1. Once the simulation is complete, output files are available for review. First we'll look at the results in the Additive application.

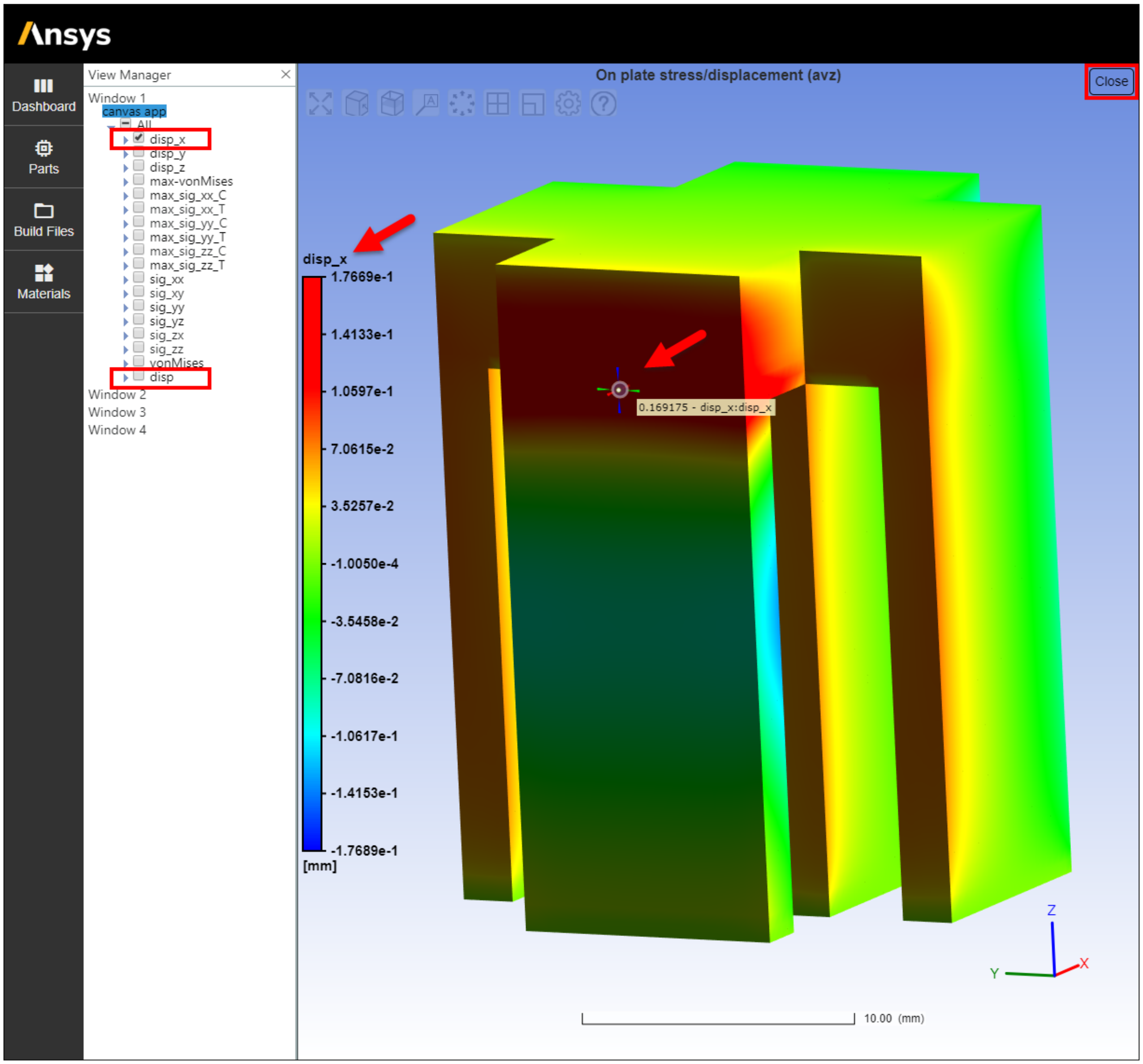
   a. Click **View** next to On plate stress/displacement (avz) to bring up the viewer.



b. The result item displayed by default is the total displacement vector. We need to change the display so that we are looking at the X component of displacement. Click the **View Manager** button in the viewer controls bar.

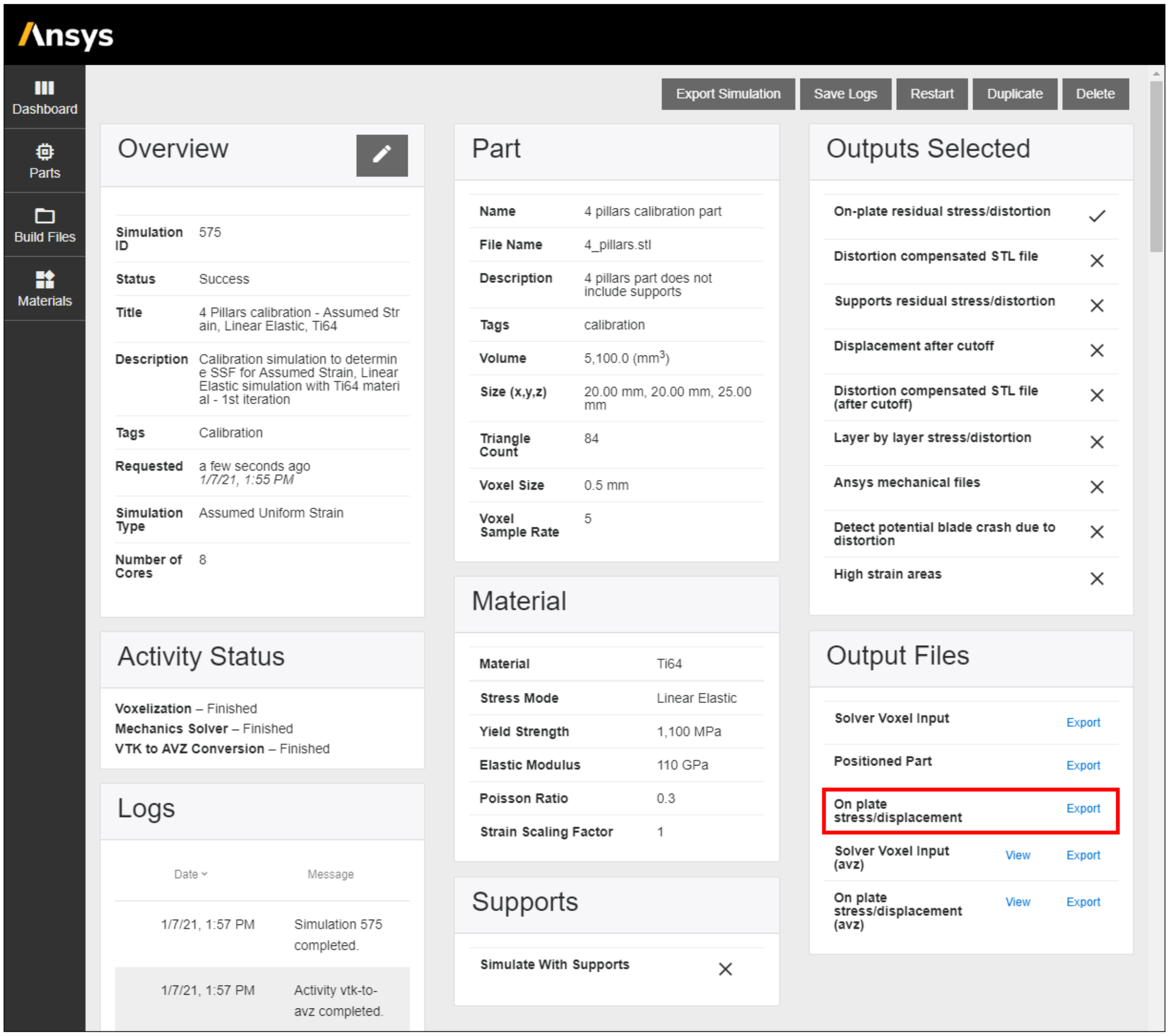


c. In the View Manager drop-down, check the box next to **disp_x** and clear the **disp** check box. Rotate the model so that the pillar with the greatest shrinkage distortion (red contour) is shown. Move the mouse around within this area to see the X-displacement. Click **Close** in the upper right corner to get back to the simulation results page.



2. Since it is a bit tricky to find the maximum displacement and to know its exact location, we will view the results in Ansys EnSight.

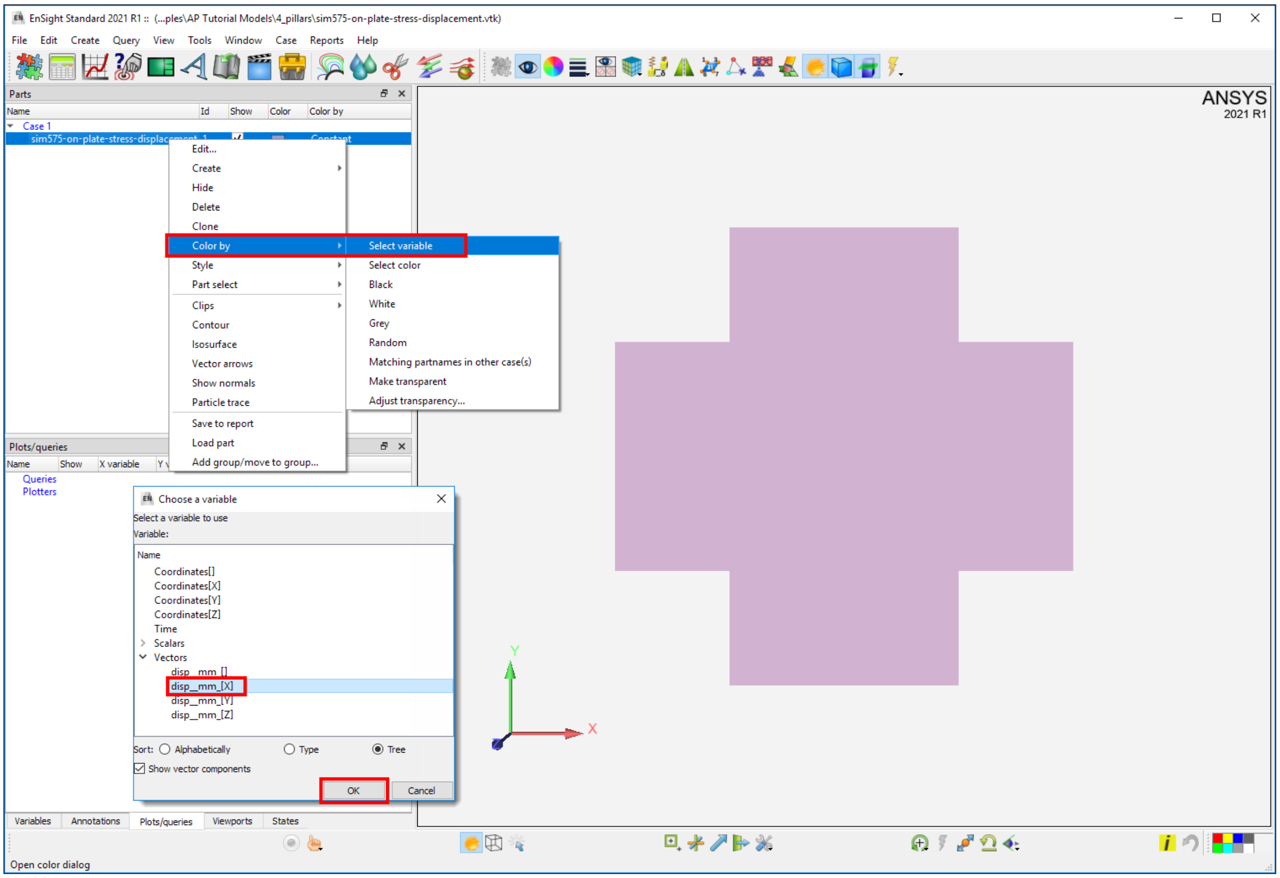
   a. Start by exporting the results to a .vtk file. Click **Export** next to On plate stress/displacement and then **Save** the file.



b. Open Ansys EnSight.

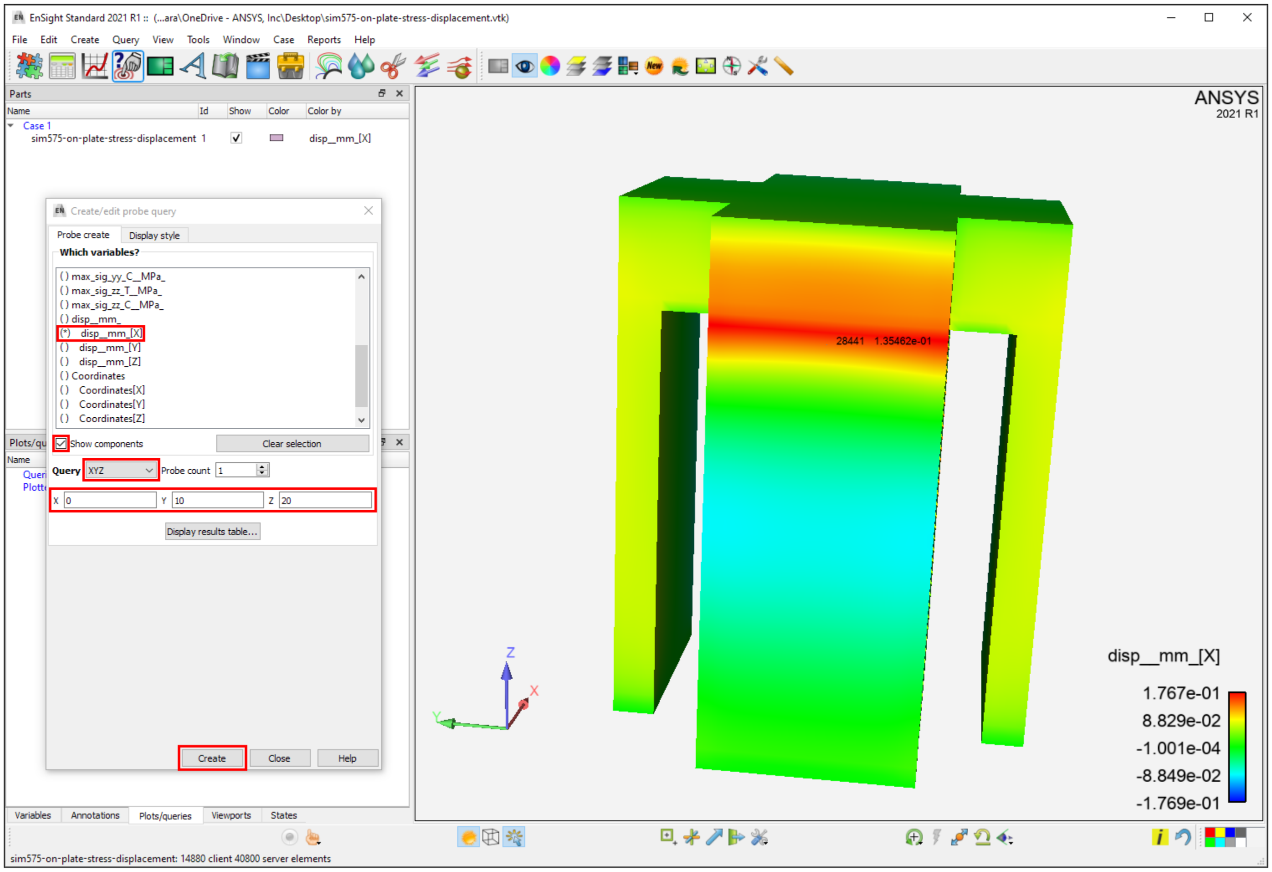
c. Click **File** > **Open**, navigate to the simulation .vtk file and click **OK**.

d. Right-click the newly added **sim file** and choose **Color by** > **Select variable**. In the dialog box that opens, click on the symbol to expand the Vectors options, choose **disp_mm_[X]** and then click **OK**.

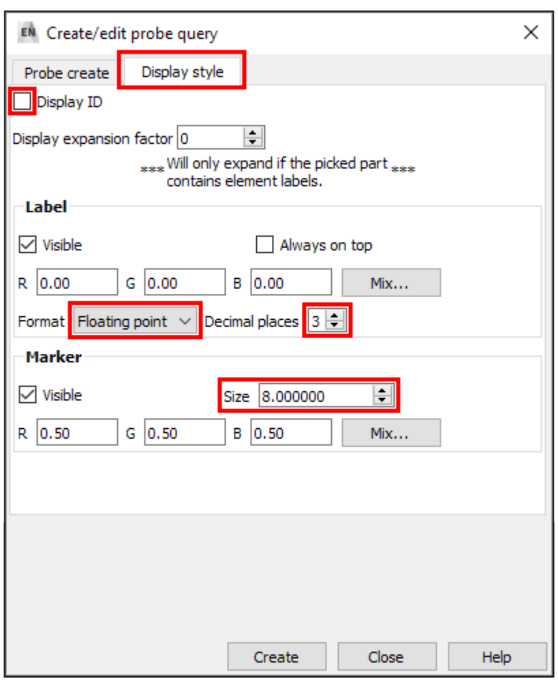


e. Rotate the model so that the pillar with the greatest shrinkage distortion (red contour) is shown. Click the **Interactive probe query** button  from the ribbon. A dialog box opens. In the Probe create tab click the **Show components** check box and then click **disp_mm_[X]** to display the X component of displacement.

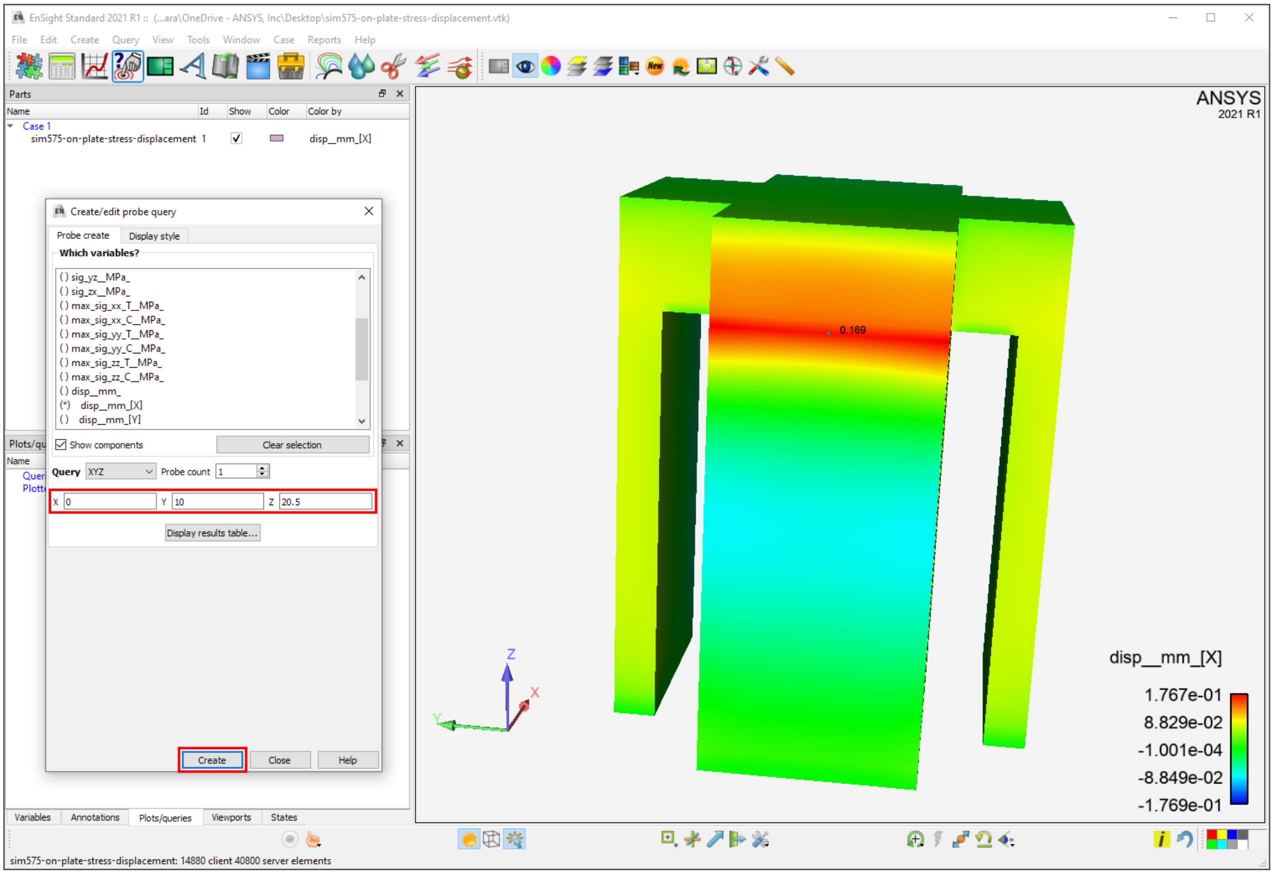
In the lower half of the dialog box, from the **Query** drop-down, choose **XYZ**. Enter X, Y, and Z values as **0**, **10**, and **20**. Click **Create**. The node number and X-displacement are shown.



f. Next we will use the settings in the Display style tab to change how the probe is displayed. In the Probe query box, switch to the **Display style** tab and change the settings as shown below. This removes the node ID number, changes the displacement value to floating point format with 3 decimal places, and increases the size of the marker dot that indicates the location.



g. Finally, we want to probe nearby node locations in the Z direction to see if we observe a higher displacement. Remember that the coordinates of (0,10,20) correspond to the location of maximum distortion in the experimental portion of the calibration. In the simulation the location of maximum distortion may shift a little bit since the element size is larger than the layer thickness. At (**0,10,20.5**) the distortion value of 0.169 mm is higher than at (0,10,20) so that is what we will use for the first calibration iteration.



2.8. Use Calibration Spreadsheet

A spreadsheet is used for the SSF calculations.

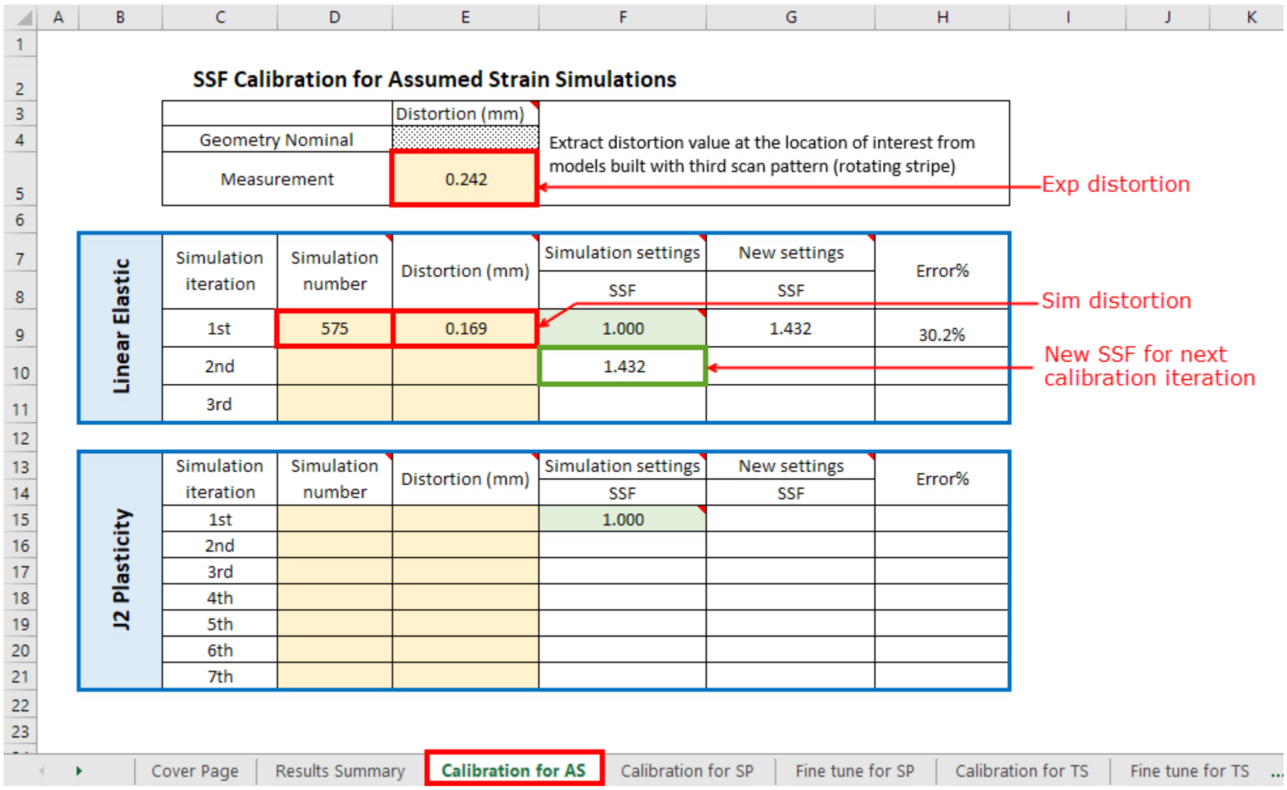
1. Open the Ansys Additive Calibration Sheet .xlxs file.

2. Click **Calibration for AS** tab.

3. Enter **0.242** mm as the experimental distortion target (measurement) in cell E5.

4. In the Linear Elastic table, record the **simulation number**, in our example it is 575, in cell D9 and then the distortion result **0.169** mm from the first calibration iteration in cell E9.

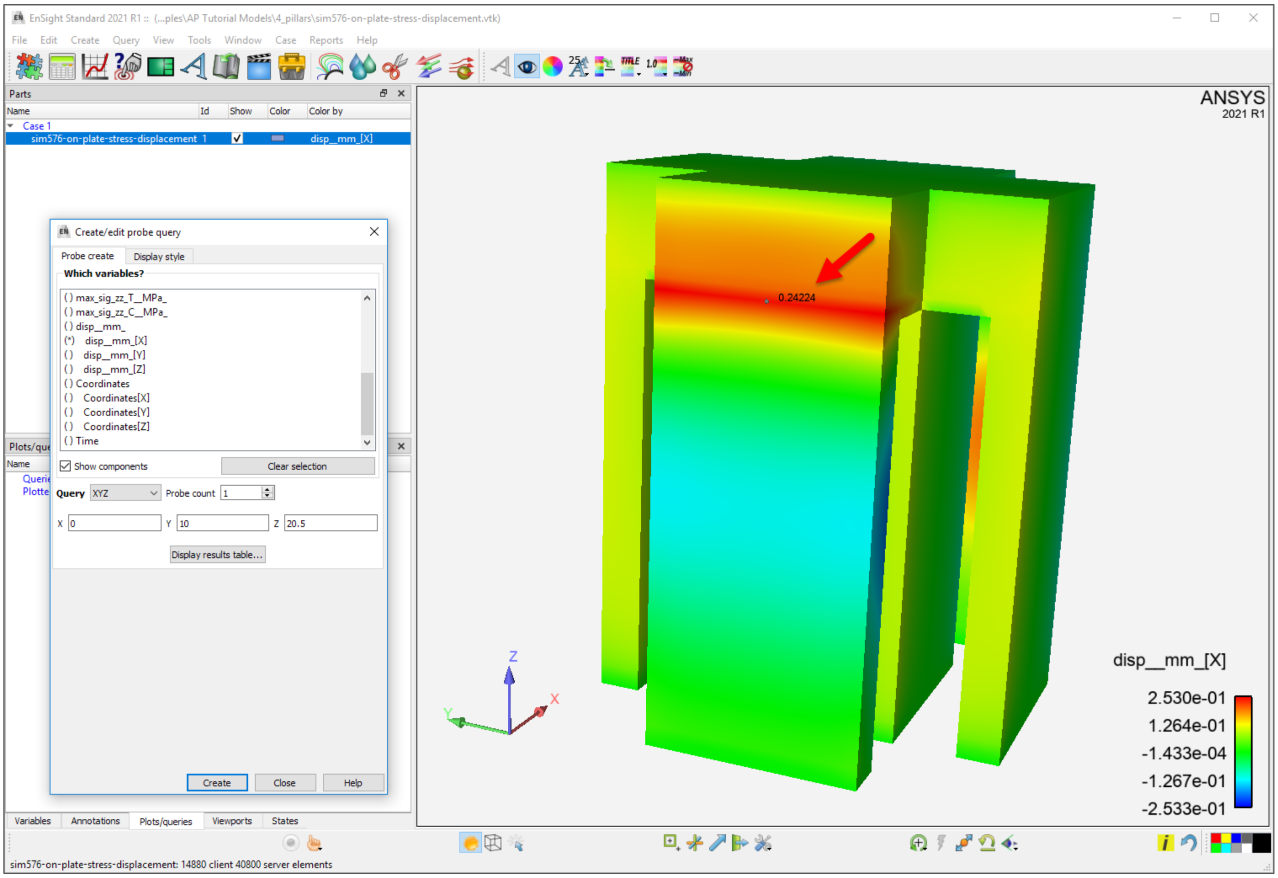
   The calibration spreadsheet will automatically calculate a new SSF value for the next simulation iteration, shown in both cell G9 and F10. At the same time, it will also calculate the % error between the simulation and the experimental distortion value.



2.9. Run Subsequent Calibration Iterations

Perform a new simulation with all inputs unchanged except for the new SSF value. To make it easy, use the **Duplicate** button on the simulation results page to copy the first iteration to a new simulation form. Then change the simulation description (to note 2nd iteration) and the SSF value and click Start. Iterate until the calculated SSF converges to an acceptable level of error between measured and simulated, or until the results will not get any better.

In our example, the best simulation result is achieved in the 2nd iteration with SSF = 1.432, which yields a 0.0% error.


EnSight Standard 2021 R1 :: (...ples\AP Tutorial Models\4_pillars\sim576-on-plate-stress-displacement.vtk)
File Edit Create Query View Tools Window Case Reports Help
Parts
Name Id Show Color Color by
Case 1
sim576-on-plate-stress-displacement 1 disp__mm_[X]
Create/edit probe query
Probe create
Display style
Which variables?
() max_sig_zz_T__MPa_
() max_sig_zz_C__MPa_
() disp__mm_
(*) disp__mm_[X]
() disp__mm_[Y]
() disp__mm_[Z]
() Coordinates
() Coordinates[X]
() Coordinates[Y]
() Coordinates[Z]
() Time
Show components
Clear selection
Query XYZ Probe count 1
X 0 Y 10 Z 20.5
Display results table...
Create Close Help
Variables Annotations Plots/queries Viewports States
sim576-on-plate-stress-displacement: 14880 client 40800 server elements
ANSYS
2021 R1
0.24224
disp__mm_[X]
2.530e-01
1.264e-01
-1.433e-04
-1.267e-01
-2.533e-01

SSF Calibration for Assumed Strain Simulations

	Distortion (mm)	
Geometry Nominal		Extract distortion value at the location of interest from models built with third scan pattern (rotating stripe)
Measurement	0.242	

	Simulation iteration	Simulation number	Distortion (mm)	Simulation settings SSF	New settings SSF	Error%
Linear Elastic	1st	575	0.169	1.000	1.432	30.2%
	2nd	576	0.242	1.432	1.432	0.0%
	3rd			1.432		

	Simulation iteration	Simulation number	Distortion (mm)	Simulation settings SSF	New settings SSF	Error%
J2 Plasticity	1st			1.000		
	2nd					
	3rd					
	4th					
	5th					
	6th					
	7th					

2.10. Save Calibrated Material

In the Results Summary tab of the spreadsheet, record the calibrated SSF for future Assumed Strain, Linear Elastic simulations in Ansys Additive using Ti-6Al-4V.

Summary of Results

Material	Stress Mode	Assumed Strain		Scan Pattern		Thermal Strain	
316L	Linear Elastic	SSF	1.432	SSF		SSF	
				ASC ∥		ASC ∥	
				ASC ⊥		ASC ⊥	
				ASC_z	1.000	ASC_z	1.000
	J2 Plasticity	SSF		SSF		SSF	
				ASC ∥		ASC ∥	
				ASC ⊥		ASC ⊥	
				ASC_z	1.000	ASC_z	1.000

Within Additive Print, we recommend you save the final SSF by creating a customized "calibrated material." In the Materials Library, select your original material and then click Customize. This brings up an edit panel where you can change the title, description, and SSF value for your calibrated material. Then be sure to select the appropriate custom material when performing future simulations.

Congratulations! You have completed the tutorial.

Chapter 3: LPBF Simulation - Bracket

This tutorial demonstrates how to perform a laser powder bed fusion (LPBF) additive process simulation in Mechanical using the LPBF Process Add-on. The following table shows the features used.

Applicable products:	Workbench/Mechanical An Additive Suite license is required in addition to the license for Mechanical.
Level of difficulty:	Easy
Interactive time required:	10 minutes to set up, 85 - 95 minutes to solve, 10 minutes to review results[a]
Manufacturing process:	Laser powder bed fusion (LPBF)
Simulation/system type:	AM LPBF Thermal-Structural
Material:	AlSi10Mg
Geometry:	One part and one volumeless STL Support
Mesh:	Part: Layered tetrahedral elements Support: Voxel elements
AM steps simulated:	1. Build and cooldown 2. Base unbolting 3. Base removal 4. Support removal
Features demonstrated:	LPBF Process Add-on, LPBF Setup Wizard[b], animation, section plane, AM result items
Help resources:	Using the LPBF Setup Wizard

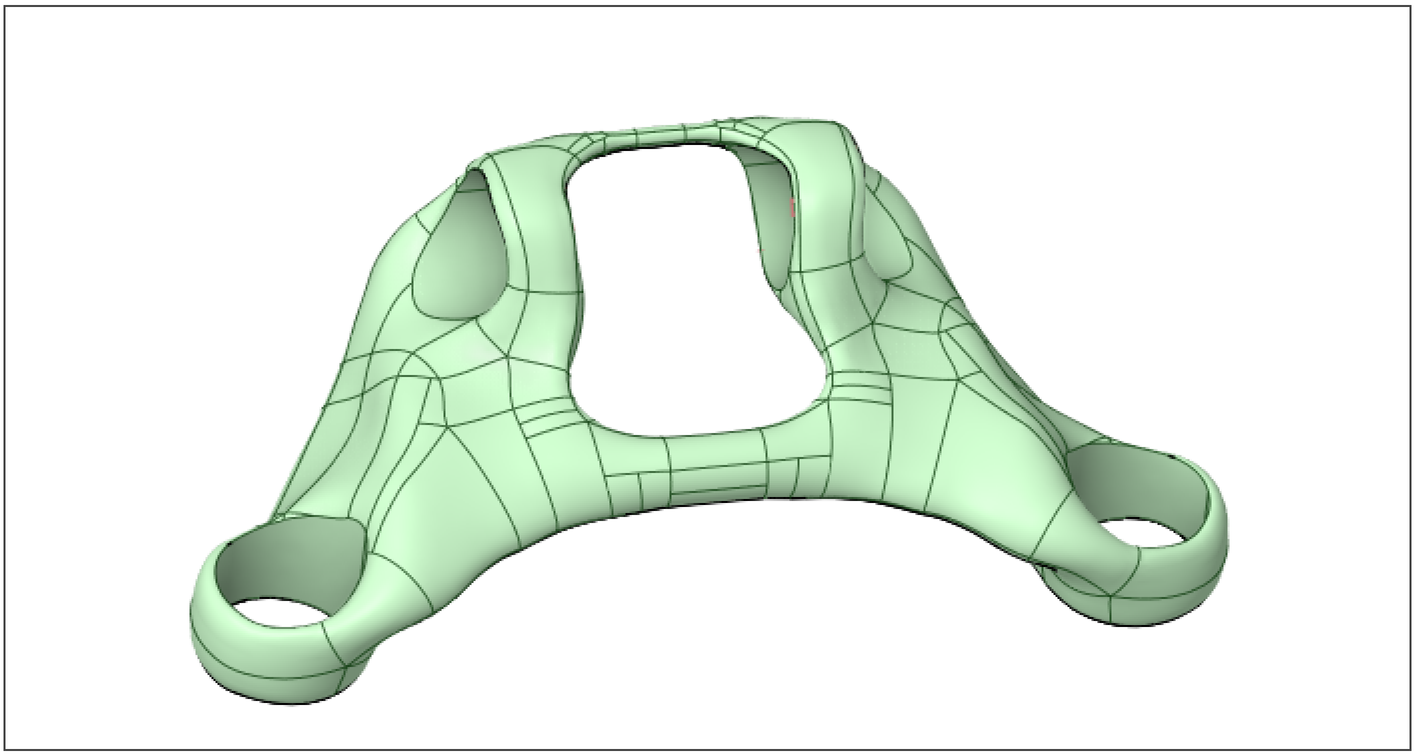
[a] This is an approximate range. The amount of time it takes you to complete the tutorial depends on the computer system and the number of CPU cores you use, the working pace that is comfortable for you, and so on.

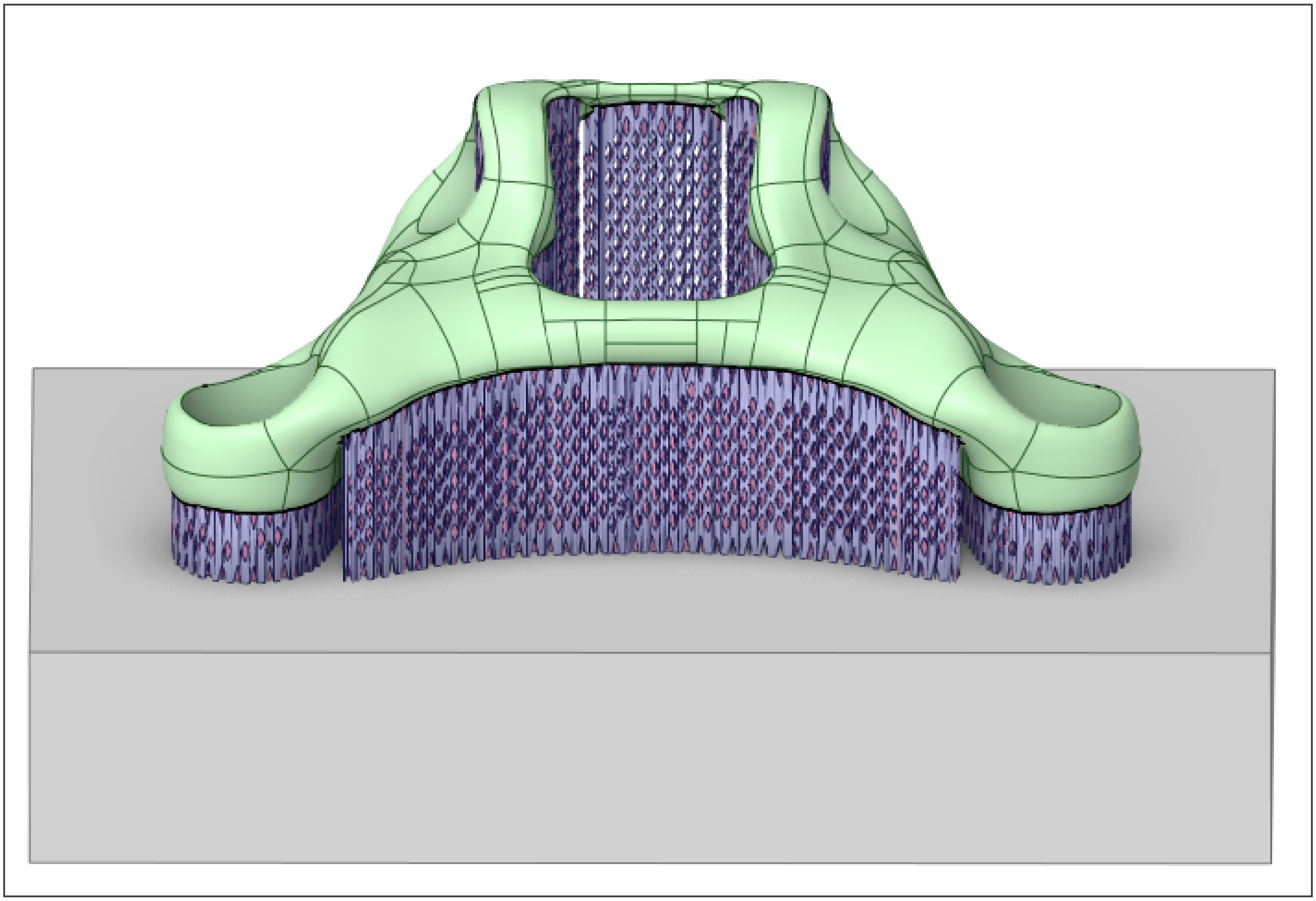
[b] This tutorial demonstrates the default Windows-based wizard introduced in Release 2023 R2, which is incompatible with Linux. Instead, Linux users will use the Legacy LPBF Setup Wizard, which is launched for Linux users by default. Although the GUI options and behaviors differ, Linux users should be able to follow along using the same engineering content.

Tutorial steps:

3.1. Problem Description

We will be simulating the additive manufacture of the bracket shown in the following figures. This geometry is one that is typical of a part that has been light-weighted using topology optimization. The first figure shows the part geometry and the second figure shows the part with its support on the additive build plate. The build material is AlSi10Mg.





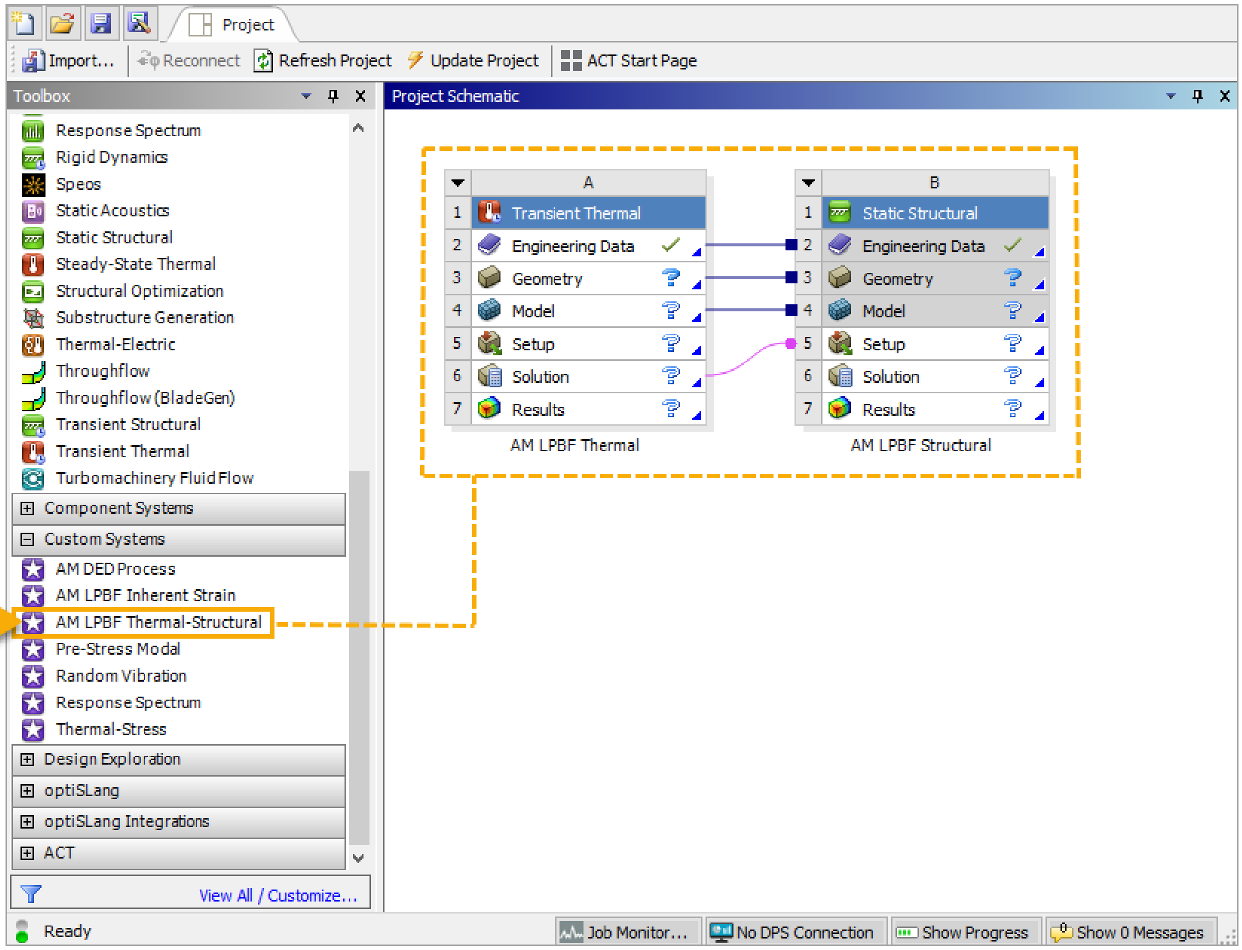
Tutorial Files

Click here to download the following:

- `bracket_on_base.scdoc` - Geometry file of the bracket part and a base plate, saved as an Ansys SpaceClaim document.
- `bracket_support_vless.stl` - Support structure for the bracket.

3.2. Create the Analysis System

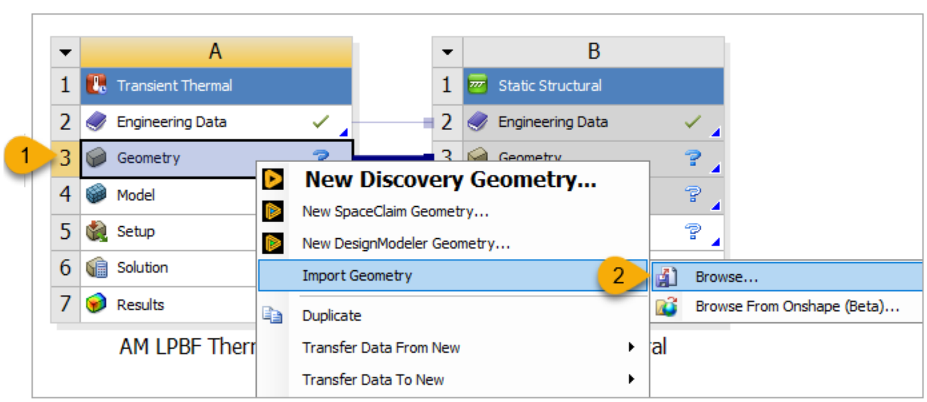
1. Open Ansys Workbench.
2. In the Toolbox on the left side of the window, scroll down to Custom Systems, expand the selection, and double-click **AM LPBF Thermal-Structural** to bring up the linked Transient Thermal, Static Structural system in the Project Schematic. The Transient Thermal system is named AM LPBF Thermal and the Static Structural system is named AM LPBF Structural.



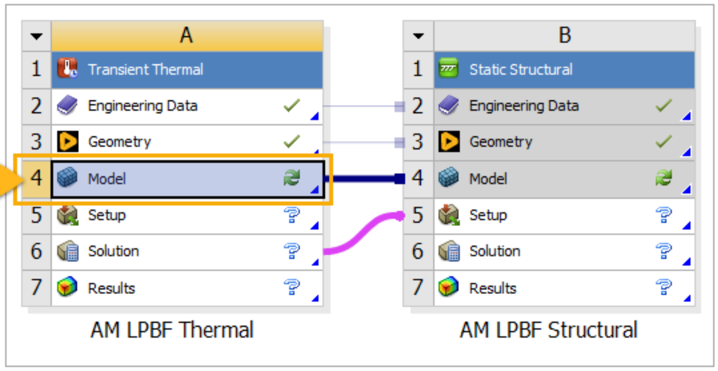
3.3. Import Geometry and Launch Mechanical

In this step, import the geometry file in Workbench and then open the Mechanical application.

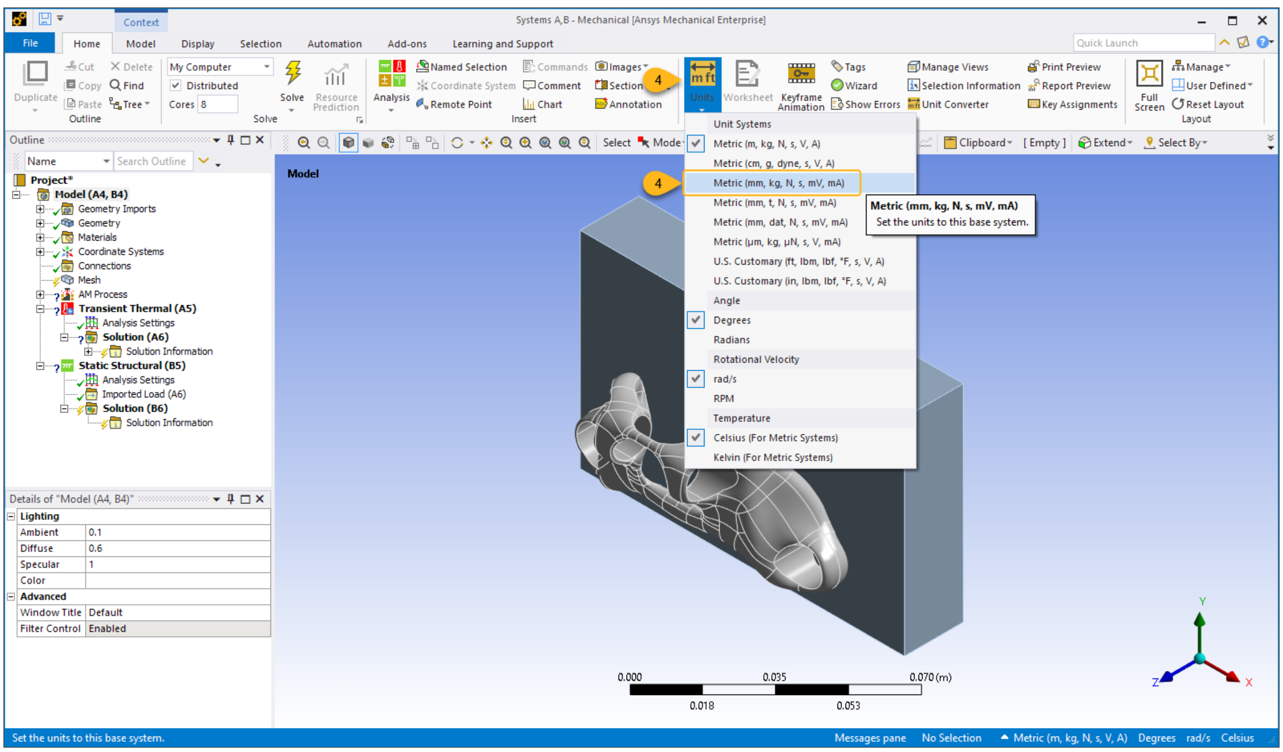
1. In the Transient Thermal system, right-click **Geometry** and select **Import Geometry**.
2. Click **Browse** and navigate to the bracket geometry file (`bracket_on_base.scdoc`), select it, and click **Open** to add it to the analysis. A green check mark appears next to the Geometry cell in the Project Schematic when the geometry is added.



3. Double-click the **Model** cell in the Transient Thermal system to launch the Mechanical application. A message "Starting Mechanical" appears in the status bar in the bottom, left corner. It may take several seconds for the application to open and attach the geometry.

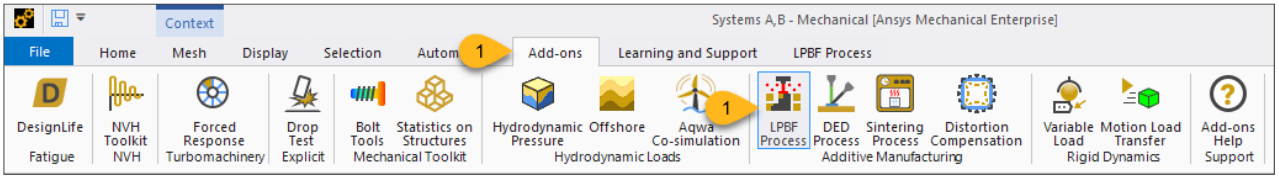


4. Once Mechanical is open and you see the model, set the units for this model to millimeters. From the **Home** tab in the ribbon, in the Tools group, select **Units** > **Metric (mm, kg, N, s, mV, mA)**.

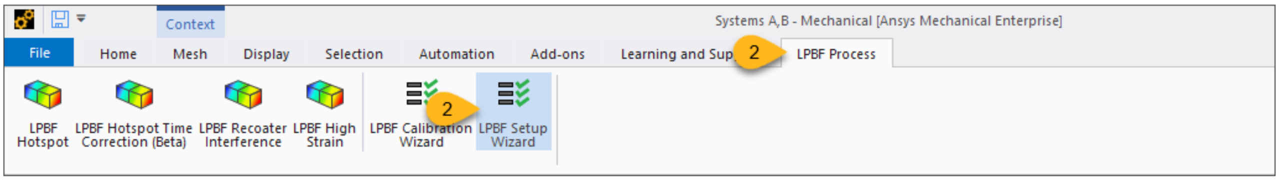


3.4. Load the LPBF Process Add-on and Open the LPBF Setup Wizard

1. From the **Add-ons** tab in the ribbon, in the Additive Manufacturing group, select **LPBF Process** to load the add-on. It takes a few seconds to load.

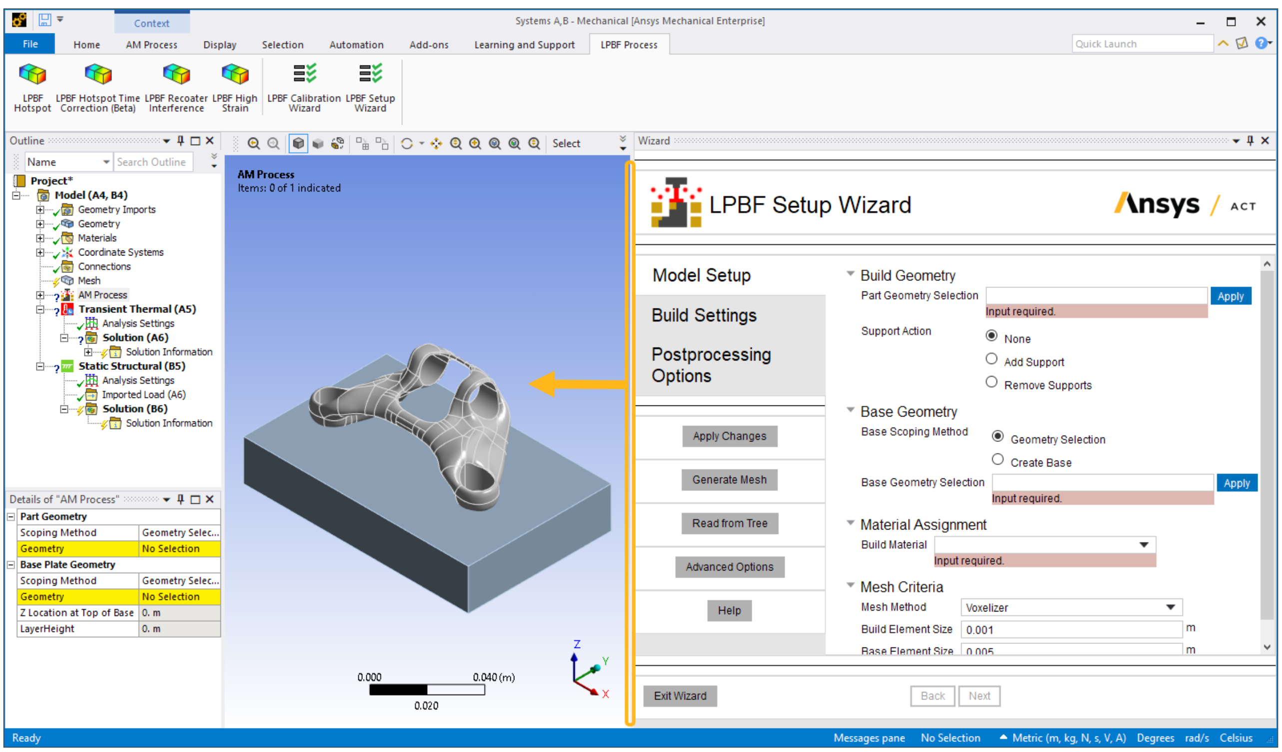


2. When the **LPBF Process** tab appears in the ribbon, select **LPBF Setup Wizard** to open the wizard.



When the wizard launches, the view of the model automatically changes so that the positive Z-axis points upward to match the build direction.

You may want to widen the application and/or wizard window to allow more space to see the wizard options.



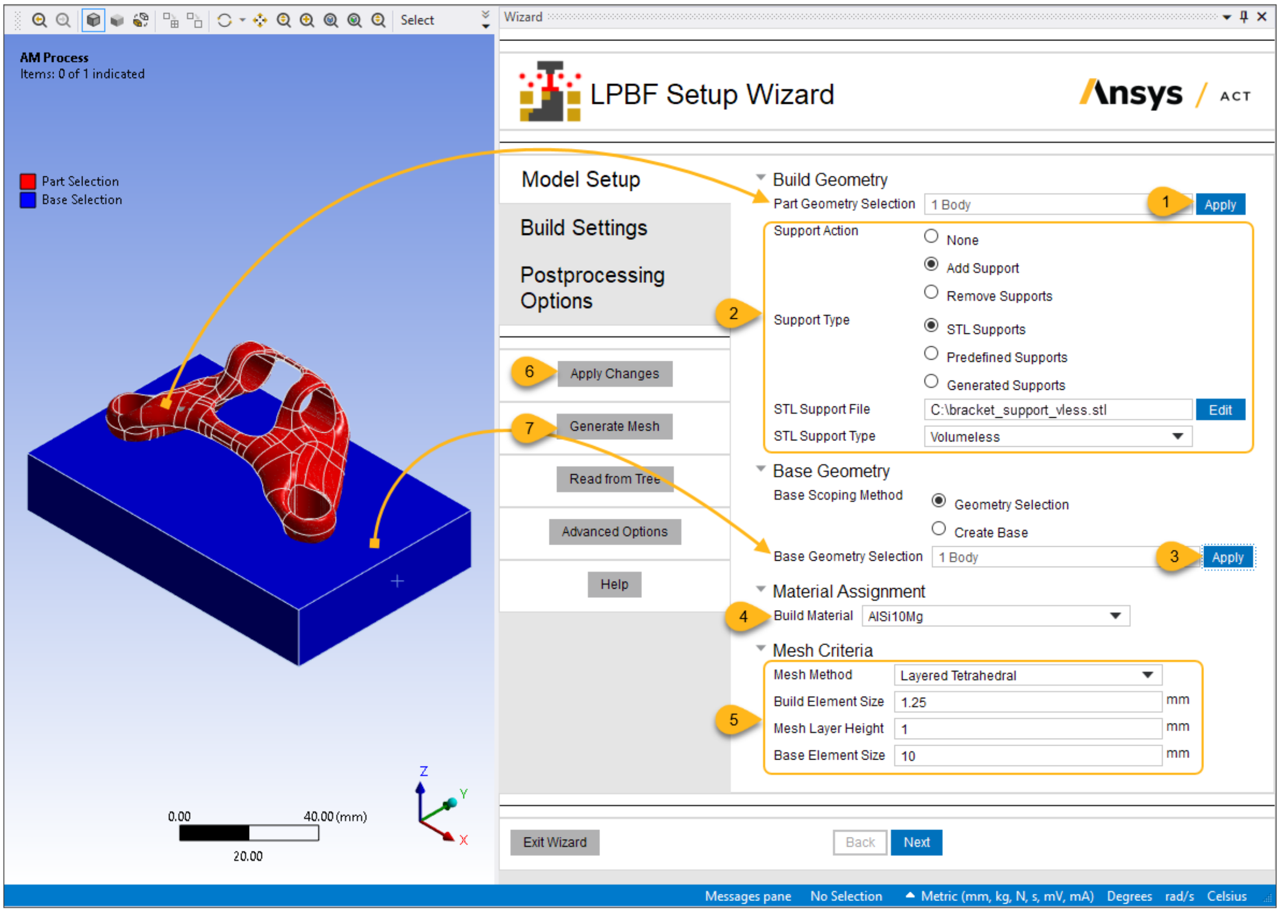
3.5. Complete Wizard Step 1 - Model Setup

In this step, set up the model by identifying the build and base geometries, adding the STL Support, assigning the material, and defining the mesh criteria. A *layered* mesh in the positive Z direction is required in an LPBF additive simulation so that the analysis follows the build process itself: layer-by-layer solidification of the part. See Methodology and Abstractions in the *LPBF Simulation Guide* for additional information.

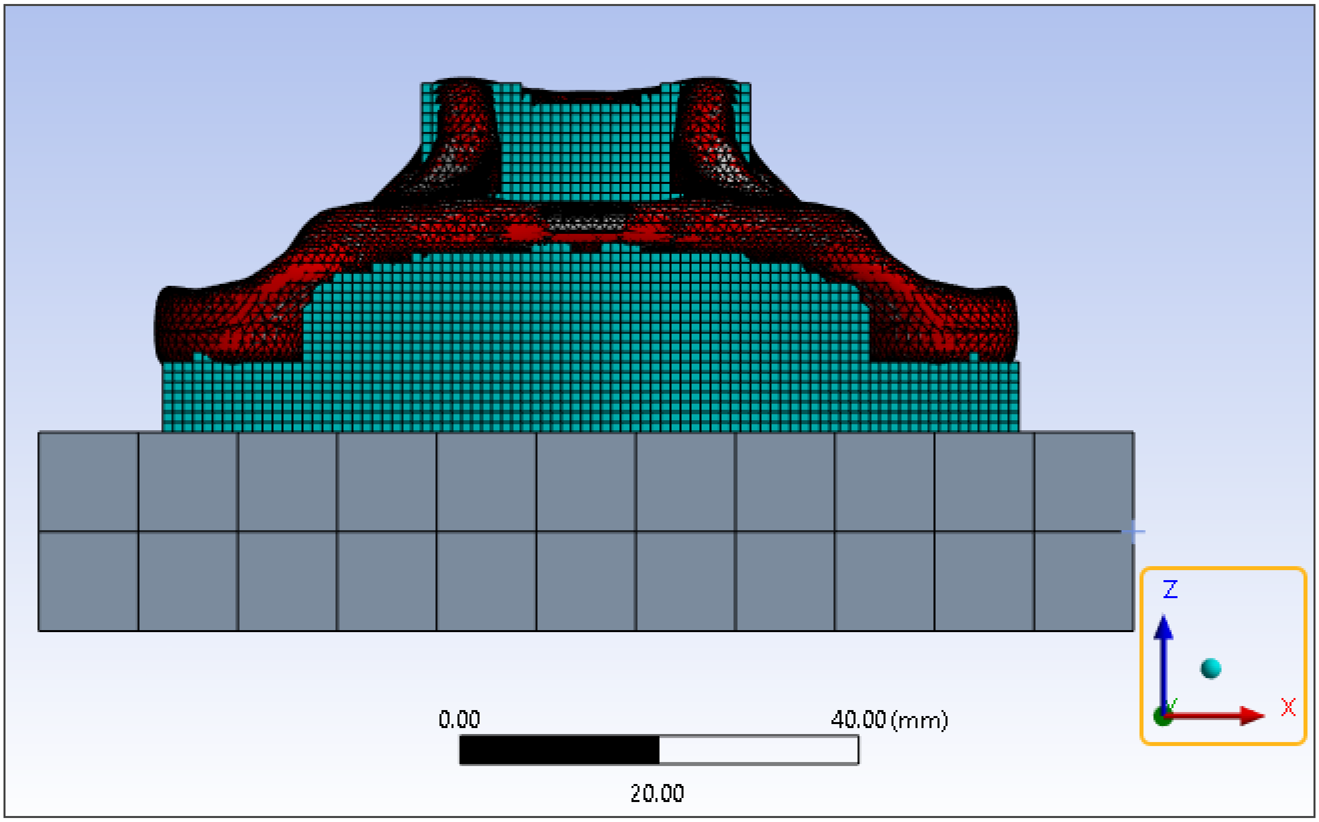
1. Select the bracket body and click **Apply** in the Part Geometry Selection field. Note the Body picker  is active by default.
2. Add the support:
   a. In Support Action, select **Add Support**.
   b. For Support Type, keep the default **STL Supports** option.
   c. Click **Edit** in the STL Support File field, then browse to the bracket support structure file (`bracket_support_vless.stl`), select it and click **Open**.
   d. For STL Support Type, keep the default **Volumeless** option.
3. Select the base body and click **Apply** in the Base Geometry Selection field.
4. Under Material Assignment, choose **AlSi10Mg** as the Build Material from the drop-down menu.
5. Set mesh criteria:
   a. Choose **Layered Tetrahedral** as the Mesh Method.
   b. Specify Build Element Size = **1.25** mm, slightly larger than Mesh Layer Height. This allows any given tetrahedral element to have a non-Z-direction edge length of up to 1.25 mm but still maintain the 1 mm layer height in the Z-direction.
   c. Specify Mesh Layer Height = **1** mm. This value is larger than recommended for the model, but it is used in the tutorial for speed considerations. Our general guideline for a "super layer" is to use 10-20 times the size of the machine deposition thickness.
   d. Set Base Element Size = **10** mm. The base plate does not need to have as fine a mesh as the part.
6. Click **Apply Changes** to update the project tree with all of our inputs.
7. Click **Generate Mesh**.

Important:

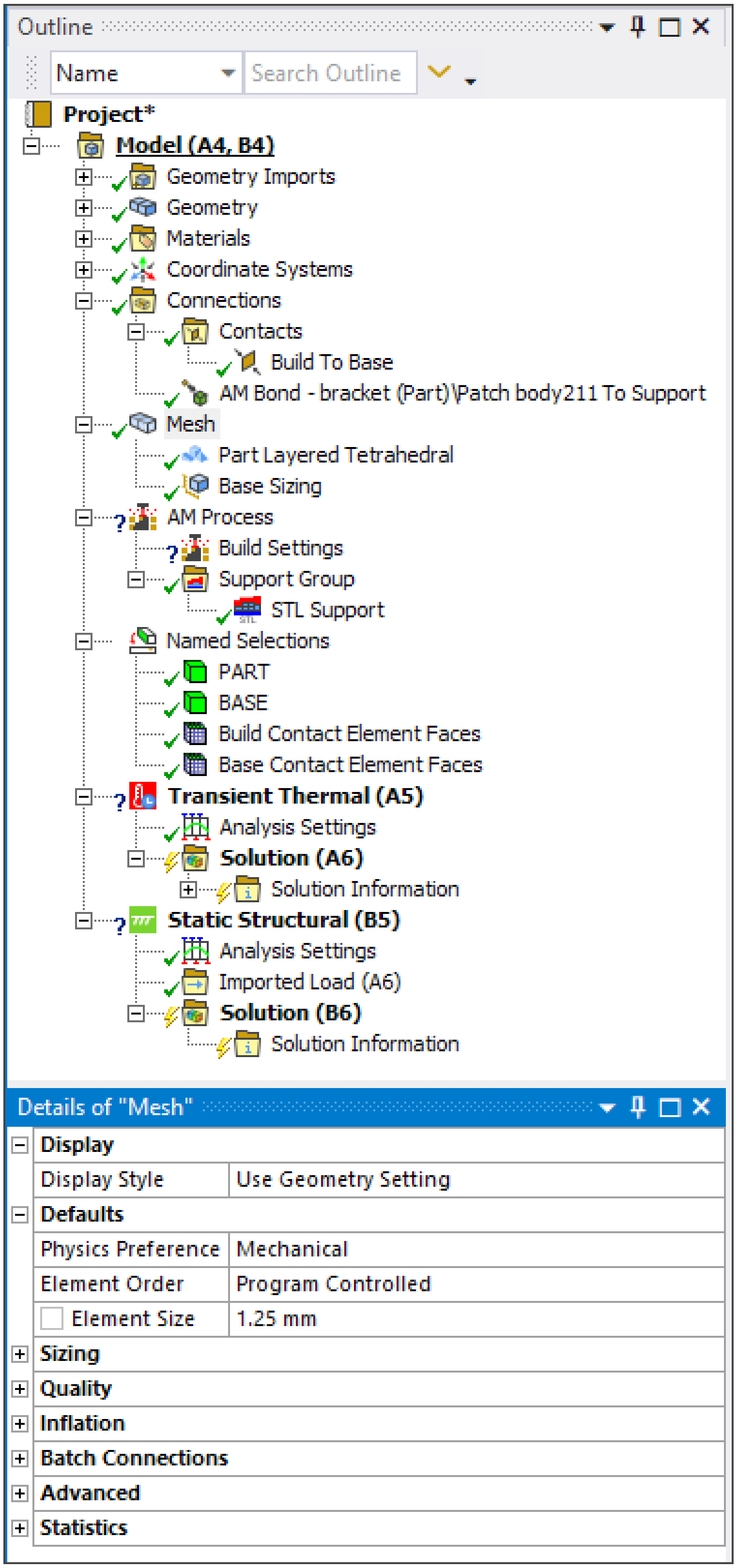
Be sure to click **Apply Changes** first before generating the mesh. This ensures the additive bodies (part/support/base plate) are identified correctly and a layered mesh will be generated.



To see a better view of the layered mesh, reorient the model by clicking the triad's axes arrows a couple times to see a back or front view.



Review the objects and options in the project tree that were created or updated when you clicked the Apply Changes and Generate Mesh action buttons. Mesh objects, contact objects and others have been added or changed automatically by the wizard. Review the Details of related objects to see the wizard options you specified. Notice the Named Selections for the build body, the base body, and sets of element faces used for contact connections. Contact connections—between the support and the base plate (Build To Base), and between the part and its support (AM Bond)—are created when the mesh is generated.



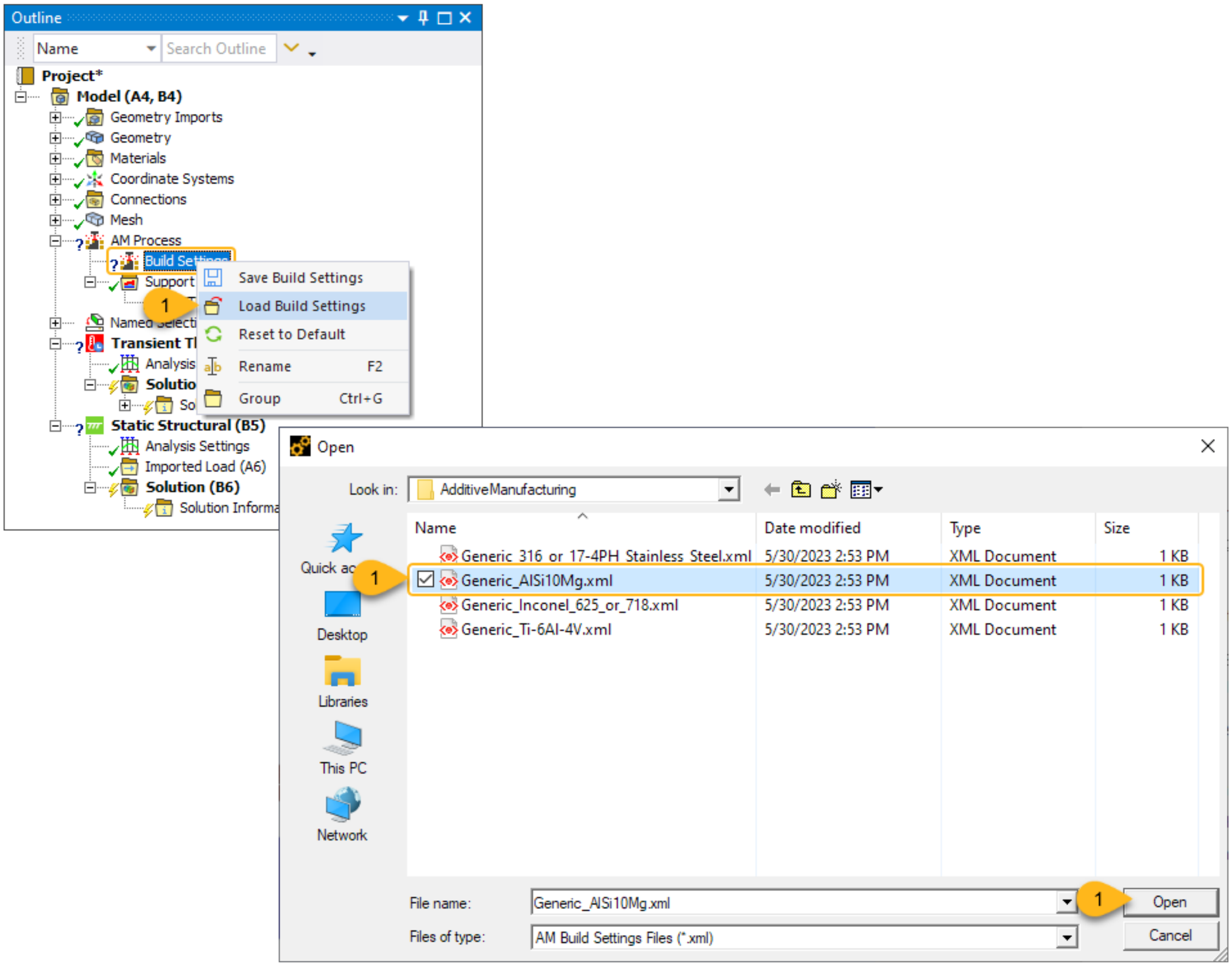
8. At the bottom of the wizard panel, click **Next** to move to the next wizard step.

3.6. Complete Wizard Step 2 - Build Settings

In this step, we consider simulation assumptions, calibration factors, machine settings, build conditions, and boundary conditions.

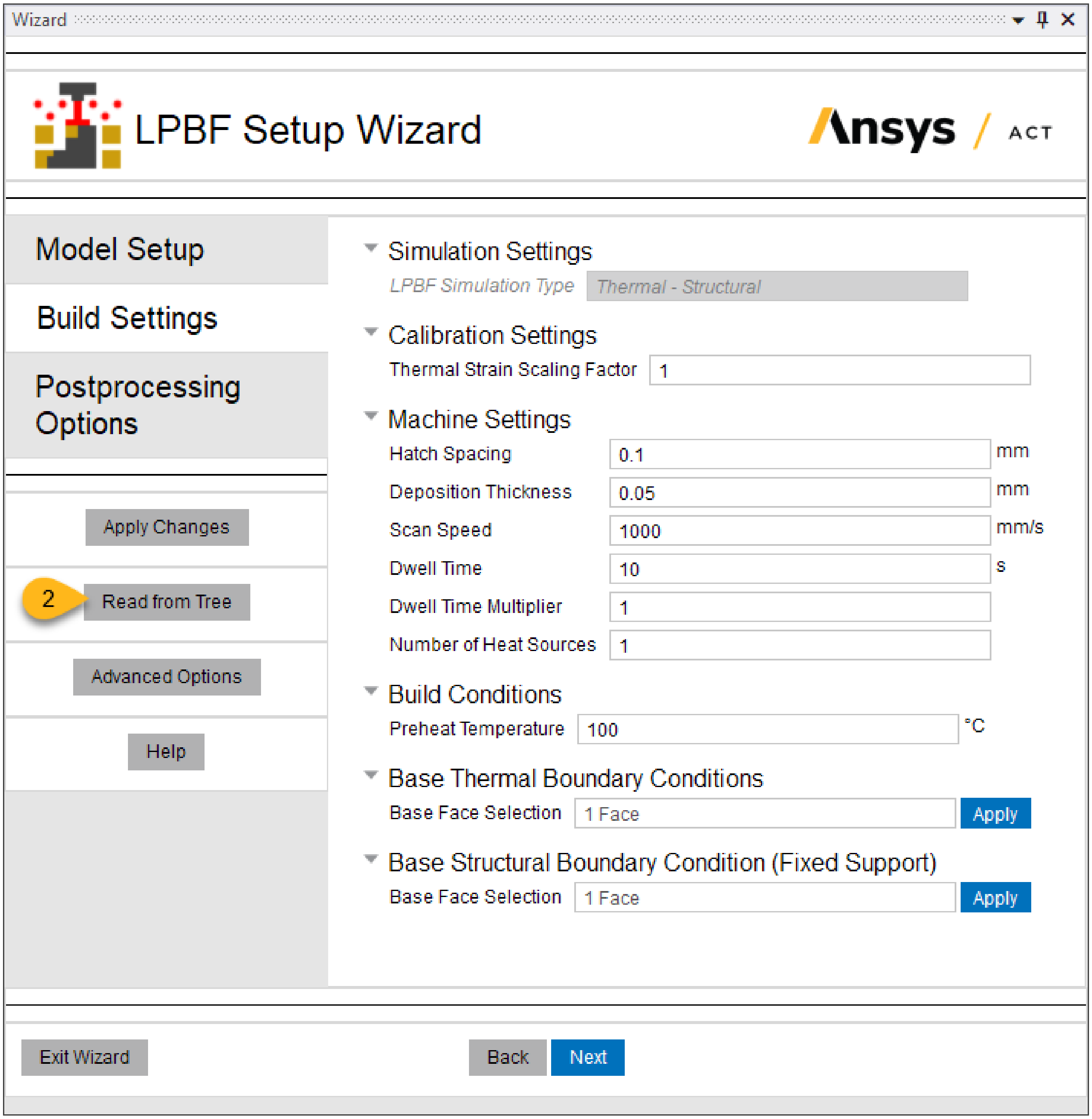
- Simulation assumptions are set by default based on the chosen AM LPBF type. In this tutorial, we used a thermal-structural system.
- We'll use the default Thermal Strain Calibration Factor of 1. This factor scales the thermal strains in a thermal-structural simulation and its value is determined from calibration experiments.
- Machine settings are process parameters that directly influence how the process deposits material. We will use Ansys-supplied values for generic AlSi10Mg material.
- Build conditions are settings pertaining to the environment *around the part* during the deposition process and during cooldown. This includes Preheat Temperature.
- For thermal boundary conditions, the underside of the base is typically heated to maintain a constant, slightly elevated temperature. By default, the underside surface of the base is already selected and a convection will be applied automatically.
- For structural boundary conditions, the underside of the base should be fixed. By default, the underside surface of the base is already selected and the fixed support will be applied automatically.

1. In the project tree (outside of the wizard), select the **Build Settings** object (under the AM Process object), right-click, and choose **Load Build Settings**. A folder of Ansys-supplied sample files appears by default. Select `Generic_AlSi10Mg.xml` and click **Open**.



The step just performed loads the material properties for AlSi10Mg into the project tree. Back in the wizard, use the Read from Tree button to update the wizard panel with those same properties.

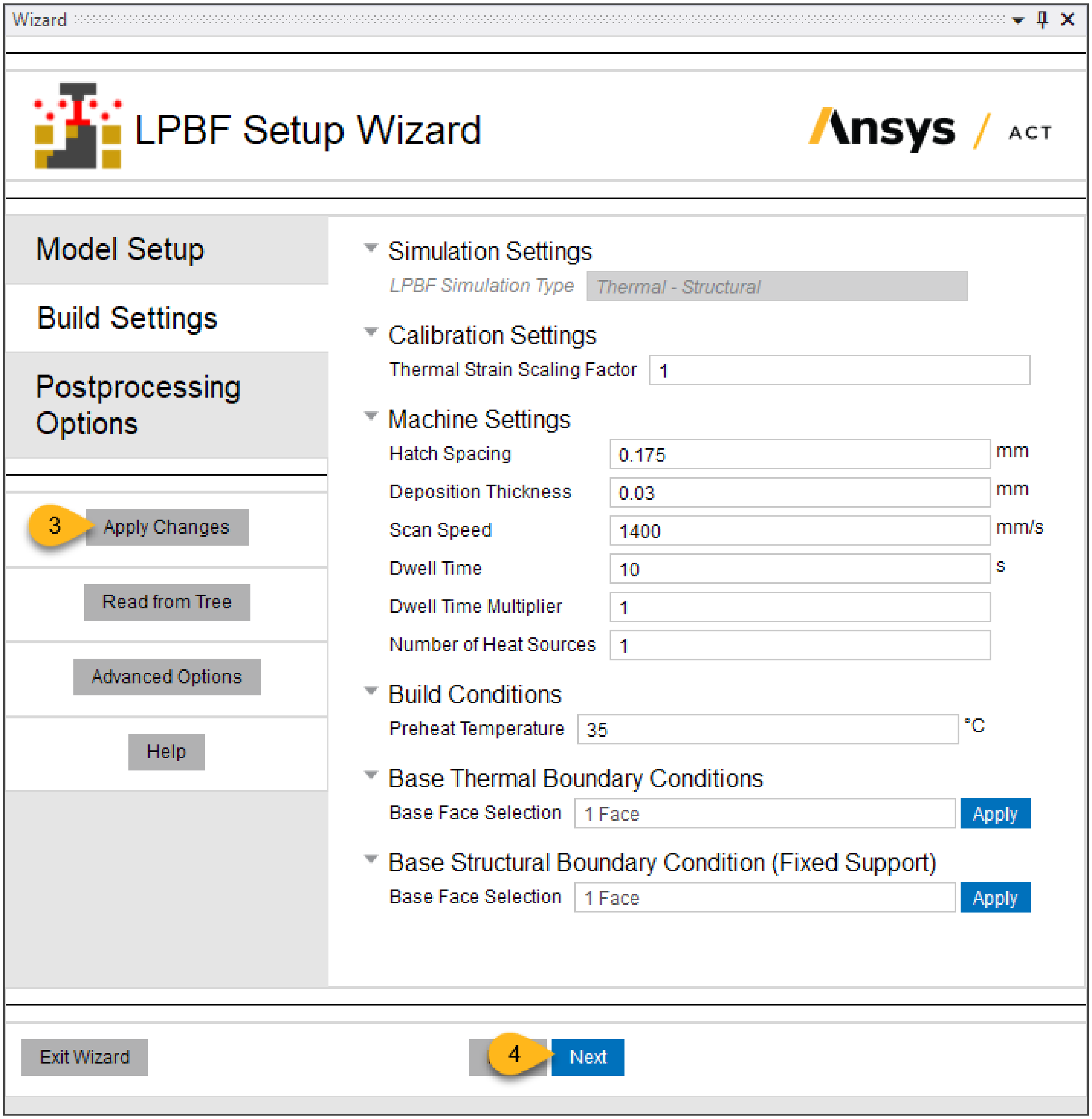
2. In the Build Settings step of the wizard, click **Read from Tree**.



Immediately the properties in the wizard panel are updated to reflect those of AlSi10Mg.

3. Click **Apply Changes** to update the project tree with the other settings from this wizard step, including the pre-populated face selections for the boundary conditions.

4. Click **Next** to move to the next wizard step.



3.7. Complete Wizard Step 3 - Postprocessing Options

In this step, select the post-build processing steps to simulate and select LPBF result items to be calculated during solution.

1. Select **Base Unbolting**, **Base Removal**, and **Support Removal** as the Postprocessing Steps.

2. Select **Directional** as the Base Removal Type, then define Removal Step Size = **10** mm and Removal Direction = **90**°. This will simulate the removal of supports in increments of 10 mm in the global Y direction on the X-Y plane.

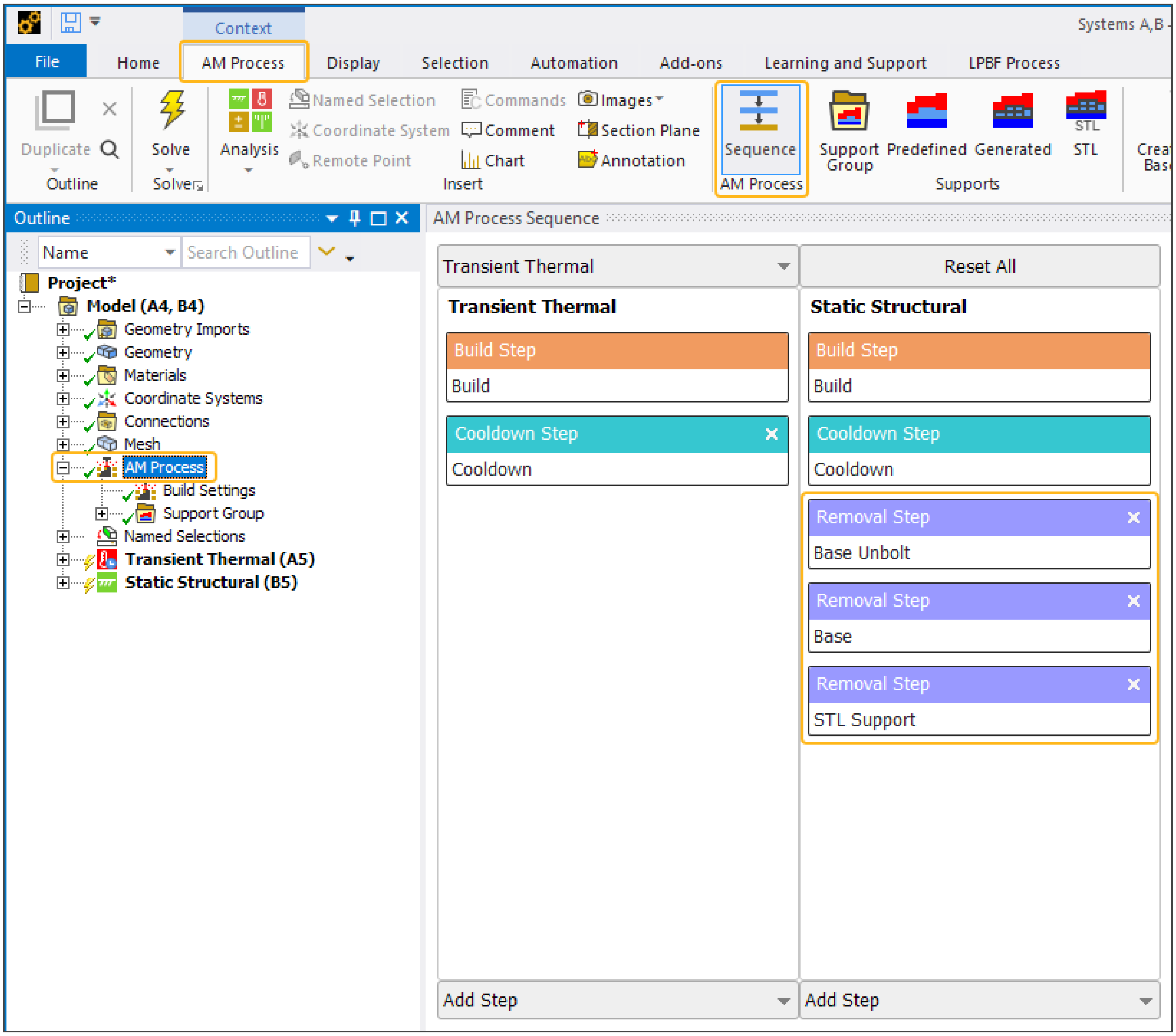
3. Keep the default check boxes to calculate all LPBF Results. We will modify their settings individually next.

4. For the Hotspot Result, select **User Defined** as the Threshold Definition, then define a Warning Threshold = **135**°C, which is 100° above the Preheat Temperature, and a Critical Threshold = **235**°C.

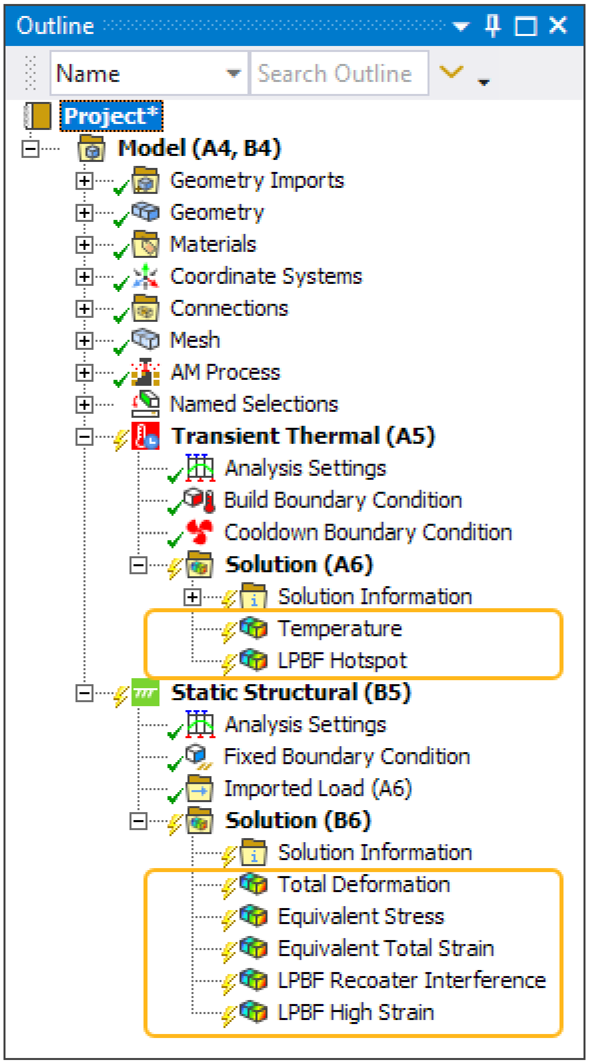
5. For the Recoater Interference Result, select **Layer Thickness Based** as the Threshold Definition, then define Powder Packing Density = **0.6**.

6. For the High Strain Result, select **User Defined** as the Threshold Definition, then define a Warning Threshold = **0.1** and a Critical Threshold = **0.2**.

7. Click **Apply Changes** to update the project tree with our selections for postprocessing steps. To review that the steps have been added, look at the AM Process Sequence worksheet. Select the **AM Process** object in the project tree, click the **AM Process tab**, and then select the **Sequence** button. The Sequencer appears. There you can see the postprocessing steps that were added by the wizard.



8. Click **Add Results**. This will add the standard result items to the project tree, specifically Temperature for the thermal system and Total Deformation, Equivalent Stress, and Equivalent Total Strain for the structural system, *and* the LPBF results. Note that the Add Results action button is similar to the Generate Mesh button in that its action is separate from the Apply Changes button. This allows the flexibility to add multiple versions of LPBF result items if desired, perhaps with different criteria defined for each.

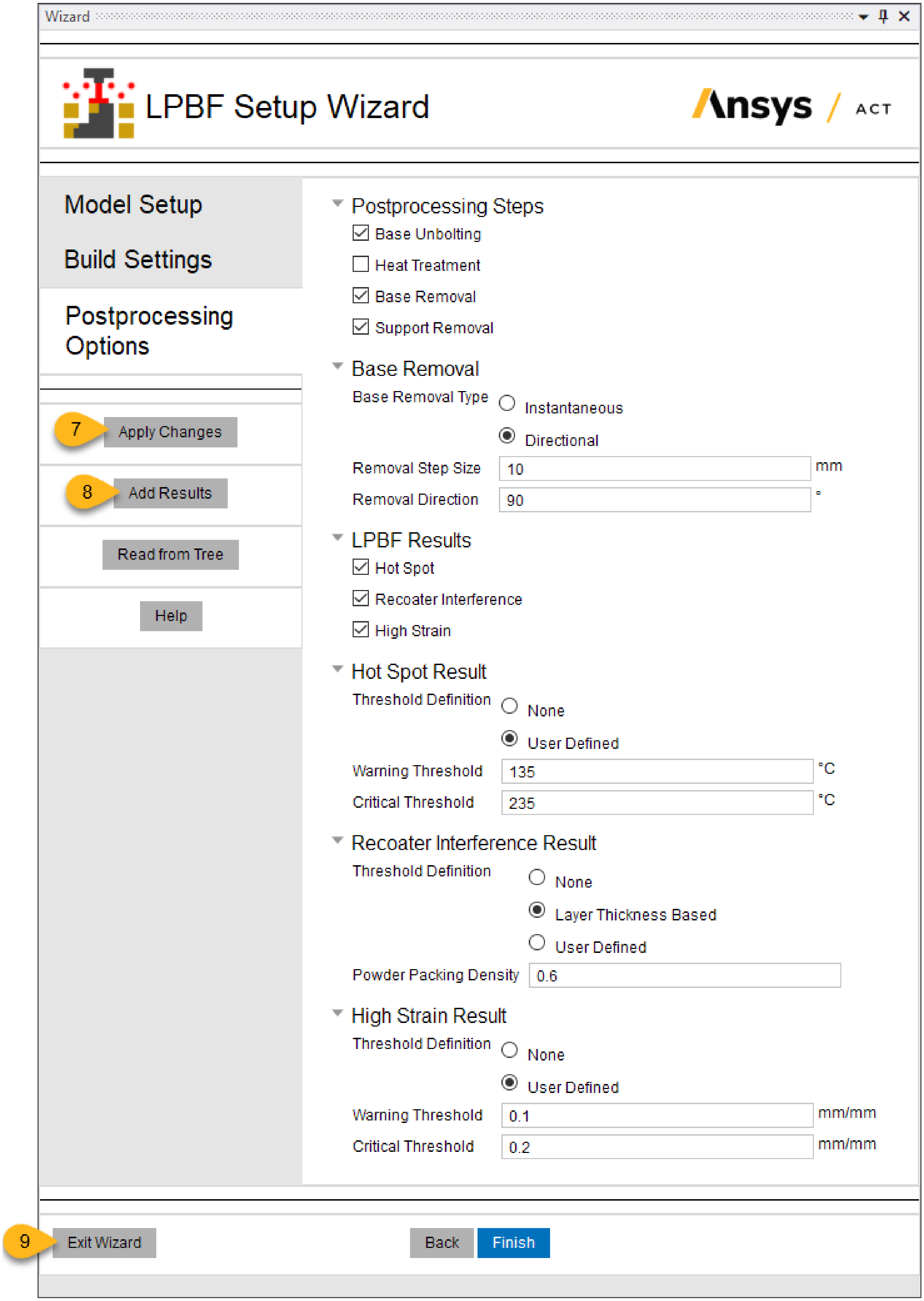


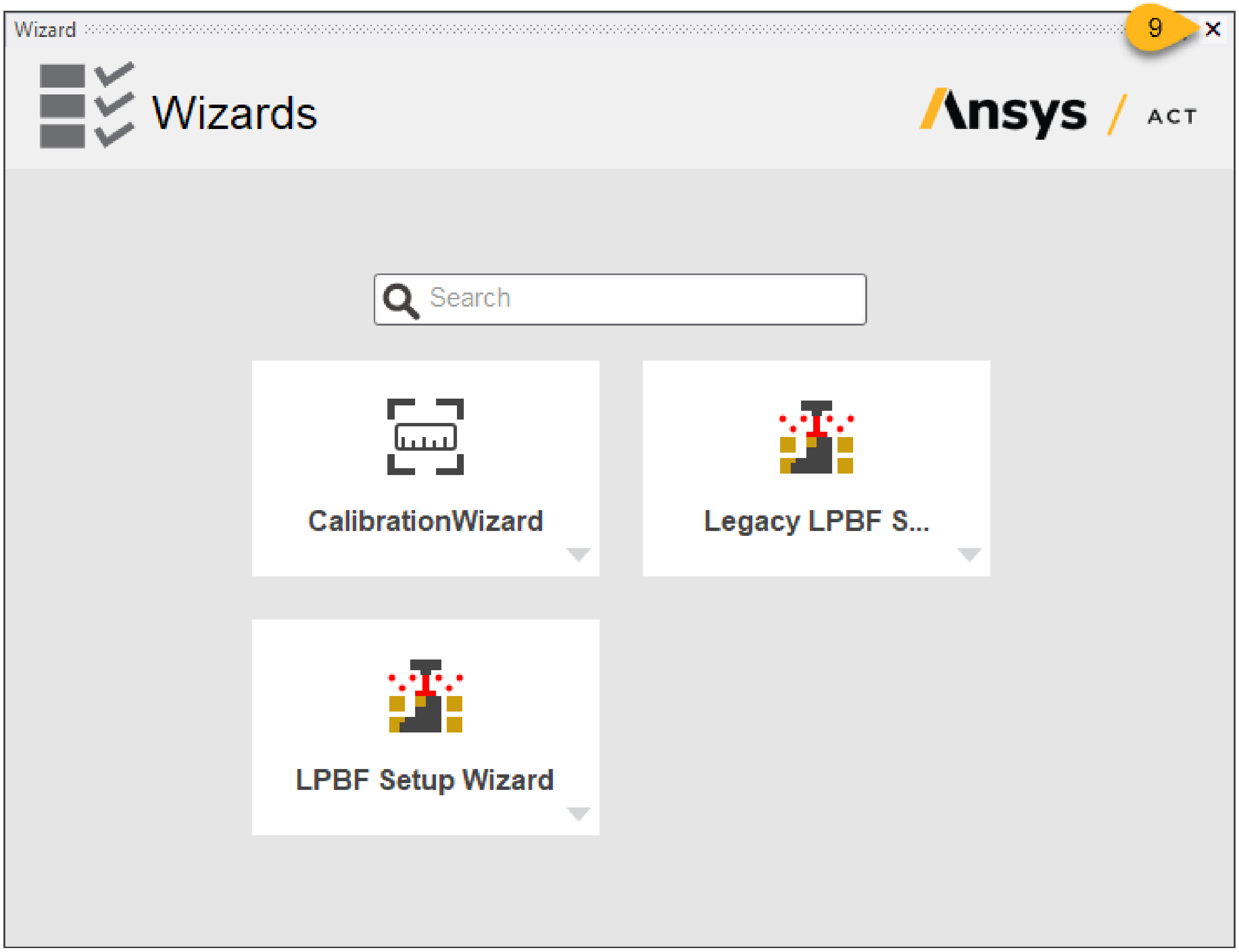
9. Click **Exit Wizard**.

Note:

Clicking Finish will apply all changes to all wizard steps sequentially and then close the wizard. Since we already clicked Apply Changes in all the wizard steps, we can simply exit the wizard.

Answer **Yes** to exit to the wizard start page and then click the **x** in the upper right corner to close the wizard panel.

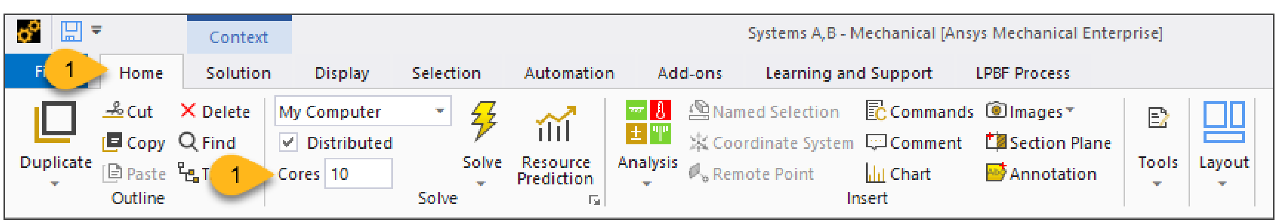

Wizard
LPBF Setup Wizard
Ansys / ACT
Model Setup
Build Settings
Postprocessing Options
7
Apply Changes
8
Add Results
Read from Tree
Help
Postprocessing Steps
Base Unbolting
Heat Treatment
Base Removal
Support Removal
Base Removal
Base Removal Type
Instantaneous
Directional
Removal Step Size
10
mm
Removal Direction
90
°
LPBF Results
Hot Spot
Recoater Interference
High Strain
Hot Spot Result
Threshold Definition
None
User Defined
Warning Threshold
135
°C
Critical Threshold
235
°C
Recoater Interference Result
Threshold Definition
None
Layer Thickness Based
User Defined
Powder Packing Density
0.6
High Strain Result
Threshold Definition
None
User Defined
Warning Threshold
0.1
mm/mm
Critical Threshold
0.2
mm/mm
9
Exit Wizard
Back
Finish



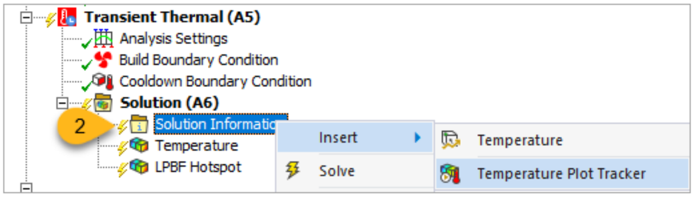
The simulation is ready to solve when you have completed the last step in the wizard, but be sure to review the status and settings of the project tree one more time. Look for green check marks to the left of required objects and yellow lightning bolts for items that are ready to solve. A question mark next to an object indicates the required inputs are not complete.

3.8. Solve

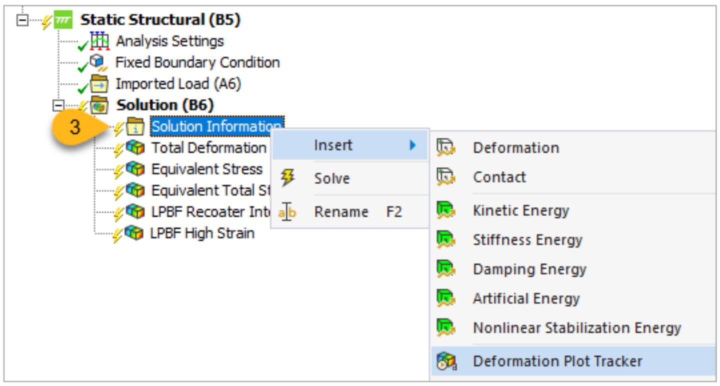
1. Under the **Home** tab in the ribbon, find the **Solve** group. Set the number of CPU **Cores** to solve with for this simulation, considering your computer capabilities and licensing for HPC. This example uses 10 cores.



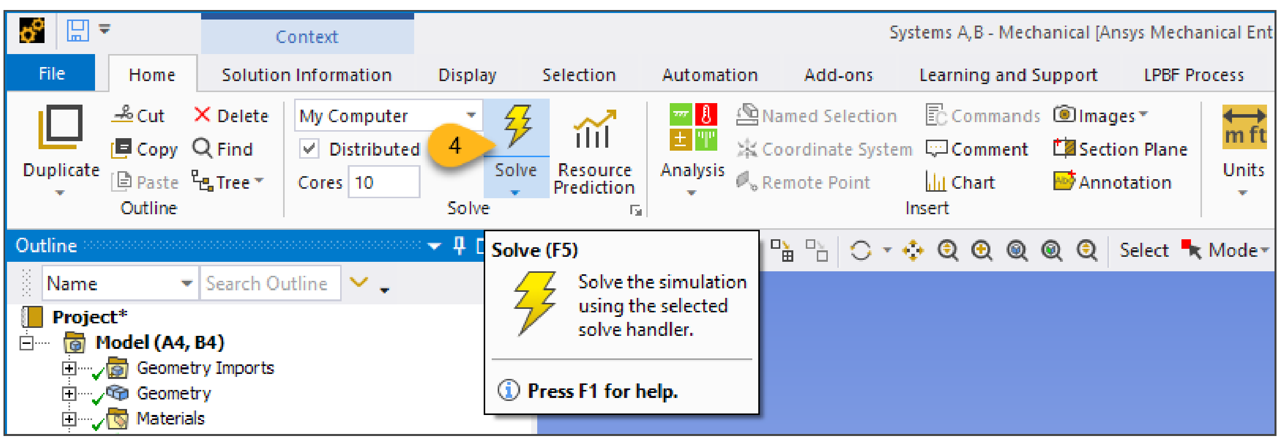
2. To set up a plot of overall temperature that may be updated throughout the solution, under Transient Thermal, Solution, right-click **Solution Information** and select **Insert** > **Temperature Plot Tracker**.



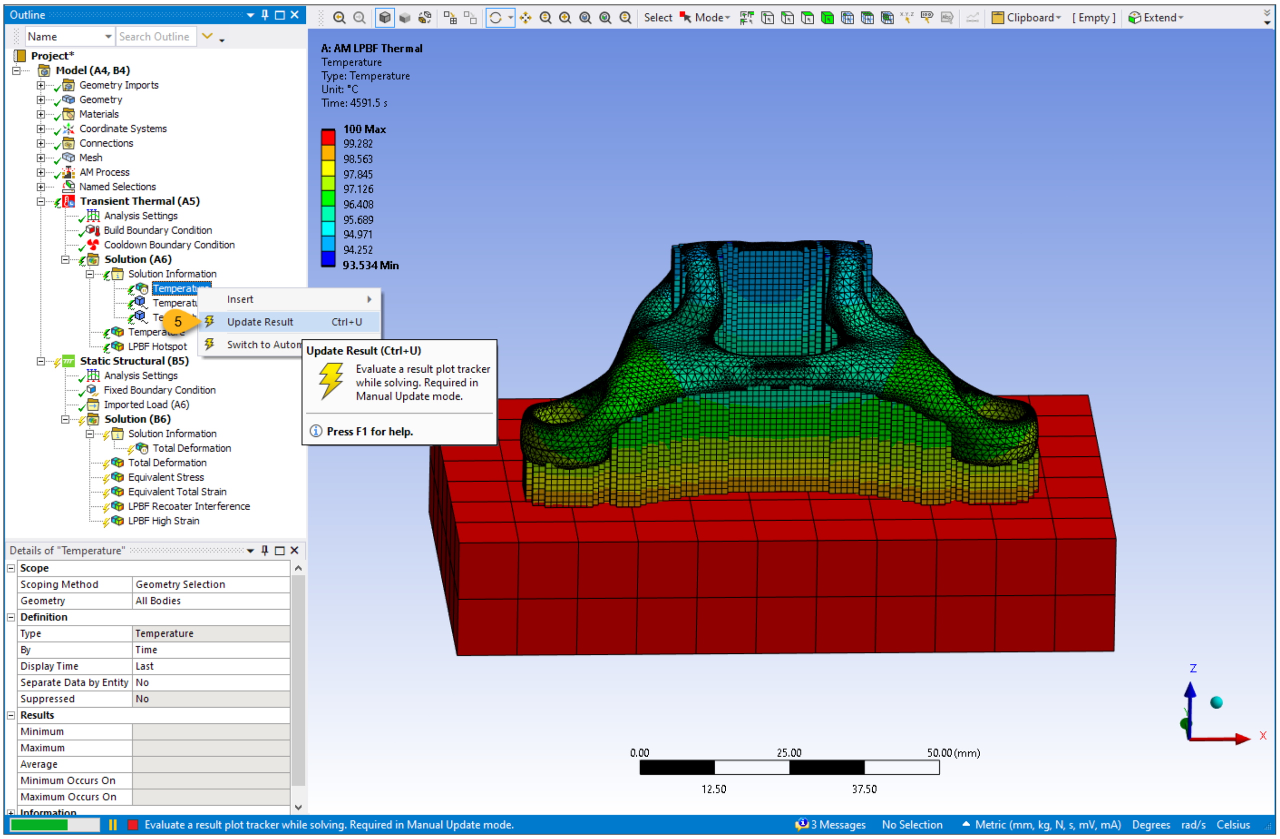
3. To set up a plot of overall deformation that may be updated throughout the solution, under Static Structural, Solution, right-click **Solution Information** and select **Insert** > **Deformation Plot Tracker**.



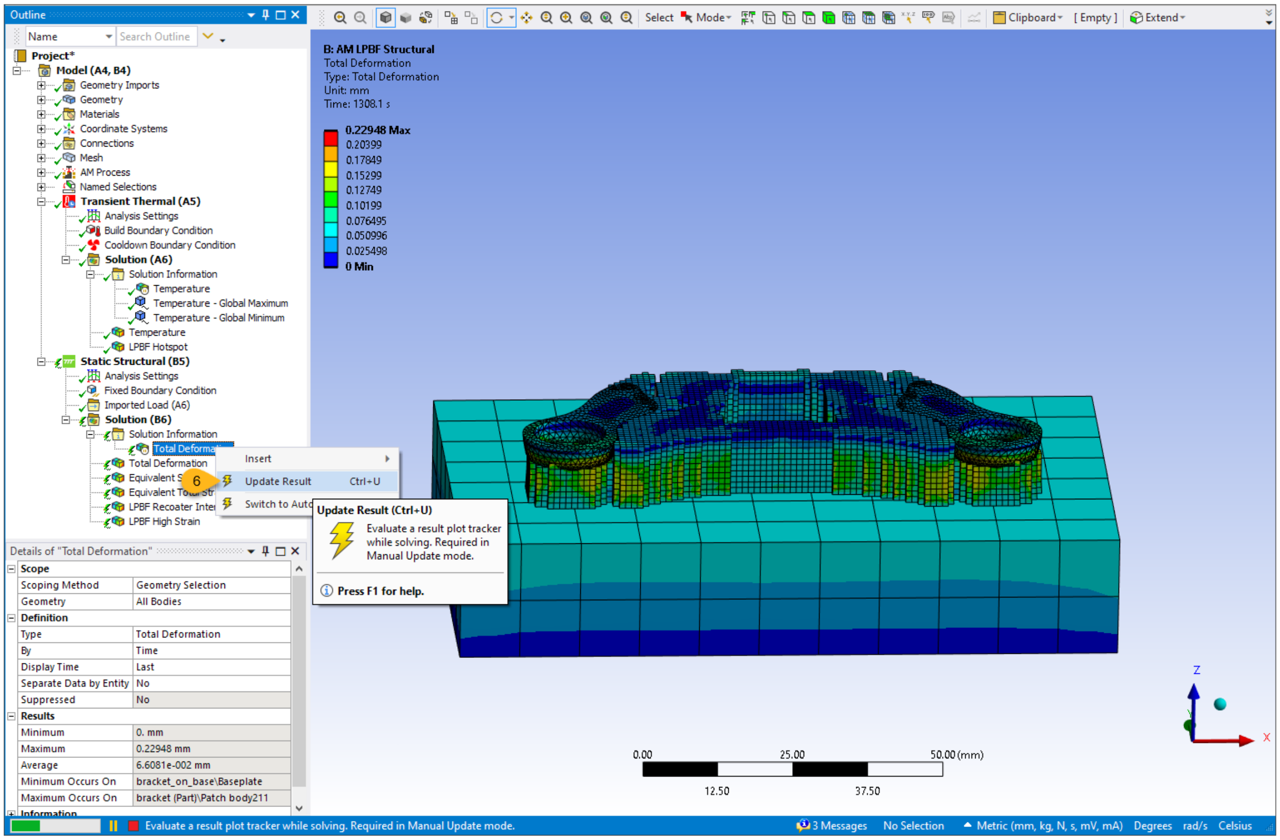
4. Click the **Solve** lightning bolt button in the ribbon to initiate the solution. The Transient Thermal analysis will solve first followed by the Static Structural analysis. Depending on the number of cores specified and the machine used to run this model, the simulation may take a couple hours to complete.



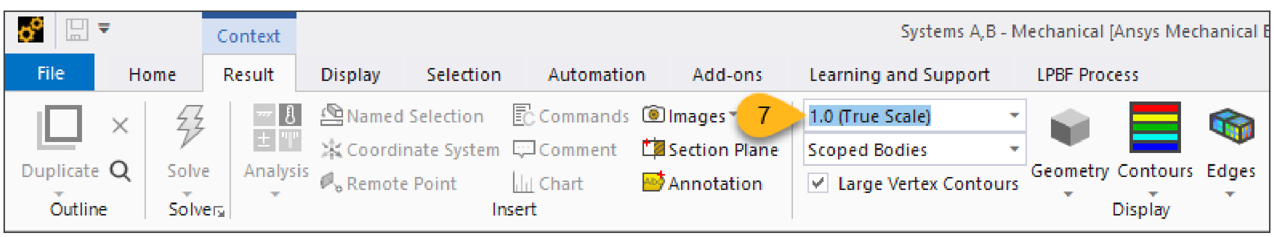
5. (Optional) Occasionally during the thermal solution, right-click the **Temperature** tracker and click **Update Result** to see a live update of temperature results.



6. (Optional) Occasionally during the structural solution, right-click the **Total Deformation** tracker and click **Update Result** to see a live update of deformation results.



7. Under the **Result** tab in the ribbon, change the display scale to **1.0 (True Scale)** for a better view of deformation. By default, the scale is set for an exaggerated display.

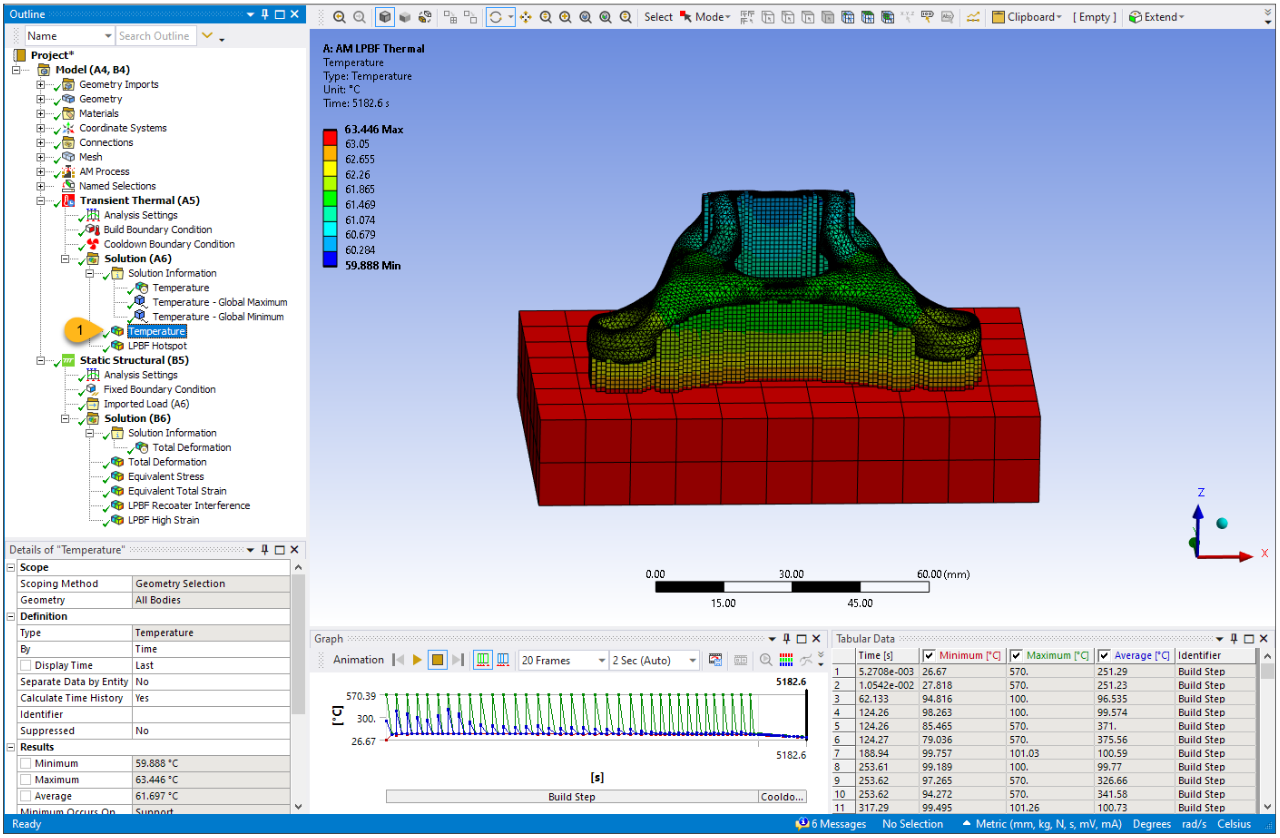


3.9. Review Results

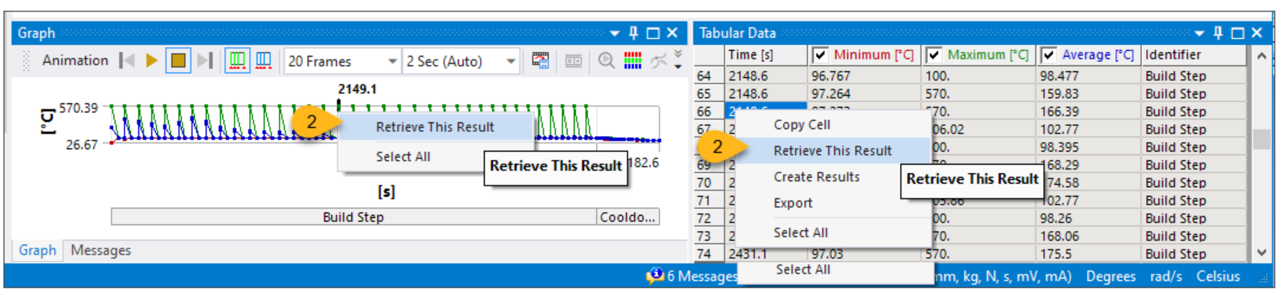
Once the solve is complete, results can be viewed by clicking result items in the tree. We'll start by reviewing the thermal results followed by the structural results.

3.9.1. Thermal Results: Temperature and Hotspot

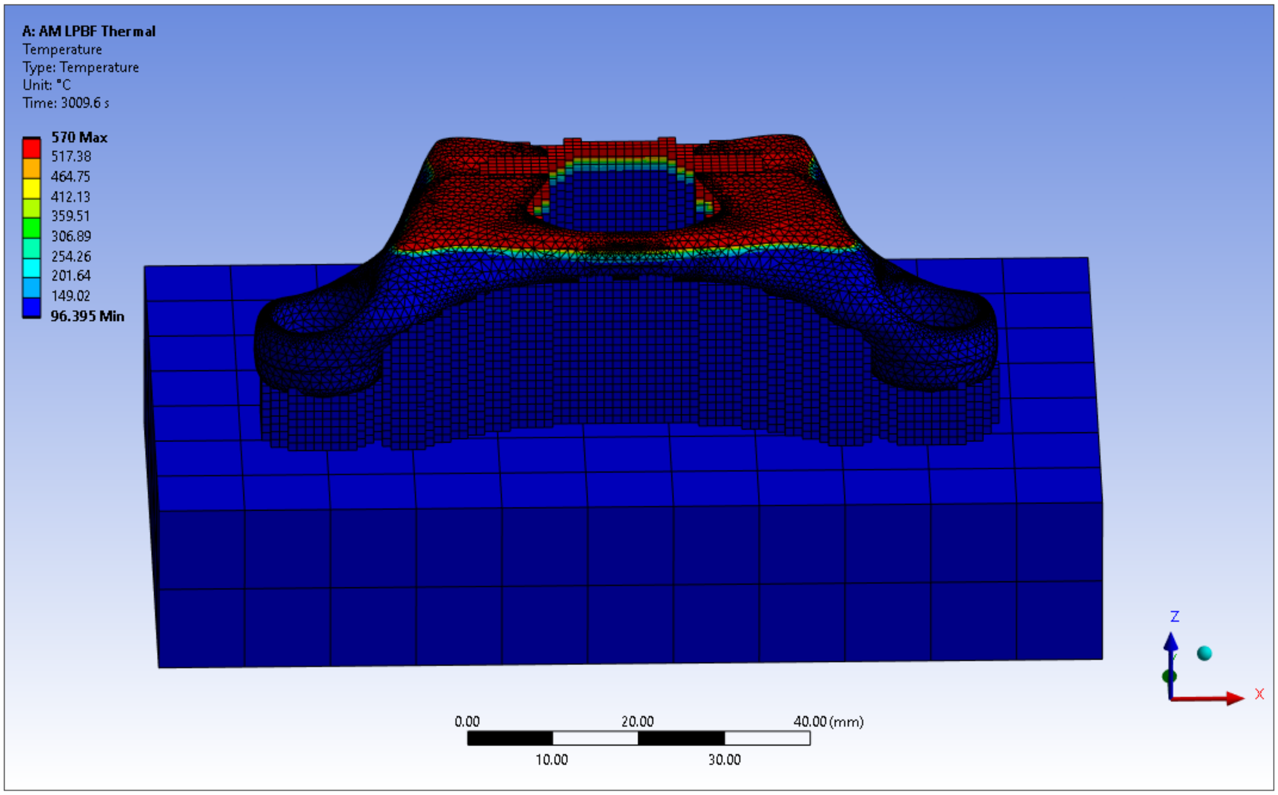
1. Under Transient Thermal > Solution, click **Temperature** to view the temperature results. (Note that the Temperature object under *Solution Information* shows only a snapshot of results whereas the Temperature object under *Solution* shows the full thermal history when the thermal solve is complete.)



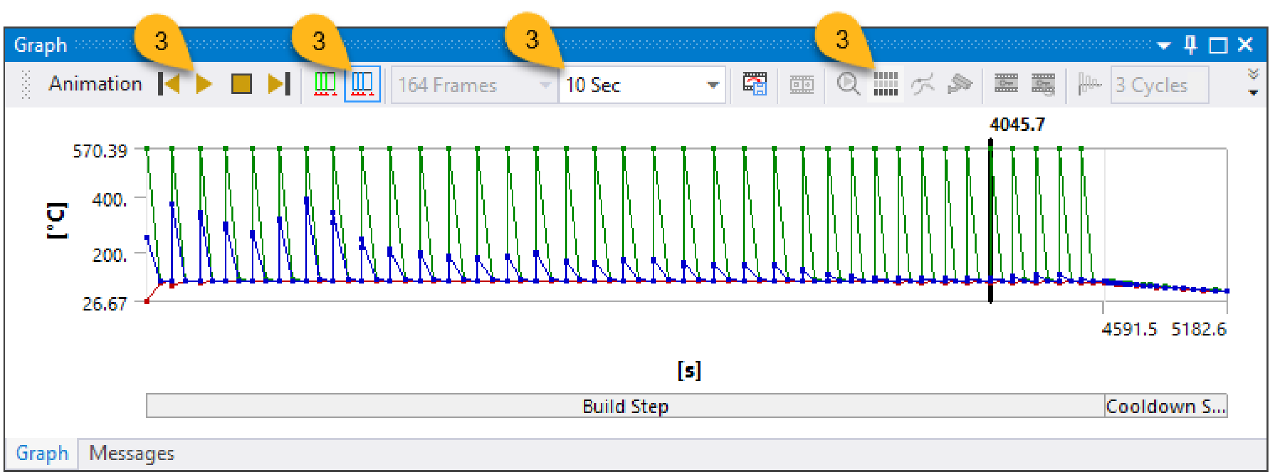
2. To view the result at different time steps, right-click anywhere within the time graph located at the bottom left corner of the screen, or at any time in the tabular data at the bottom right, and select **Retrieve This Result**.

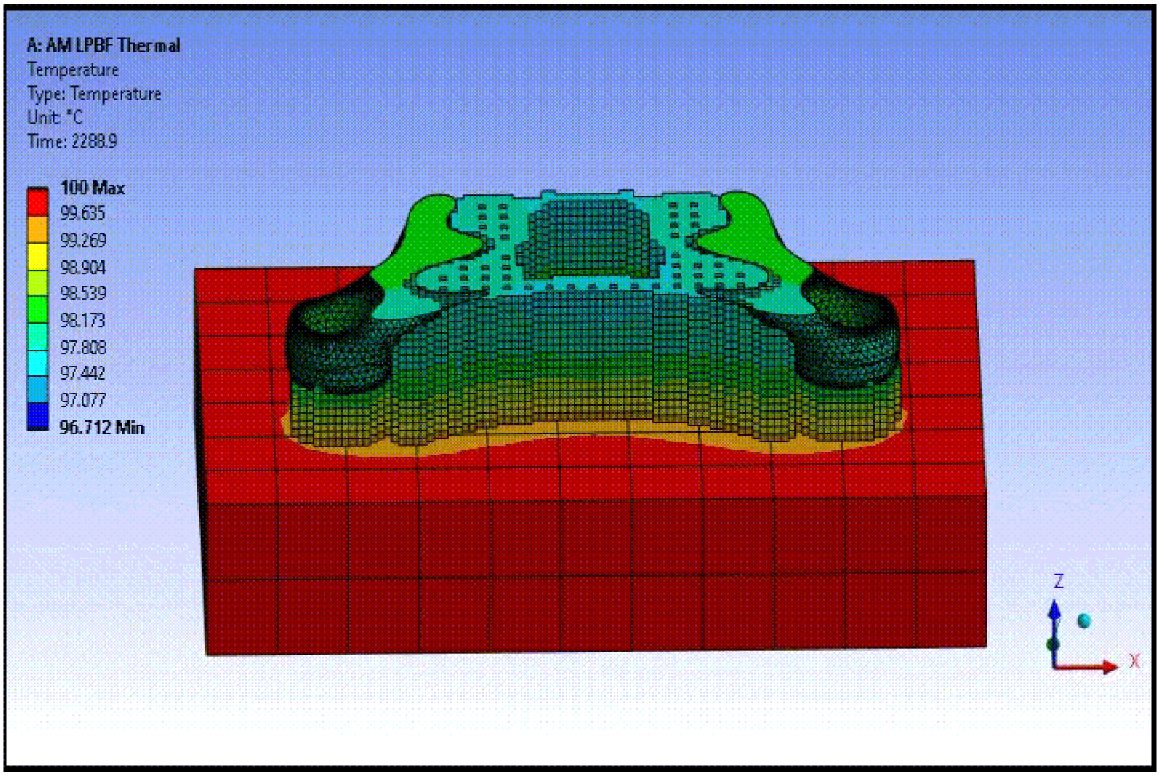


Here is the result at time = 3009.6 seconds, just as a new layer is deposited. You can see the highest temperature is 570°C, which is the melting temperature for AlSi10Mg. The layers below the new layer had already cooled rapidly.



3. Use the animation controls at the top of the Graph window. Click the **Result Sets** button and the **Update Contour Range at Each Animation Frame** button. Adjust the number of seconds for the animation, then click **Play**. See Animation in the *Mechanical User's Guide* for more information about animation controls.





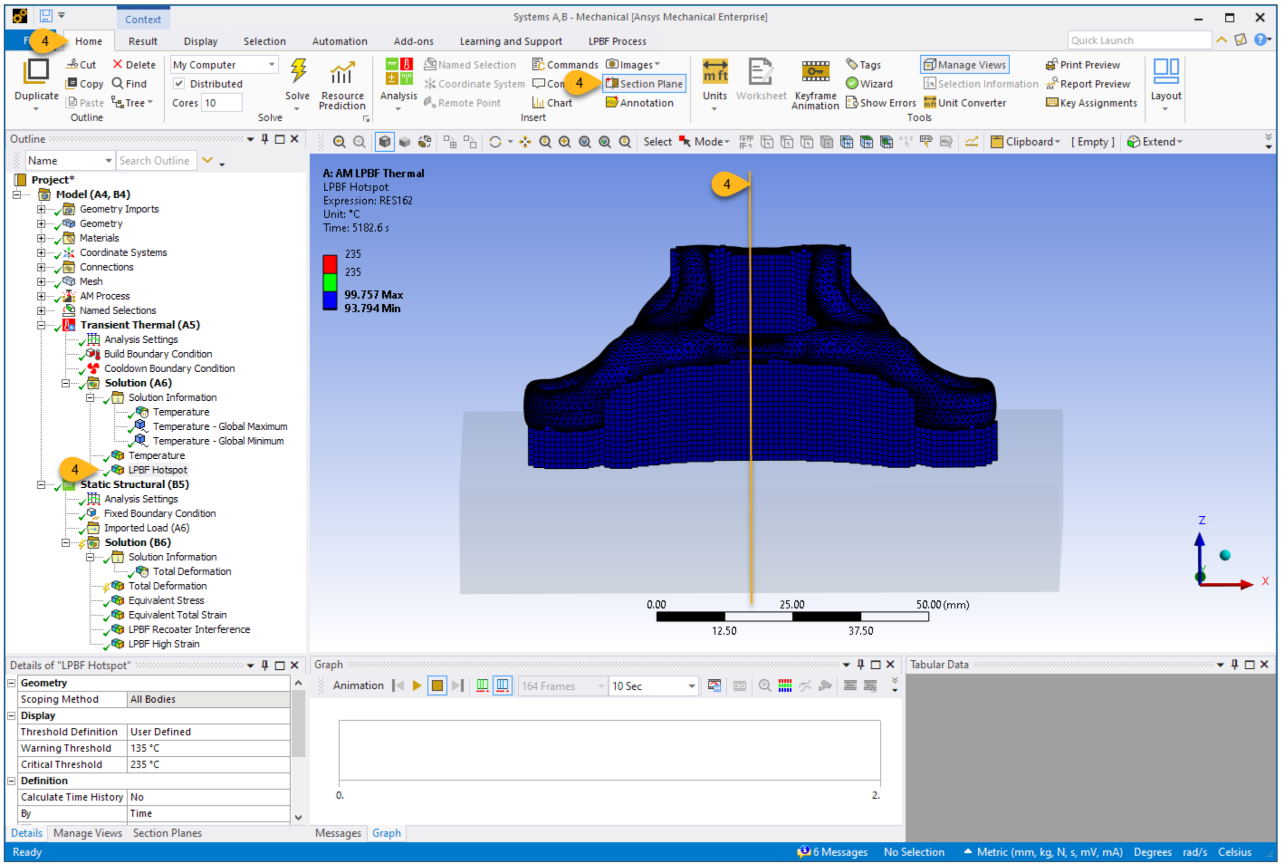
The image above is an animated gif. The animation is not viewable in PDF format.

4. Under Transient Thermal > Solution, click **LPBF Hotspot**. This result shows the temperature that each layer cools down to before a new layer is added. The worst hotspots are going to be the areas with the highest temperature for that layer. This result can reveal areas of overheating that may be of concern. Overheating can lead to poorly shaped melt pools that can affect material properties and porosity in the part.

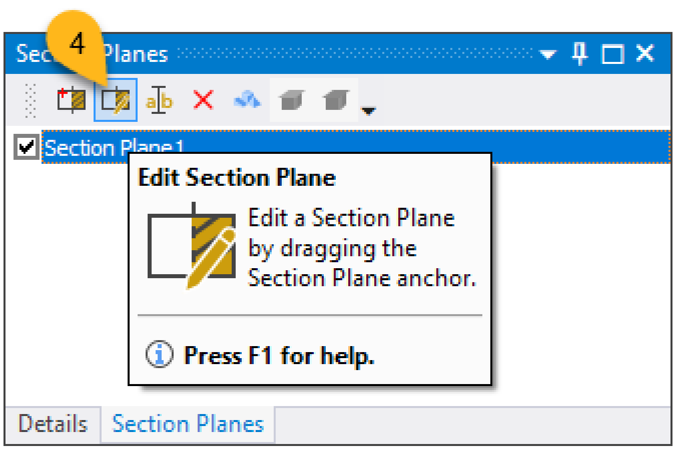
   For this bracket, the entire part looks blue, meaning there are no hotspots to worry about given the threshold criteria we specified. But hotspot results are localized (based on nodal values), not averaged across the layer. We'll use a section plane to reveal any possible hotspots inside the part.

   a. Click the **Home** tab, then **Section Plane**.

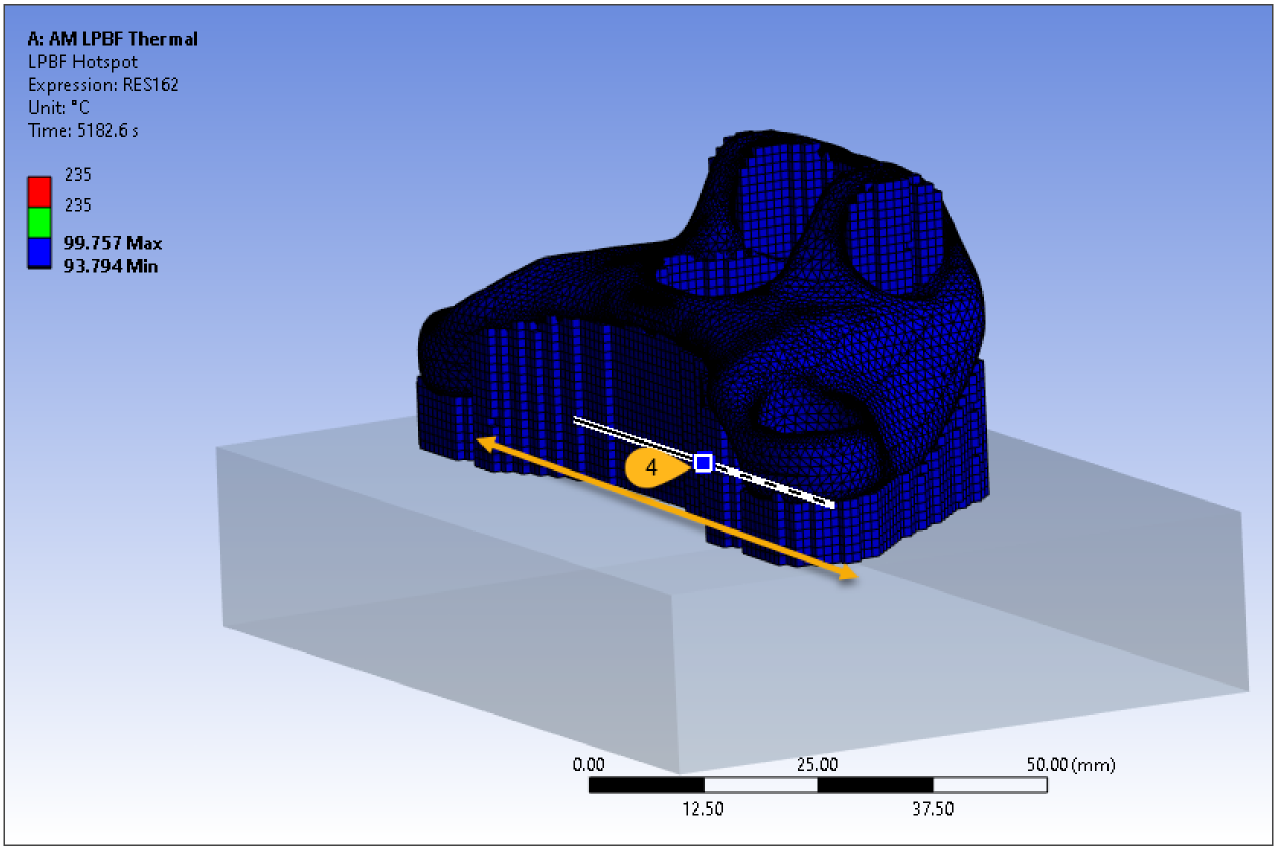
   b. Holding the left mouse button, **draw a line** anywhere through the part to create the section plane. When you release the mouse button, the part appears cut at the plane. Rotate the part so that you can see the interior.



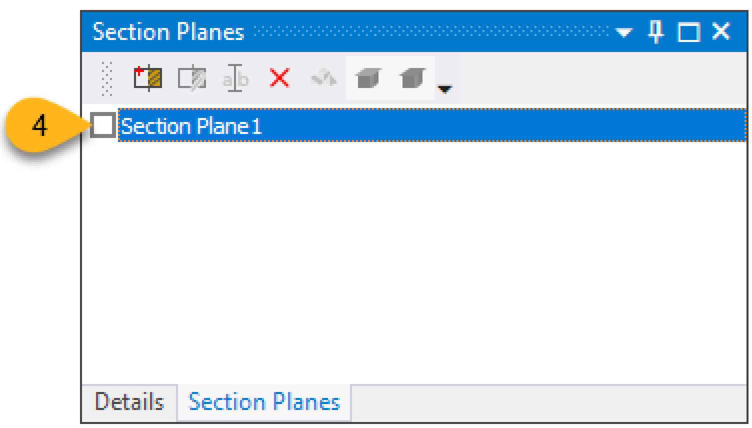
c. In the Section Planes panel at the bottom left corner of the UI, select the newly created section plane and click **Edit Section Plane**.



Drag the section plane anchor back and forth to see the inside of the part. Again, based on our hotspot threshold criteria, there are no green or red areas, indicating that there are no hotspots to be concerned about.



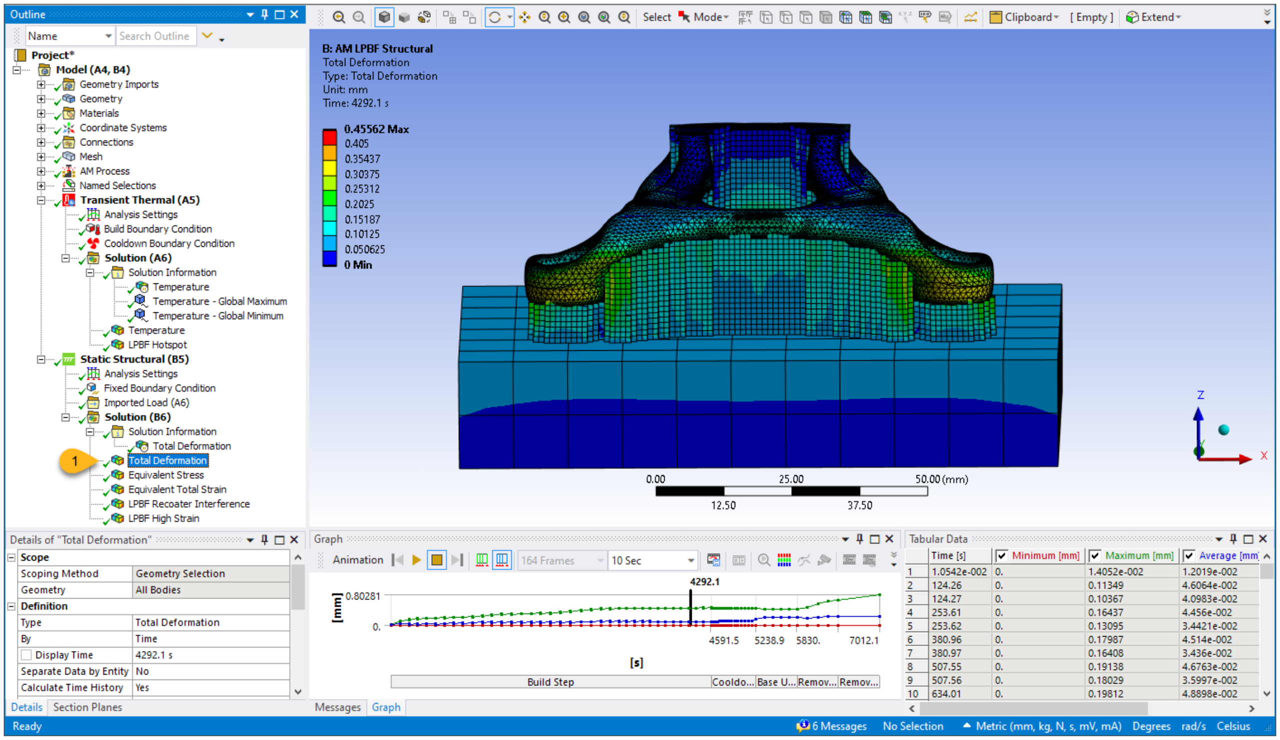
d. Clear the check box in the Section Planes panel to see the entire part again.



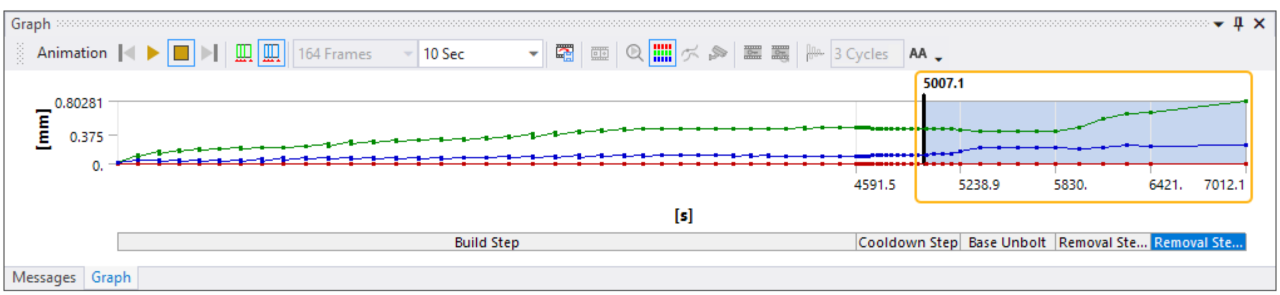
3.9.2. Structural Results: Deformation, Recoater Interference, and High Strain

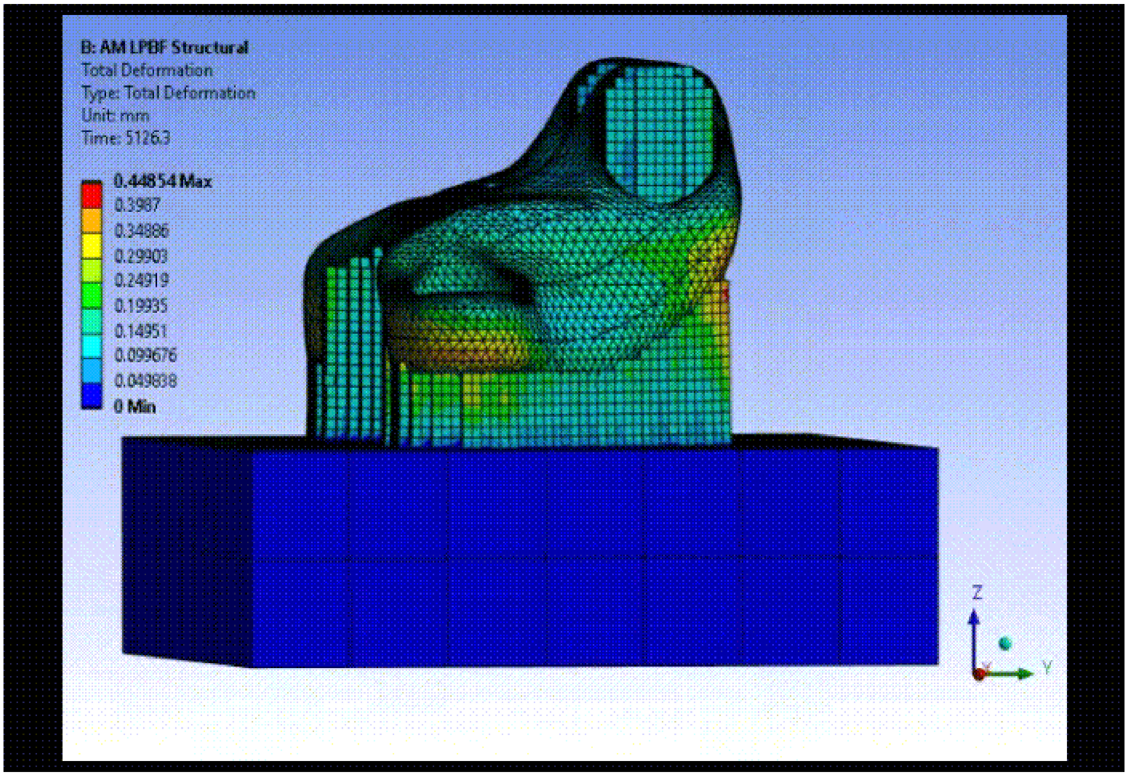
Now let's look at structural results.

1. Under Static Structural > Solution, click **Total Deformation** to view the deformation results.



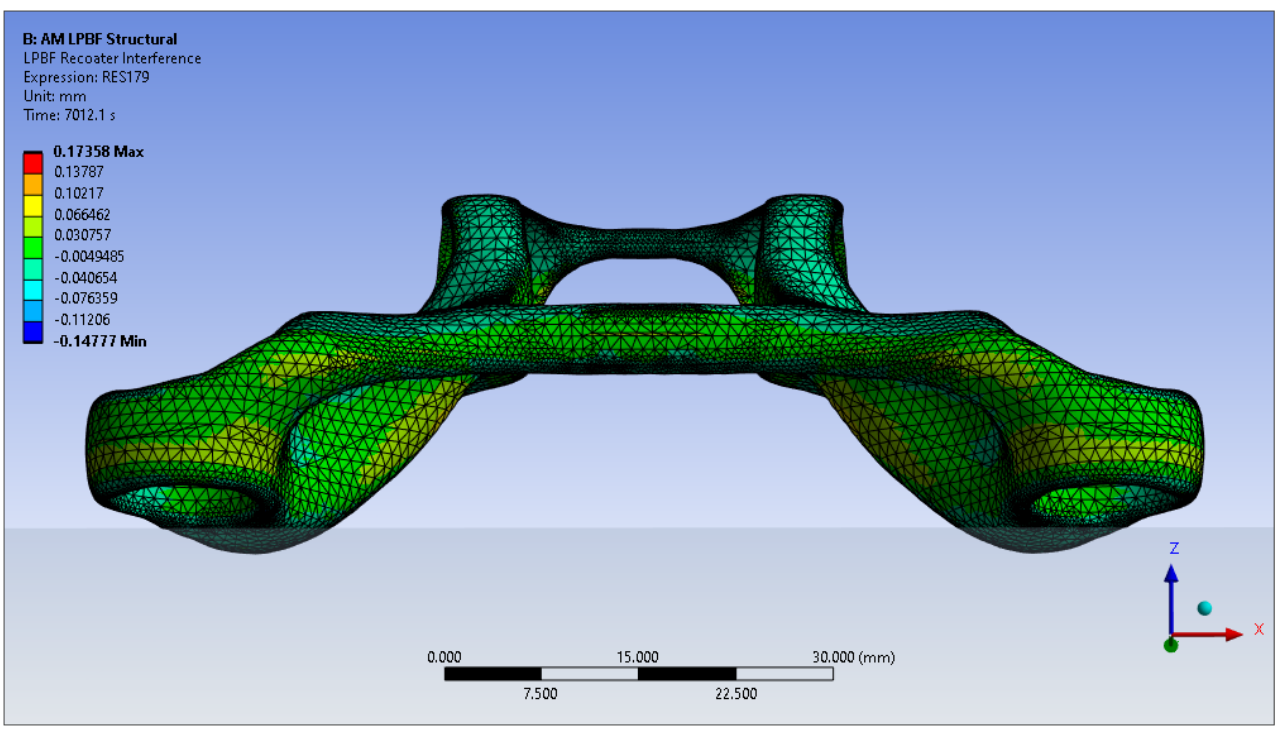
You can zoom in on a specific subset of time steps and animate the results. Below is an animation of total deformation showing just the zoomed-in results which include the last few substeps of the cooldown step, and the base unbolting, base removal, and support removal steps.



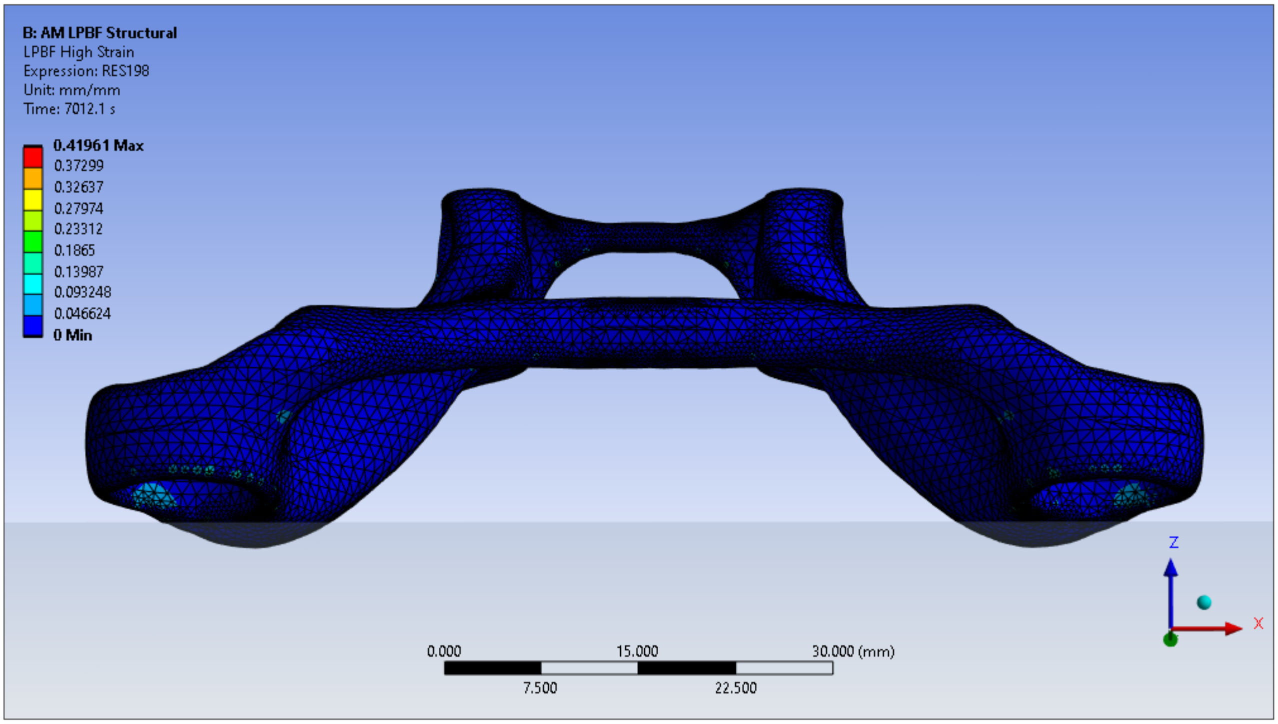


The image above is an animated gif. The animation is not viewable in PDF format.

2. Under Static Structural > Solution, click **LPBF Recoater Interference** to view recoater interference results.



3. Under Static Structural > Solution, click **LPBF High Strain** to view the high strain results.



Congratulations! You have completed the tutorial.

Chapter 4: DED Simulation - Racetrack

This tutorial demonstrates how to perform a Directed Energy Deposition (DED) process simulation using the DED Process Add-on in Mechanical. The following table shows the features used.

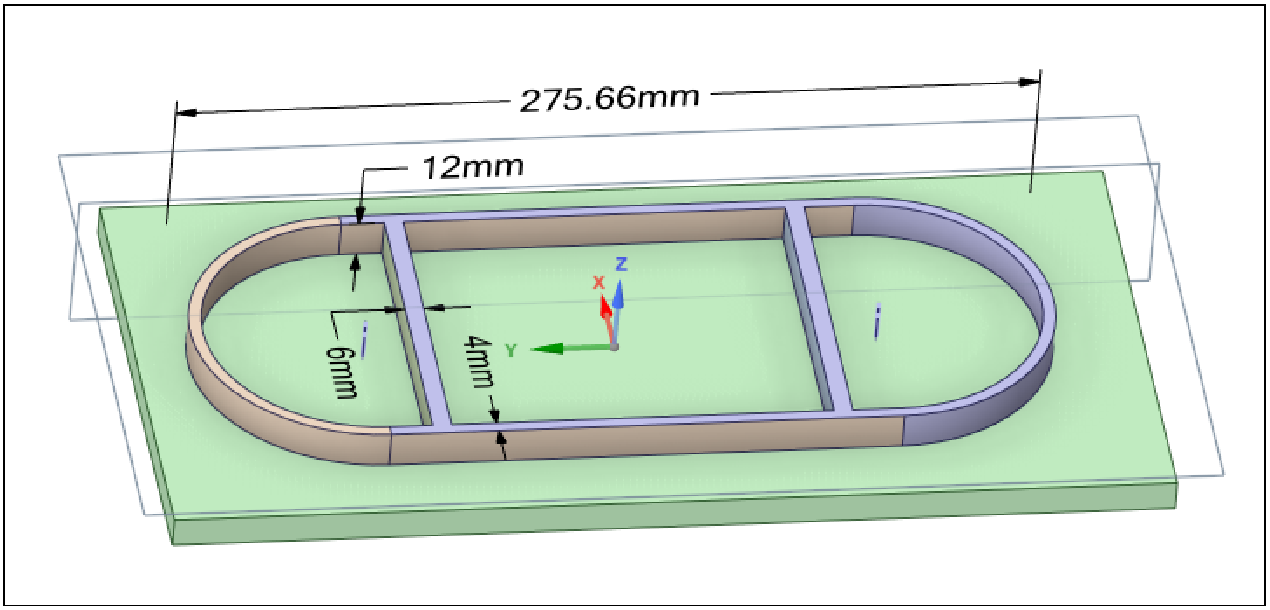
Applicable products:	Workbench/Mechanical An Additive Suite license is required in addition to the license for Mechanical.
Level of difficulty:	Easy
Interactive time required:	10 minutes to set up, 55 - 60 minutes to solve, 10 minutes to review results[a]
Manufacturing process:	Directed Energy Deposition (DED)
Simulation/system type:	AM DED Process
Material:	Inconel 718
Geometry:	One part on planar base plate
Mesh:	Cartesian elements
AM steps simulated:	Build and cooldown
Features demonstrated:	DED Process Add-on, DED Setup Wizard, G-Code file defining tool path, G-Code clustering approach
Help resources:	Introduction to DED Additive Manufacturing

[a] This is an approximate range. The amount of time it takes you to complete the tutorial depends on the computer system and the number of CPU cores you use, the working pace that is comfortable for you, and so on.

Tutorial steps:

4.1. Problem Description

We will simulate the DED printing process of this racetrack-shaped model, shown here on a planar baseplate. The print direction is in the global Z-direction. A G-Code file is used to control the tool path and we will use that for element clustering. The geometry is sliced such that there are four layers through the height of the track.



Inputs	
Material	Inconel 718
Deposition Thickness (mm)	3
Deposition Rate (mm^3/s)	72
Preheat Temperature (°C)	80
Process Temperature (°C)	1370

Tutorial Files

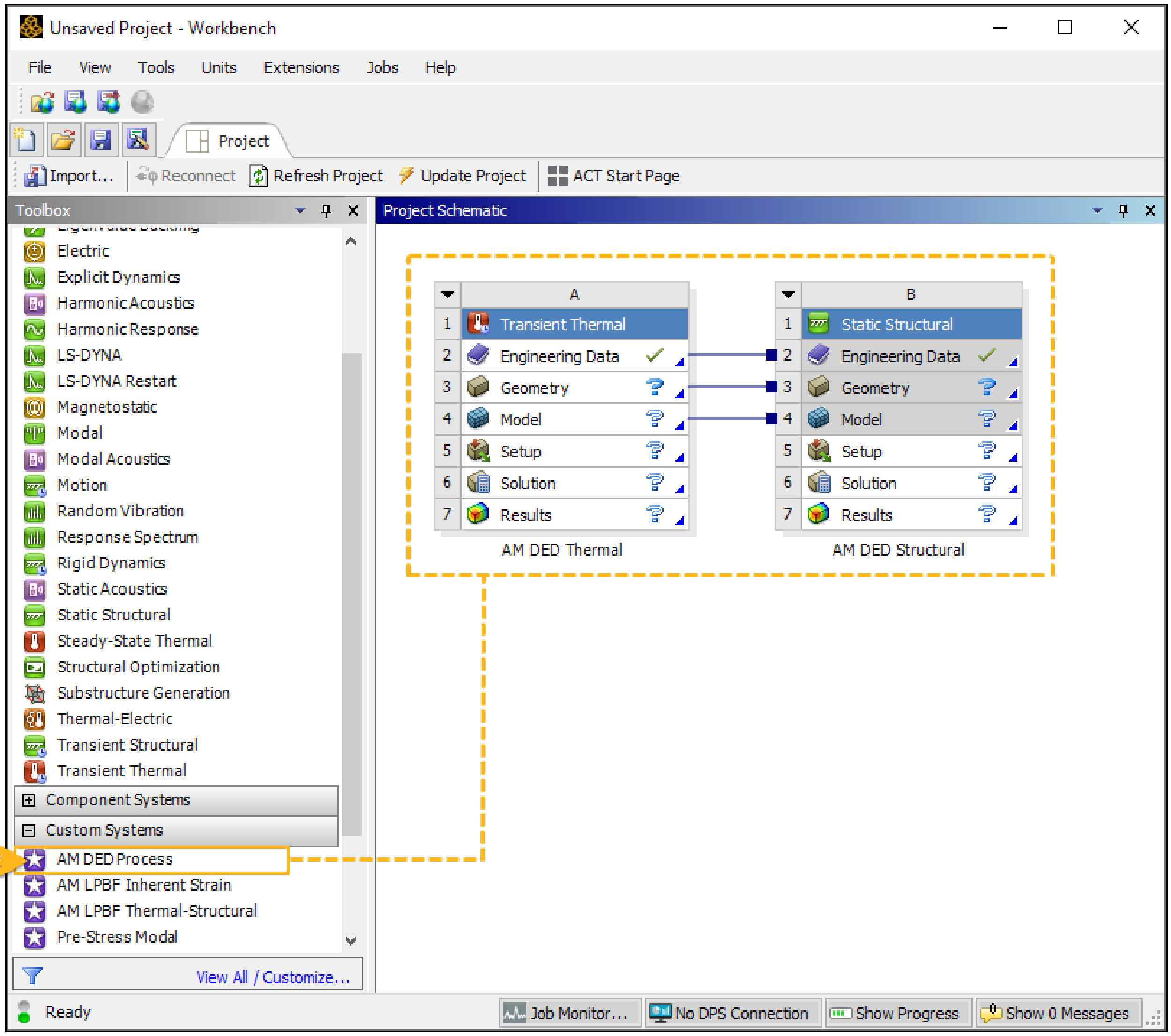
Click here to download the following:

- `DED_Racetrack.scdoc` — Geometry file of the racetrack part and a base plate, saved as an Ansys SpaceClaim document.
- `DED_Racetrack_G-Code.txt` — G-Code machine file that controls the build order.

4.2. Create the Analysis System

1. Open Ansys Workbench.

2. In the toolbox on the left side of the window, scroll down to Custom Systems, expand the selection, and double-click **AM DED Process** to bring up the linked Transient Thermal, Static Structural system in the Project Schematic.

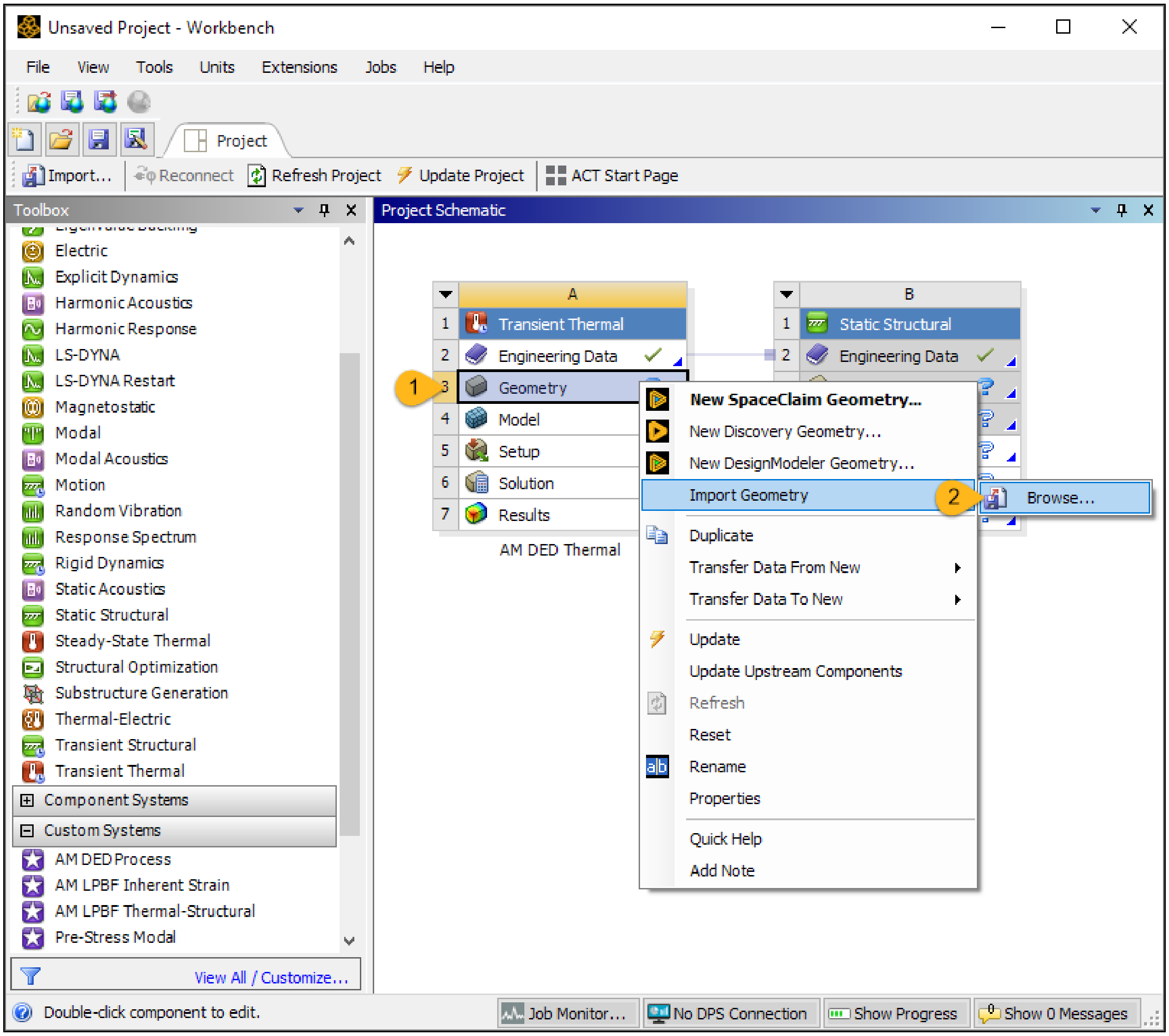


4.3. Import Geometry and Launch Mechanical

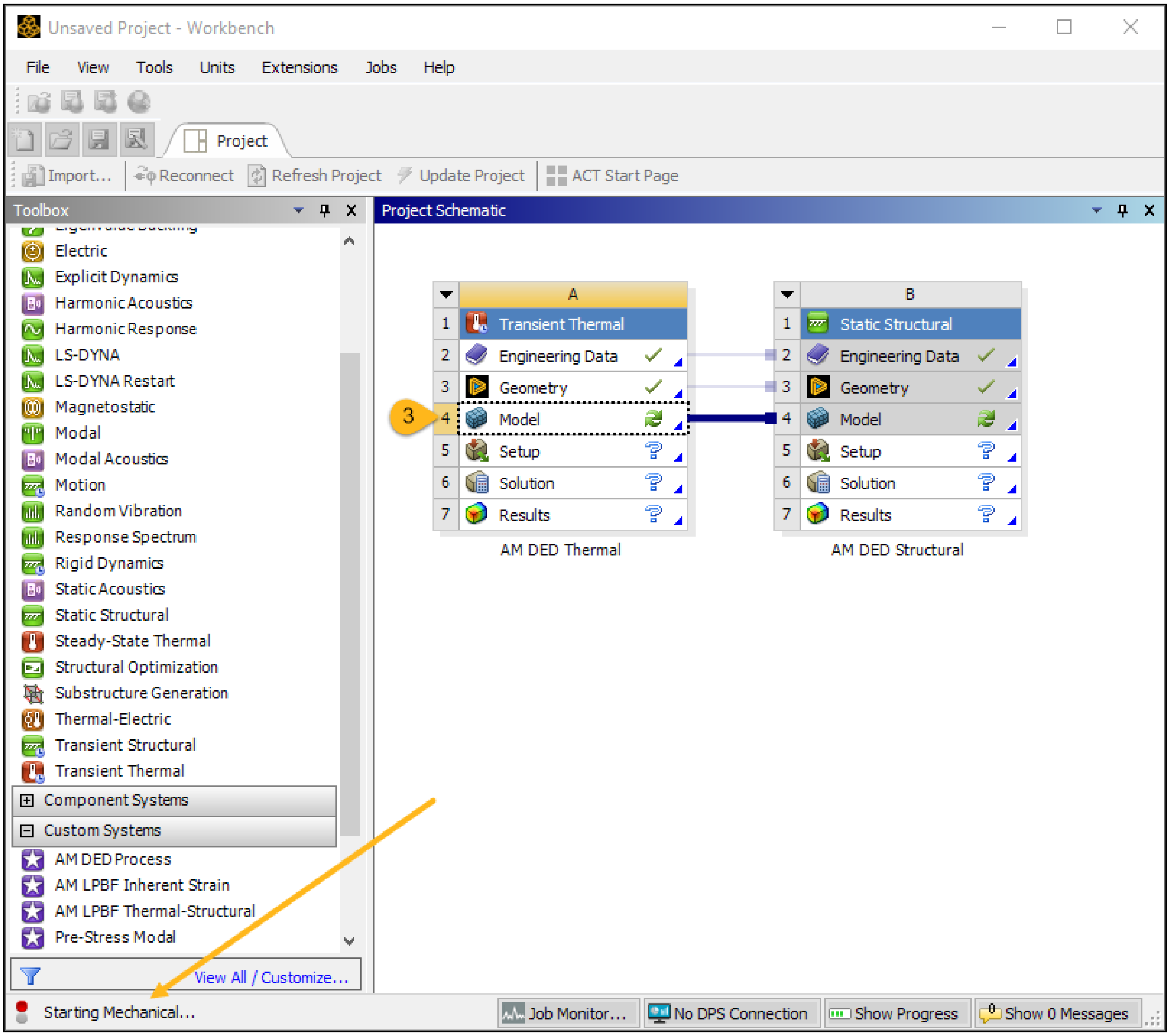
In this step, we import the geometry file in Workbench and then open the Mechanical application.

1. Right-click the **Geometry** cell in the Transient Thermal system and select **Import Geometry** > **Browse**.

2. Navigate to the racetrack geometry file, select it, and click **Open** to add it to the analysis. A green checkmark appears next to the Geometry cell in the Project Schematic when the geometry is added.



3. Double-click the **Model** cell in the Transient Thermal system to launch the Mechanical application. A message "Starting Mechanical" appears in the status bar in the bottom, left corner. It may take several seconds for the application to open and attach the geometry.



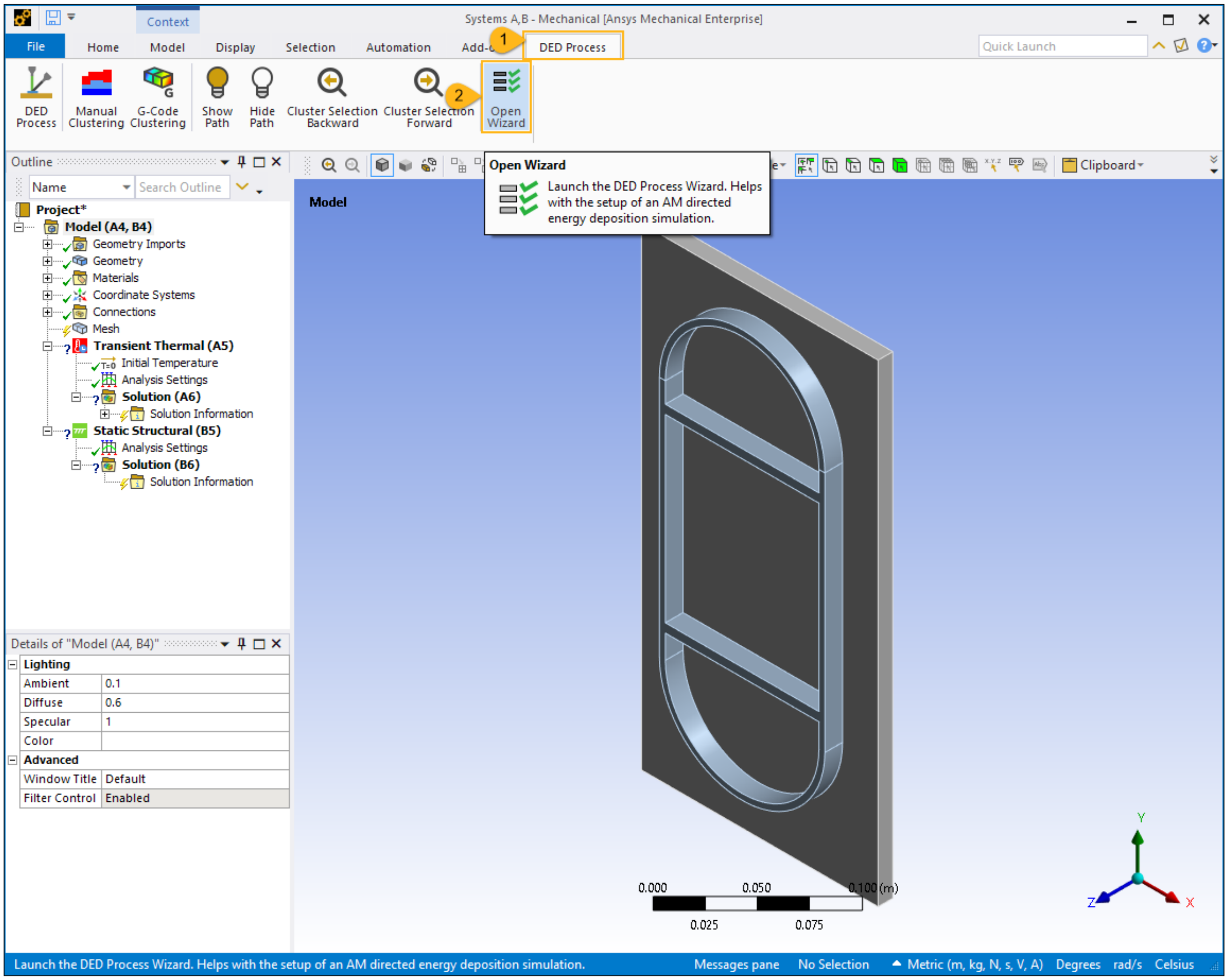
Once the Mechanical application is open, look for a "Ready" message in the status bar.

4.4. Open the DED Process Wizard

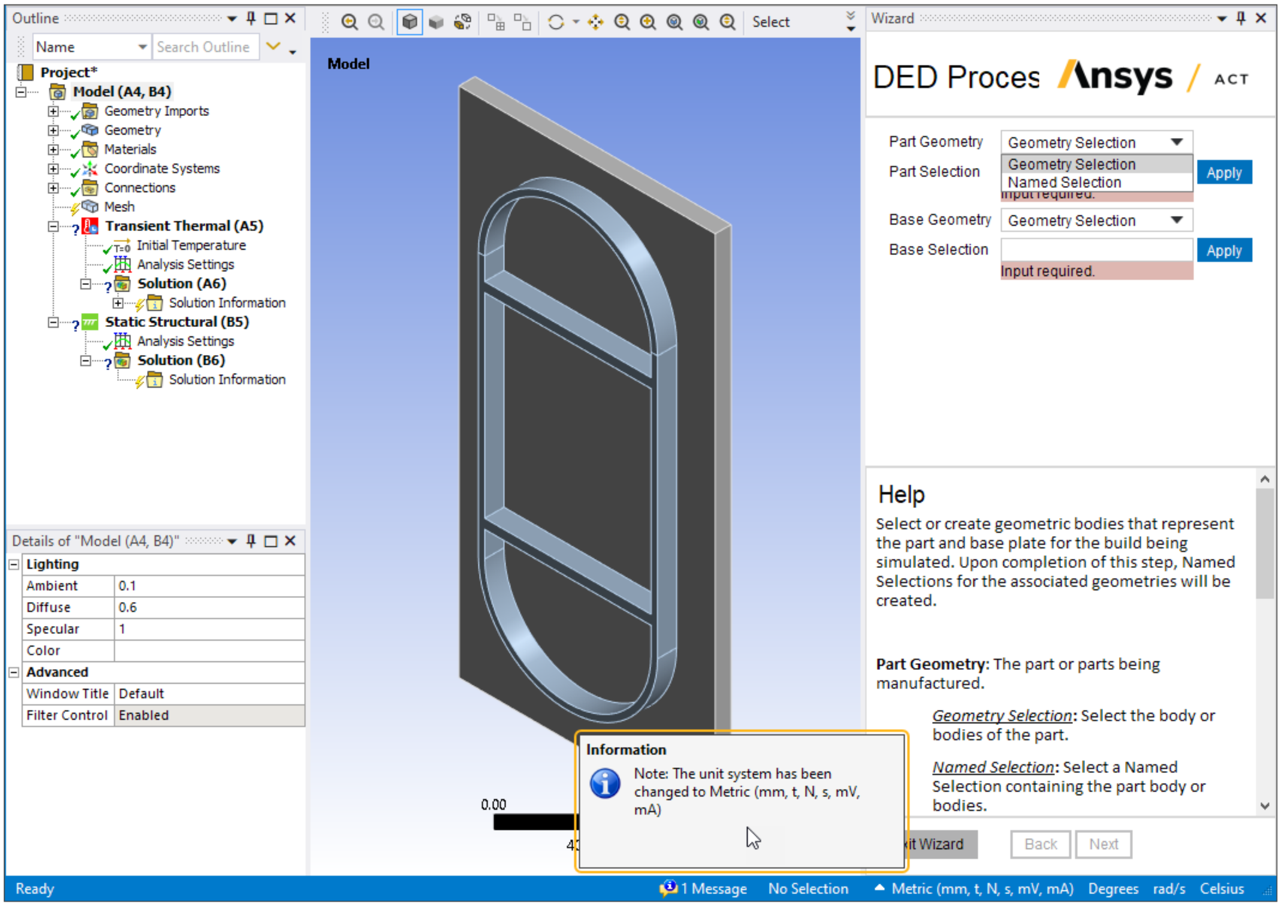
The DED Process Add-on is loaded automatically when you use the AM DED Process custom system.

1. Click the **DED Process** tab at the top of the user interface to access the add-on's custom ribbon.

2. Click **Open Wizard** from the ribbon to open the wizards panel.



Upon opening the wizard, the units are automatically changed to millimeters, as this unit system is required for simulations with the DED Process Add-on.



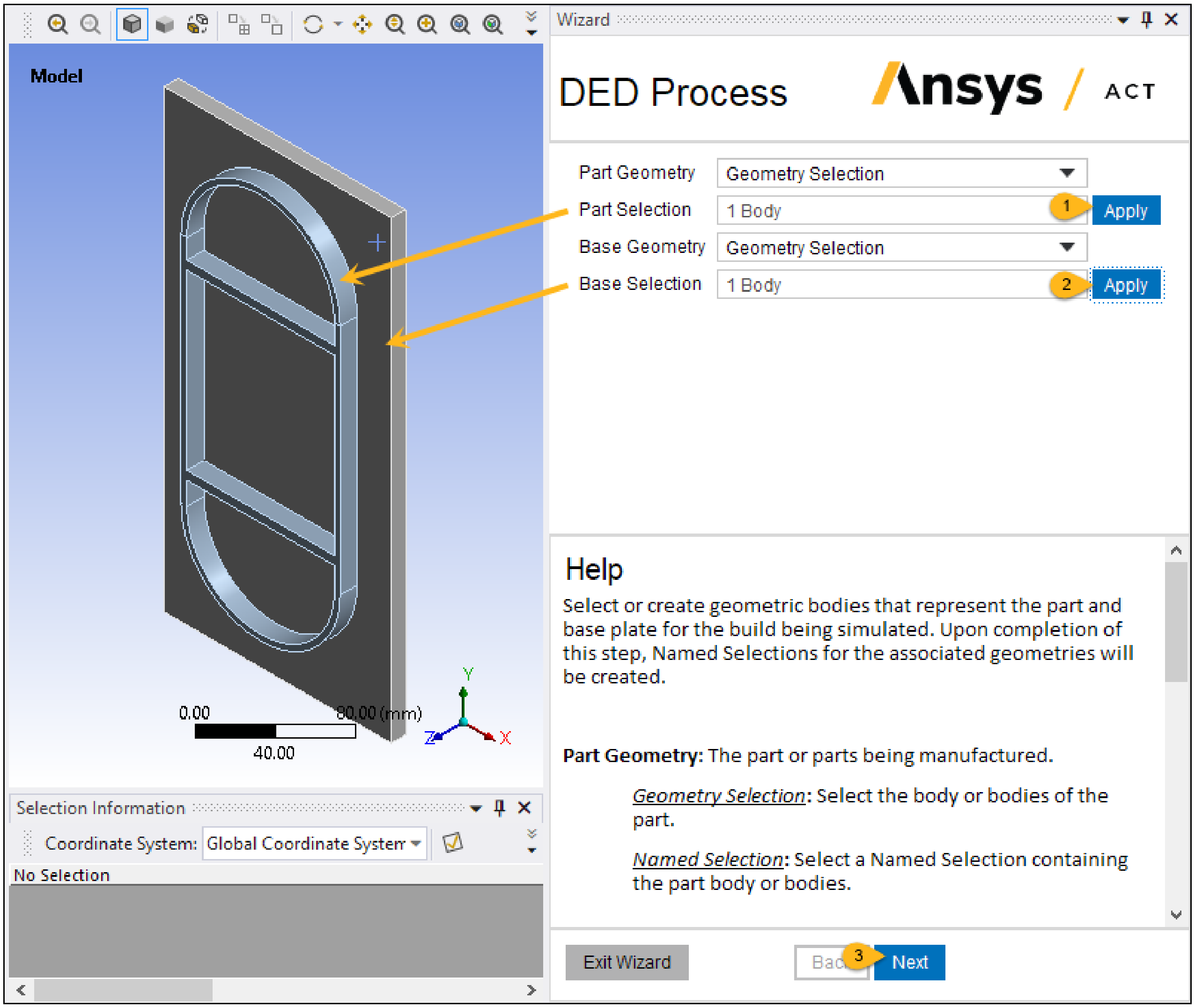
4.5. Wizard Step 1 - Identify Geometries

Identify which body is the part and which body is the base.

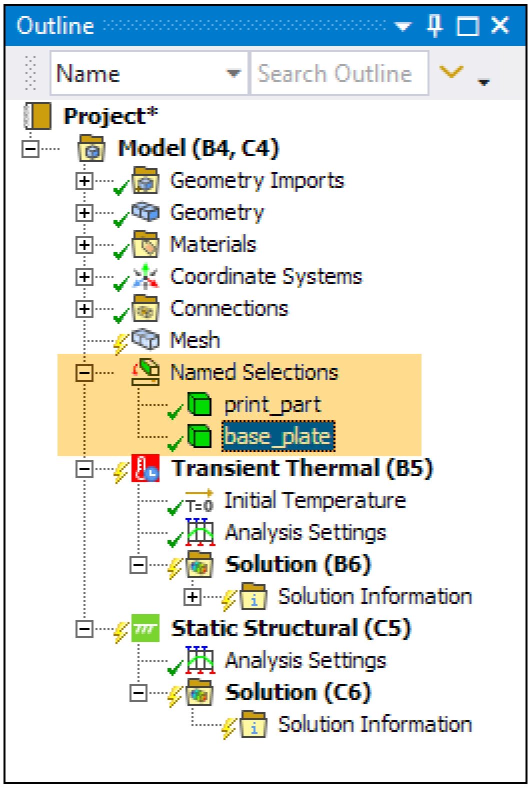
1. Select the racetrack body and click **Apply** in the Part Selection field. (Note the Body picker  is active by default.)

2. Select the base body and click **Apply** in the Base Selection field.

3. Click **Next** to move to the next step.



When this step is completed, Named Selections for the part body (print_part) and base body (base_plate) have been added to the Project tree. These will be used later for the creation of other Named Selections.



4.6. Wizard Step 2 - Generate Mesh

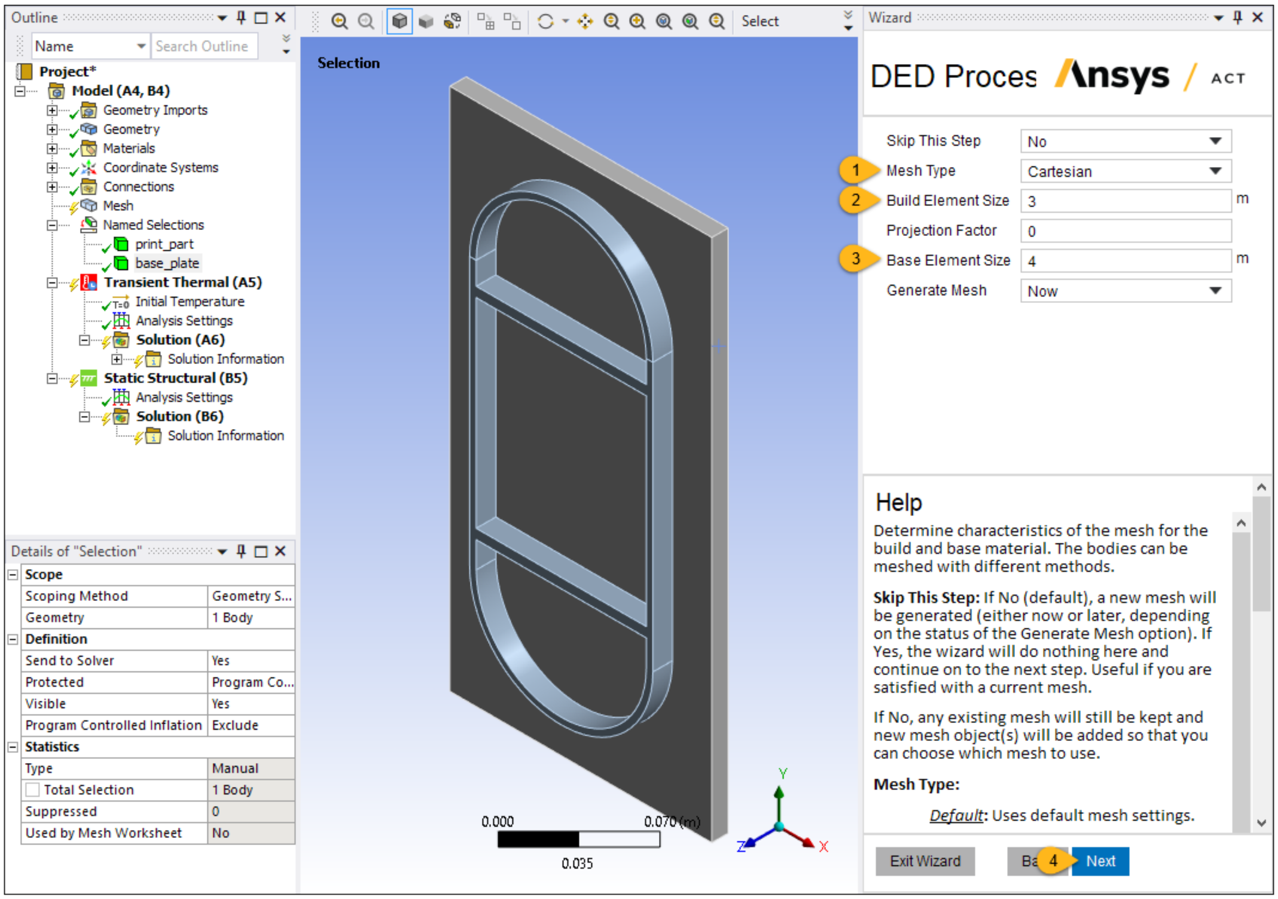
Because this is a simple geometry, we will mesh the part with Cartesian elements. We will use an element size of 3 mm for the part to match the deposition thickness. Reviewing the G-Code file below shows the first layer has a Z-coordinate of 3.0, the second layer's Z-coordinate is 6.0, and so on. This will result in 4 layers through the height of the model. We will mesh the base with standard brick elements with a slightly larger element size of 4 mm.

```
G00 X-100.0 Y-83.5 Z0.0
; First layer
G00 X-54.0 Y-83.5 Z3.0
G01 X-49.9 Y-104.2 ; Circle 1
G01 X-38.2 Y-121.7
G01 X-20.7 Y-133.4
G01 X0.0 Y-137.5
G01 X20.7 Y-133.4
G01 X38.2 Y-121.7
G01 X49.9 Y-104.2
G01 X54.0 Y-83.5
G01 X54.0 Y80.2 ; Y-Line 1
G01 X49.9 Y100.9 ; Circle 2
G01 X38.2 Y118.4
G01 X20.7 Y130.1
G01 X0.0 Y134.2
G01 X-20.7 Y130.1
G01 X-38.2 Y118.4
G01 X-49.9 Y100.9
G01 X-54.0 Y80.2
G01 X-54.0 Y-83.5 ; Y-Line 2
G00 X-54.0 Y-64.6
G01 X54.0 Y-64.6 ; X-Line 1
G00 X-54.0 Y63.4
G01 X54.0 Y63.4 ; X-Line 2
; Next layer
```

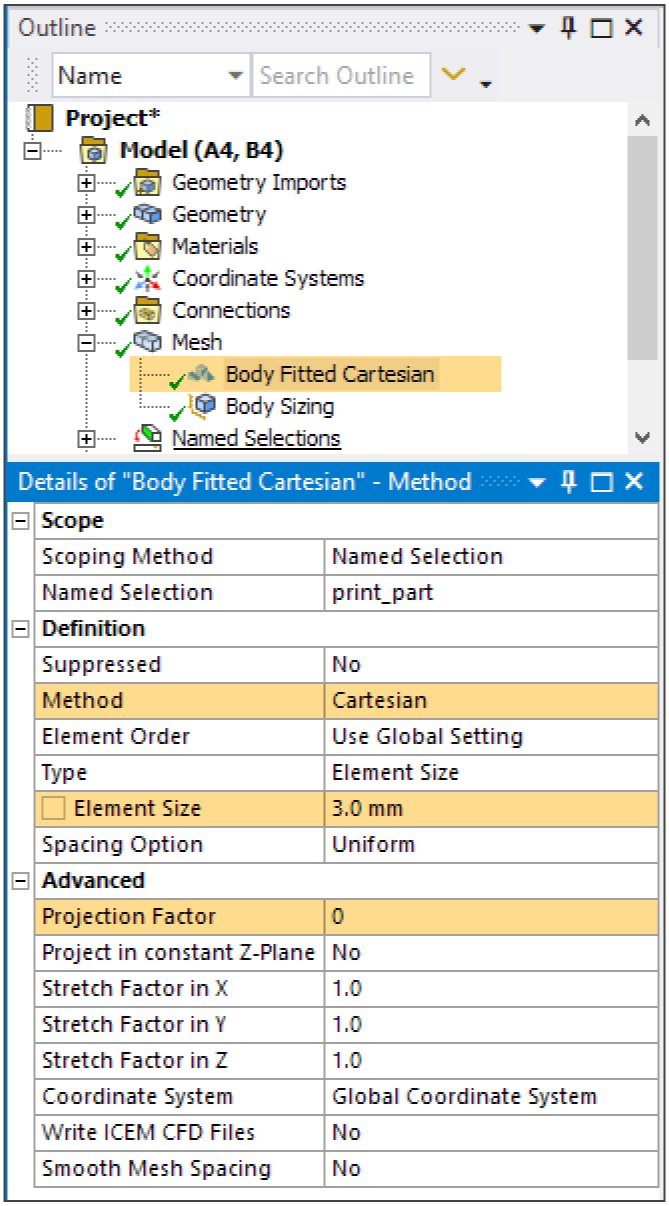
```
G00 X-54.0 Y-83.5 Z6.0
G01 X-49.9 Y-104.2 ; Circle 1
G01 X-38.2 Y-121.7
G01 X-20.7 Y-133.4
G01 X0.0 Y-137.5
G01 X20.7 Y-133.4
G01 X38.2 Y-121.7
G01 X49.9 Y-104.2
G01 X54.0 Y-83.5
G01 X54.0 Y80.2 ; Y-Line 1
G01 X49.9 Y100.9 ; Circle 2
G01 X38.2 Y118.4
G01 X20.7 Y130.1
G01 X0.0 Y134.2
G01 X-20.7 Y130.1
G01 X-38.2 Y118.4
G01 X-49.9 Y100.9
G01 X-54.0 Y80.2
G01 X-54.0 Y-83.5 ; Y-Line 2
G00 X-54.0 Y-64.6
G01 X54.0 Y-64.6 ; X-Line 1
G00 X-54.0 Y63.4
G01 X54.0 Y63.4 ; X-Line 2
; Next layer
G00 X-54.0 Y-83.5 Z9.0
G01 X-49.9 Y-104.2 ; Circle 1
G01 X-38.2 Y-121.7
G01 X-20.7 Y-133.4
G01 X0.0 Y-137.5
G01 X20.7 Y-133.4
G01 X38.2 Y-121.7
G01 X49.9 Y-104.2
G01 X54.0 Y-83.5
G01 X54.0 Y80.2 ; Y-Line 1
G01 X49.9 Y100.9 ; Circle 2
G01 X38.2 Y118.4
G01 X20.7 Y130.1
G01 X0.0 Y134.2
G01 X-20.7 Y130.1
G01 X-38.2 Y118.4
G01 X-49.9 Y100.9
G01 X-54.0 Y80.2
G01 X-54.0 Y-83.5 ; Y-Line 2
G00 X-54.0 Y-64.6
G01 X54.0 Y-64.6 ; X-Line 1
G00 X-54.0 Y63.4
G01 X54.0 Y63.4 ; X-Line 2
; Next layer
G00 X-54.0 Y-83.5 Z12.0
G01 X-49.9 Y-104.2 ; Circle 1
G01 X-38.2 Y-121.7
G01 X-20.7 Y-133.4
G01 X0.0 Y-137.5
G01 X20.7 Y-133.4
G01 X38.2 Y-121.7
G01 X49.9 Y-104.2
G01 X54.0 Y-83.5
G01 X54.0 Y80.2 ; Y-Line 1
G01 X49.9 Y100.9 ; Circle 2
G01 X38.2 Y118.4
G01 X20.7 Y130.1
G01 X0.0 Y134.2
G01 X-20.7 Y130.1
G01 X-38.2 Y118.4
G01 X-49.9 Y100.9
G01 X-54.0 Y80.2
G01 X-54.0 Y-83.5 ; Y-Line 2
G00 X-54.0 Y-64.6
G01 X54.0 Y-64.6 ; X-Line 1
G00 X-54.0 Y63.4
G01 X54.0 Y63.4 ; X-Line 2
```

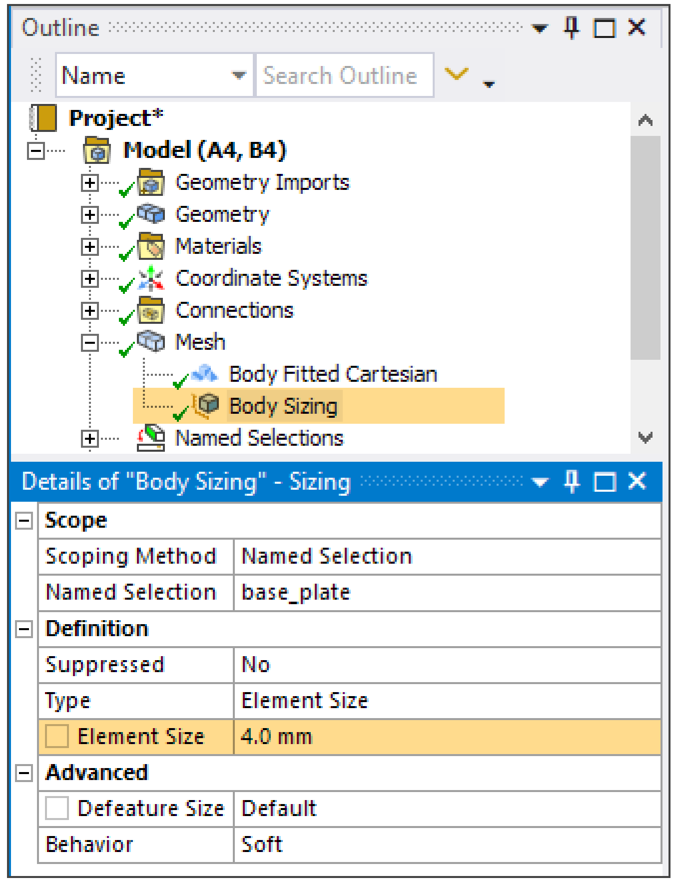
1. Choose **Cartesian** for Mesh Type.
2. Enter **3** for Build Element Size.

   Keep 0 (default) for Projection Factor.
3. Enter **4** for Base Element Size.
4. Click **Next** to move to the next step.

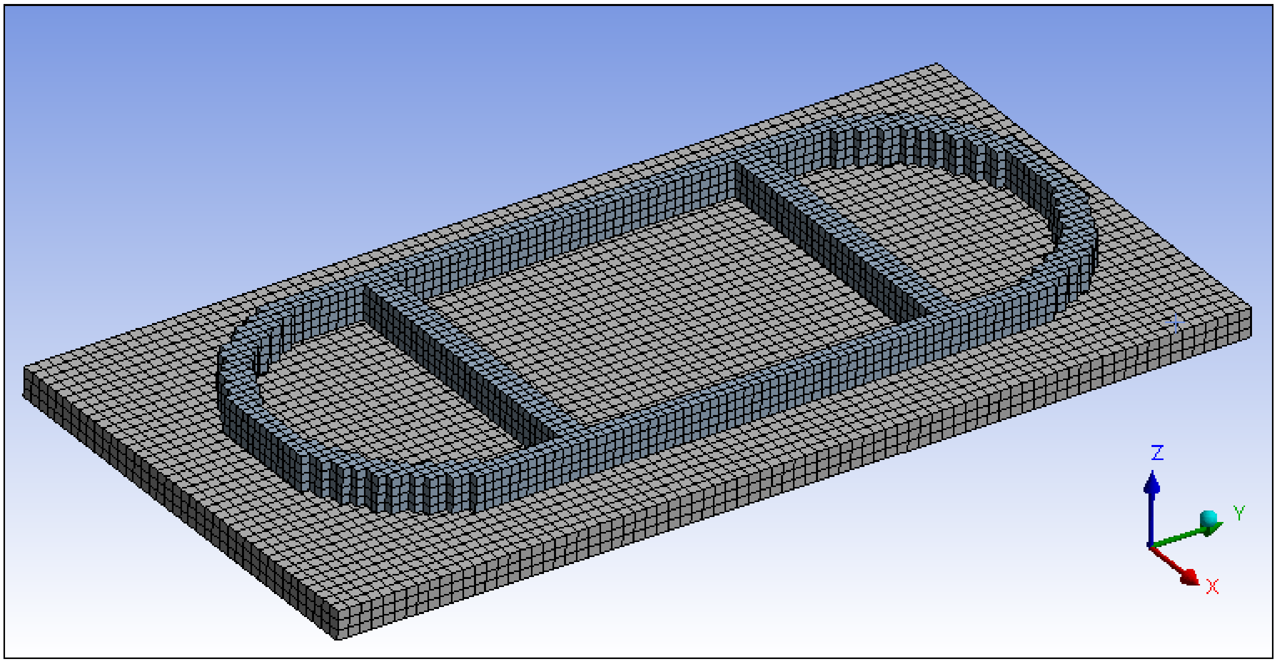


When this step is completed, mesh objects have been added to the Project tree: Body Fitted Cartesian object for the part and Body Sizing for the base plate.





The resulting mesh is shown here.



See the following topics in the *DED Simulation Guide* for additional information:

- Apply Mesh Controls and Generate Mesh

4.7. Wizard Step 3 - Set Up for Element Clustering and Create Contact Connections

In this step, we will set up for element clustering. Note that the actual *generation* of element clusters is not performed in the wizard; we will generate clusters after we finish the wizard.

Also in this step, the contact connection between the part and the base plate is made automatically. Connections ensure that the part and base bodies in the simulation are aware of each other and are able to share data (temperatures and displacements) across boundaries.

1. Choose **G-Code Clustering** for Input Source because we have a G-Code file defining the tool path.

2. Click **Edit** and browse to the DED_Racetrack_G-Code.txt file. Click **Open**.

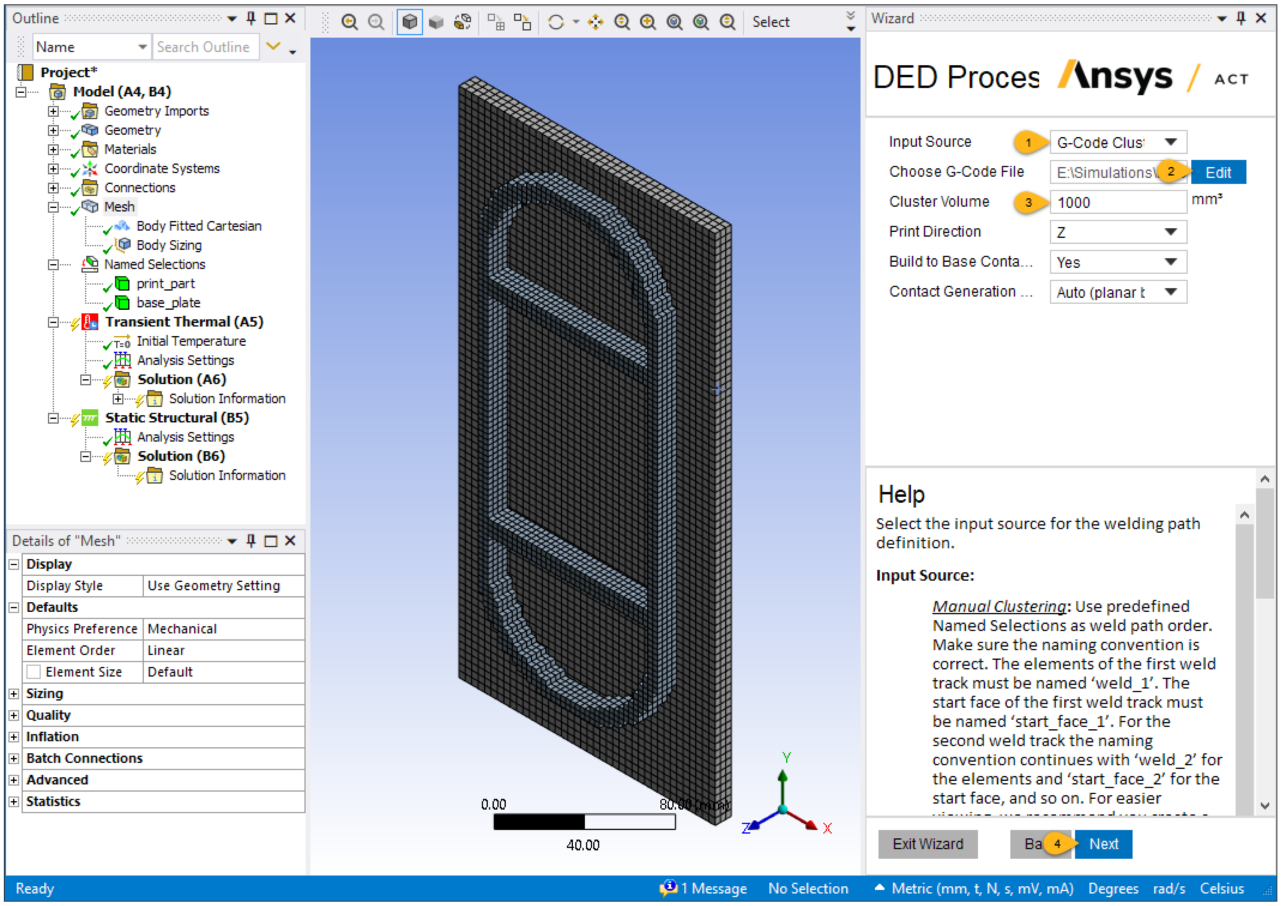
3. Enter **1000** for Cluster Volume. This value determines how many elements are activated per load step. The time for this load step is then determined by volume/deposition rate. A smaller cluster volume tends to provide a more accurate result. Based on the overall dimension of the build geometry, this value should be determined by balancing the computational cost and desired accuracy.

   Keep Z (default) for Print Direction.

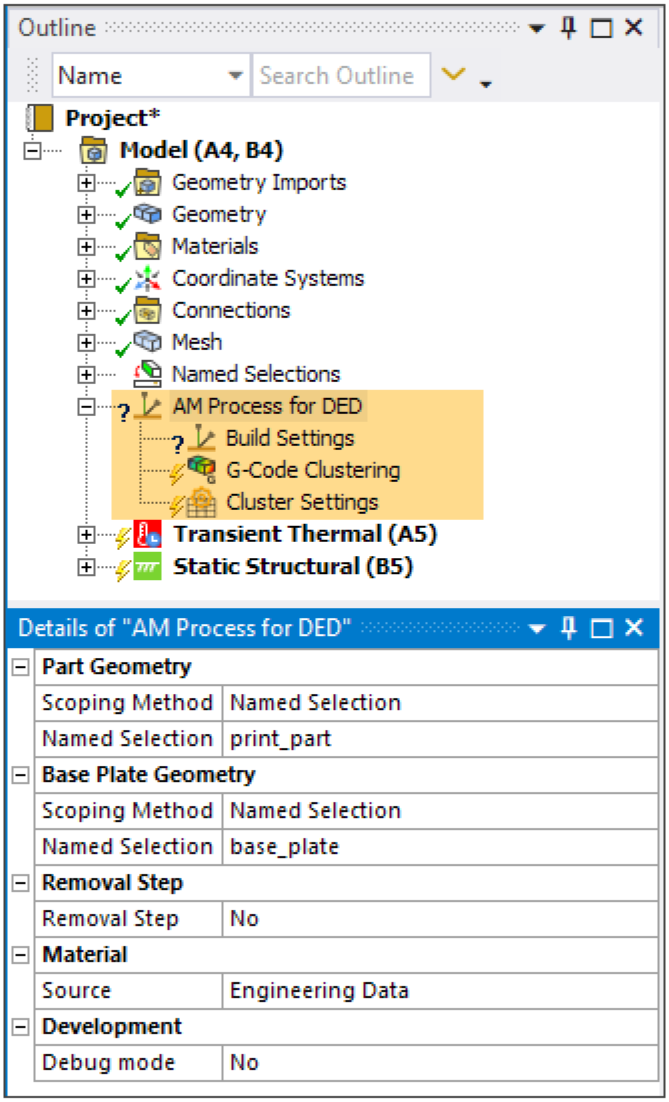
   Keep Yes (default) for Build to Base Contact Generation.

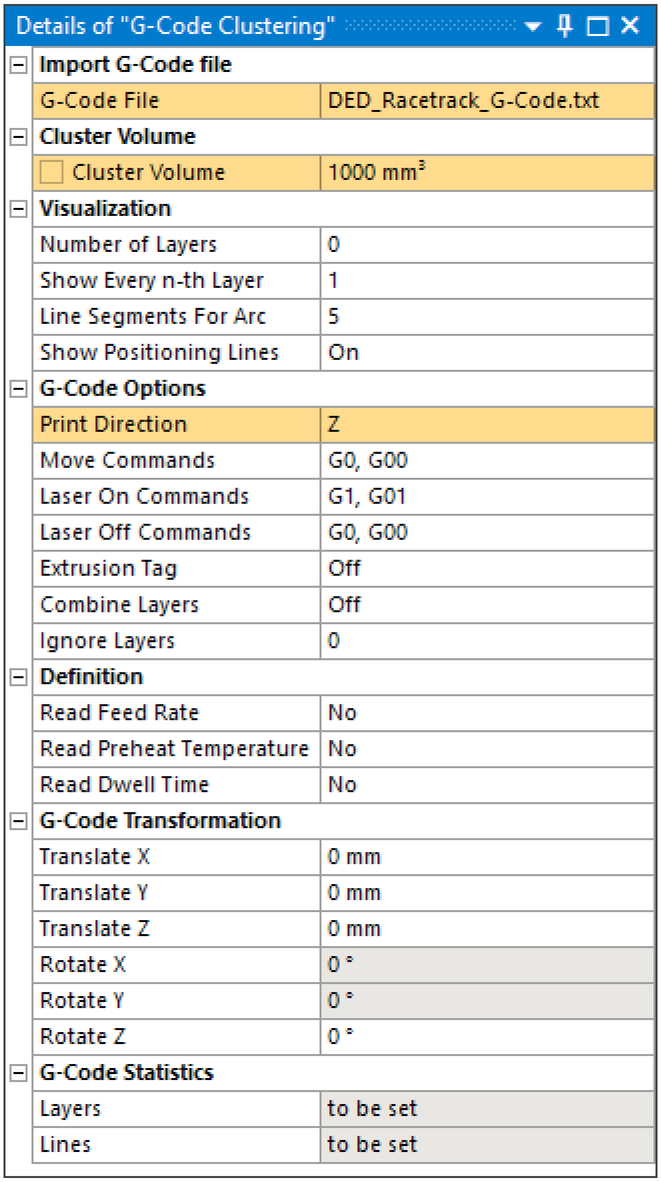
   Keep Auto (default) for Contact Generation Method. This assumes a planar baseplate, which we have for this model.

4. Click **Next** to move to the next step.



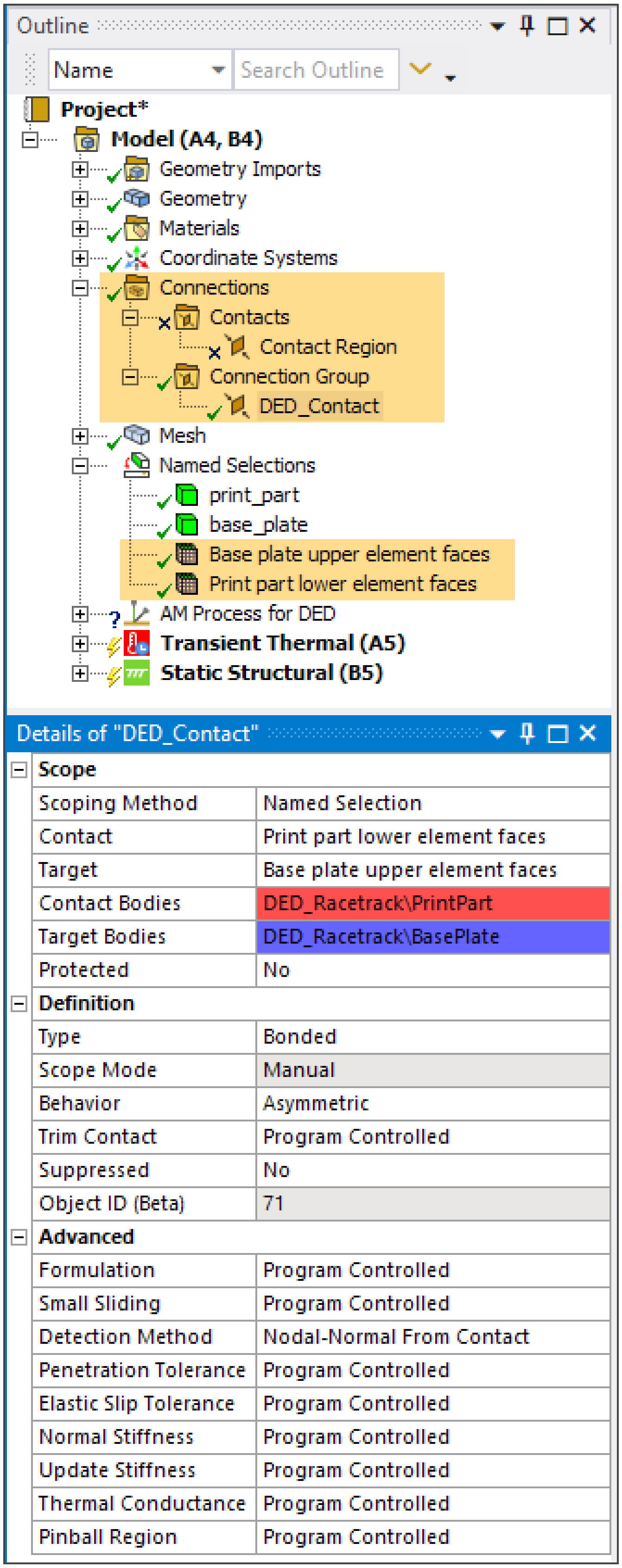
In this step, the AM Process for DED object and its child objects are added to the project tree. AM Process for DED establishes the options and assumptions appropriate for a DED simulation.


Outline
Name
Search Outline
Project*
Model (A4, B4)
Geometry Imports
Geometry
Materials
Coordinate Systems
Connections
Mesh
Named Selections
AM Process for DED
Build Settings
G-Code Clustering
Cluster Settings
Transient Thermal (A5)
Static Structural (B5)
Details of "AM Process for DED"
Part Geometry
Scoping Method
Named Selection
Named Selection
print_part
Base Plate Geometry
Scoping Method
Named Selection
Named Selection
base_plate
Removal Step
Removal Step
No
Material
Source
Engineering Data
Development
Debug mode
No


Details of "G-Code Clustering"

Import G-Code file	
G-Code File	DED_Racetrack_G-Code.txt
Cluster Volume	
Cluster Volume	1000 mm³
Visualization	
Number of Layers	0
Show Every n-th Layer	1
Line Segments For Arc	5
Show Positioning Lines	On
G-Code Options	
Print Direction	Z
Move Commands	G0, G00
Laser On Commands	G1, G01
Laser Off Commands	G0, G00
Extrusion Tag	Off
Combine Layers	Off
Ignore Layers	0
Definition	
Read Feed Rate	No
Read Preheat Temperature	No
Read Dwell Time	No
G-Code Transformation	
Translate X	0 mm
Translate Y	0 mm
Translate Z	0 mm
Rotate X	0 °
Rotate Y	0 °
Rotate Z	0 °
G-Code Statistics	
Layers	to be set
Lines	to be set

Also, two new Named Selections have been added defining the element faces required for contact generation. Finally, a DED_Contact object has been added to establish the contact connection between the part and the base plate. It uses the element faces Named Selections.



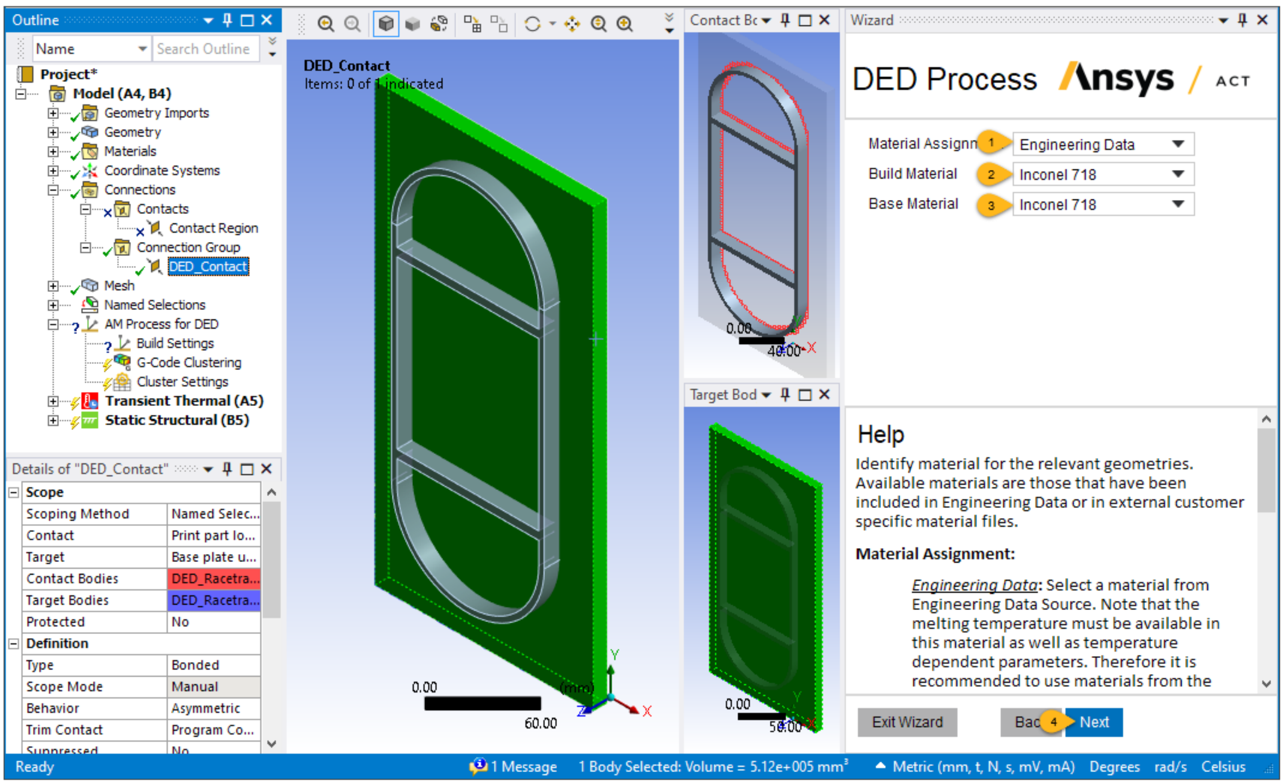
See the following topics in the *DED Simulation Guide* for additional information:

- Perform Element Clustering
- Define Connections

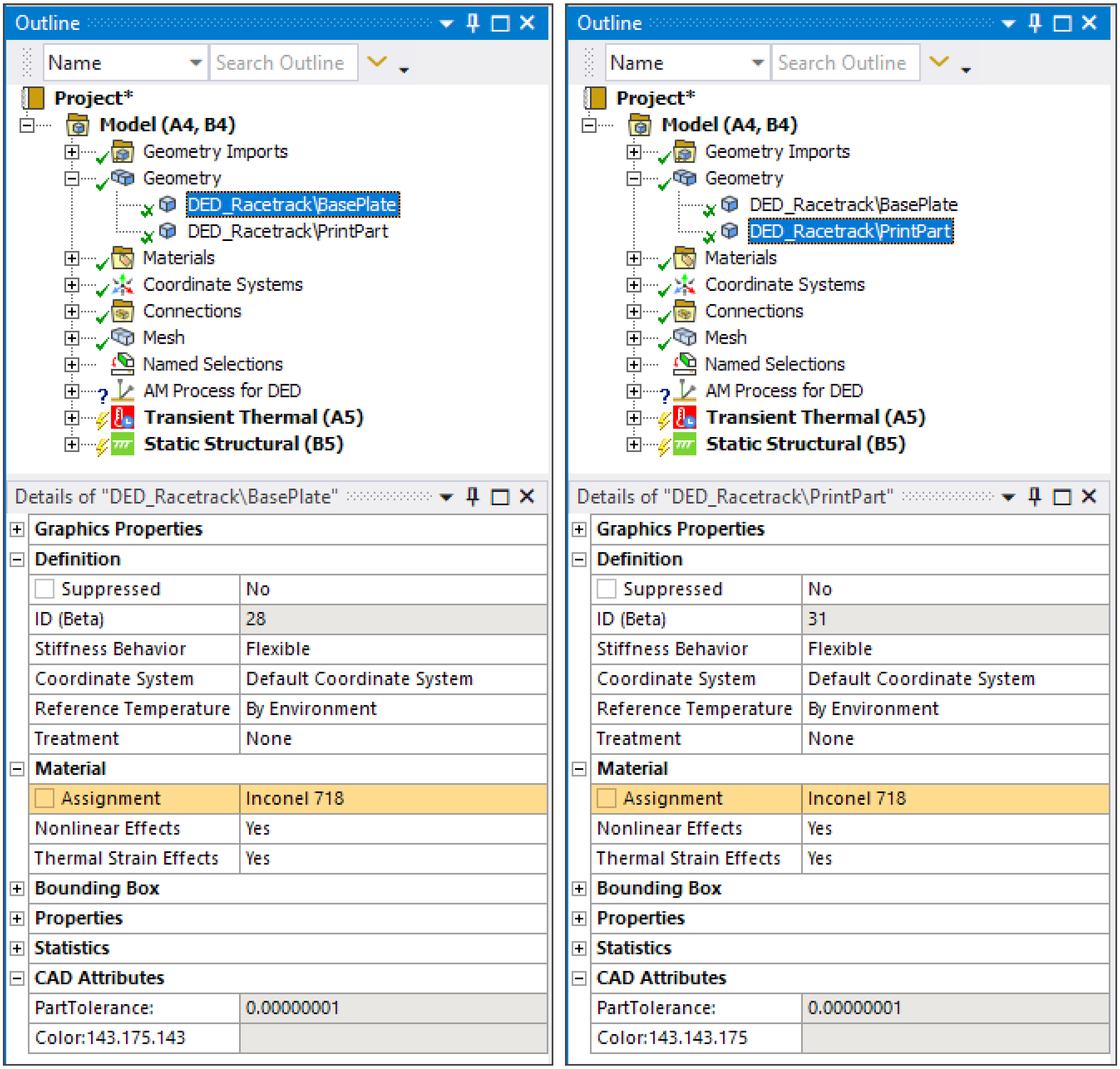
4.8. Wizard Step 4 - Assign Materials

Assign the AM material Inconel 718 to both bodies.

1. Choose **Engineering Data** for Material Assignment.
2. Choose **Inconel 718** for the Build Material.
3. Choose **Inconel 718** for the Base Material.
4. Click **Next** to move to the next step.



In this step, Inconel 718 material is assigned to both the part and the base plate. The Reference Temperature is set to By Environment for now but the Reference Temperature for the part will be updated in the Build Settings step next.



See the following topics in the *DED Simulation Guide* for additional information:

- Assign Materials

4.9. Wizard Step 5 - Define Build Settings and Thermal Boundary Conditions

Specify settings and conditions related to the DED machine and the process. We will use many of the defaults, such as values for gas convection coefficients, advanced options, and so on. Use the links to the DED Simulation Guide provided at the end of this section for more information about these options.

Machine Settings:

1. Enter **72** for Material Deposition Rate. The value should match the actual machine setting for the printing process.

Build Conditions:

2. Enter **80** for Preheat Temperature. This is determined by the actual preheating condition of the printing process.

3. We apply the preheat to the bottom surface of the base plate. Rotate the model in the graphics window, select the underside surface of the base, and click **Apply** in the Preheat Geometry field. We will keep the preheat off during printing.

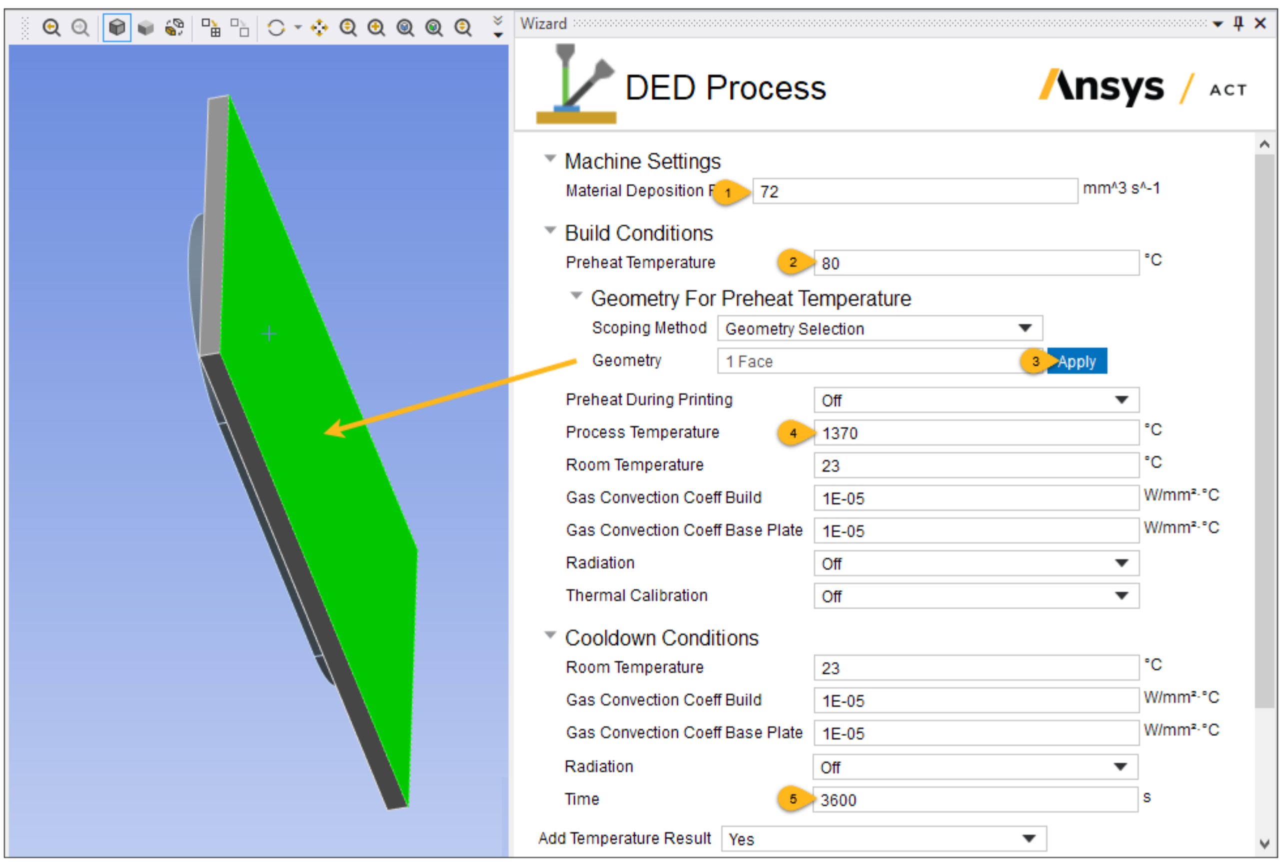
4. Enter **1370** for Process Temperature. This is the temperature value assigned to newly activated elements at each load step, normally set as the melting temperature.

Cooldown Conditions:

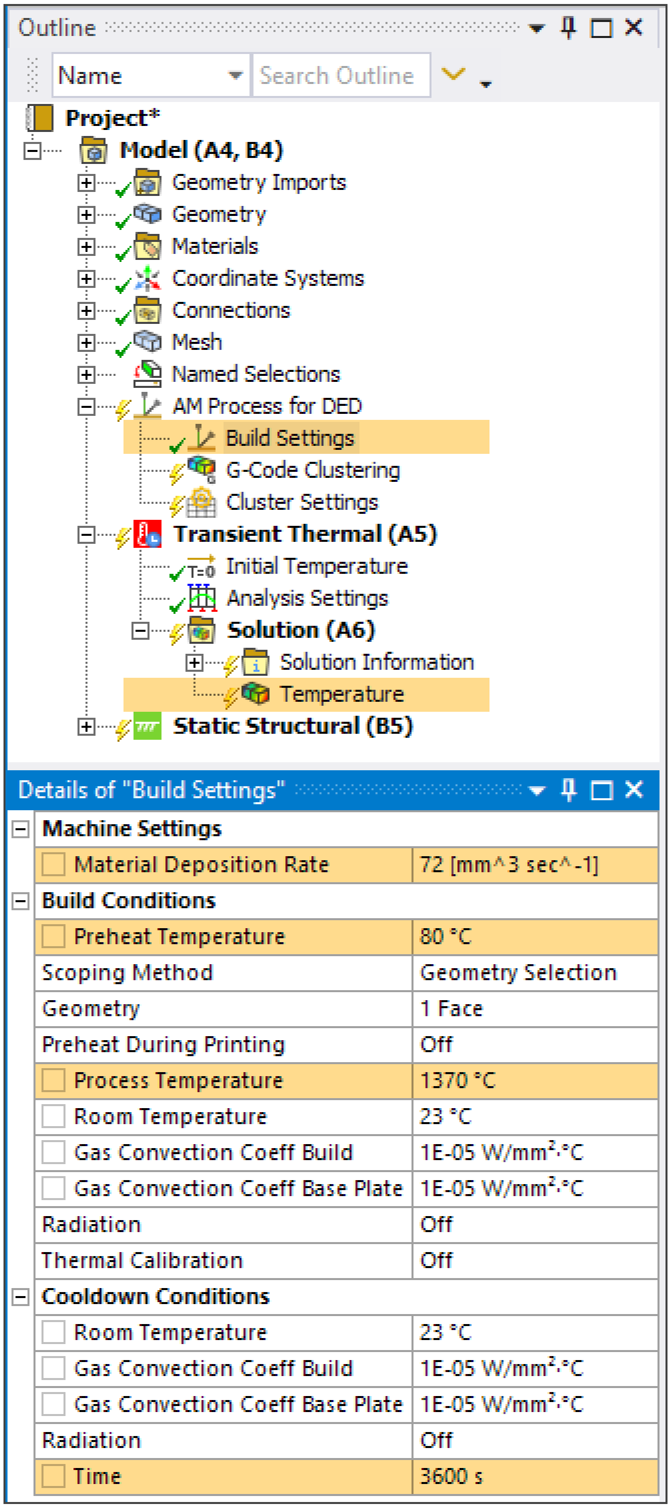
5. Enter **3600** for Time. This allows an hour for cooldown.

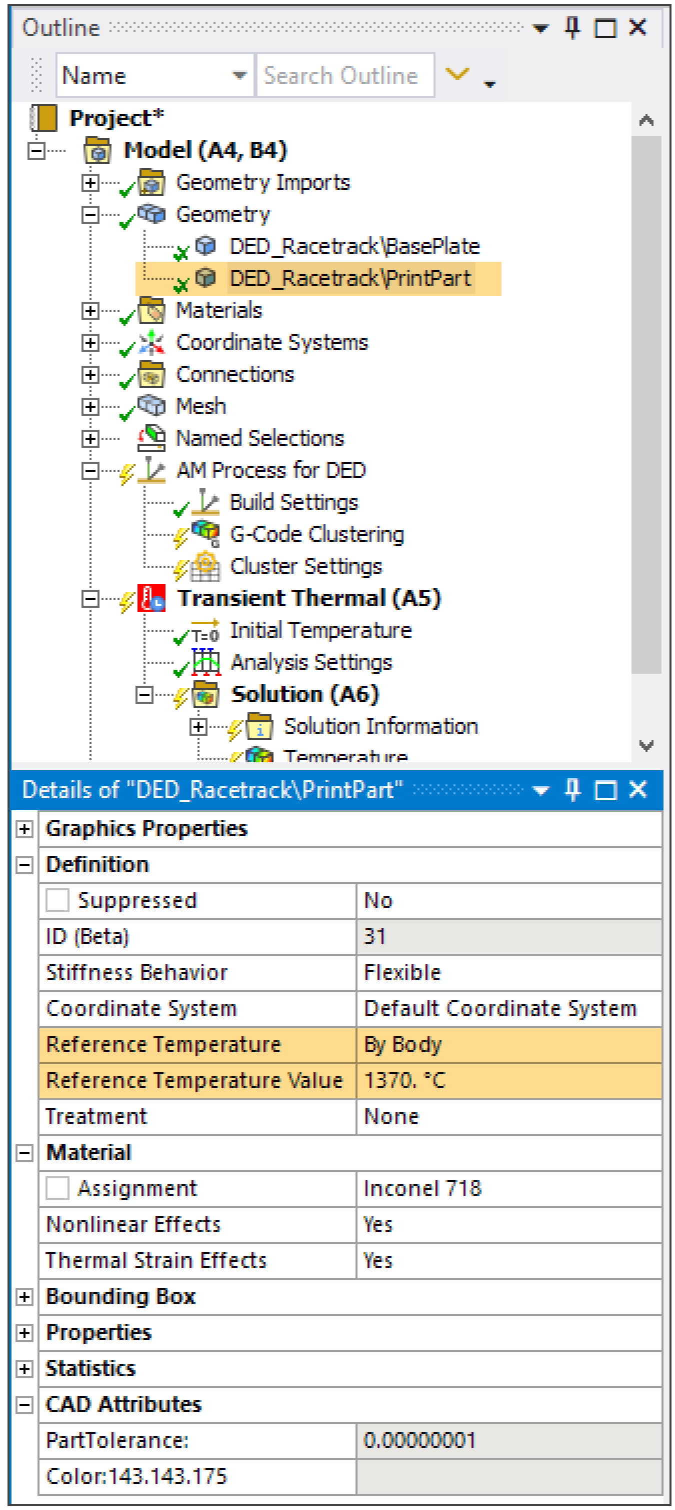
   Keep Yes (default) for Add Temperature Result. This is simply a convenience option related to the simulation and not related to anything in the printing process. A value of Yes adds a result item object into the project tree before solution so that the calculated temperatures will be populated upon solution completion and viewable with one click.

6. Click **Next** to move to the next step.



In this step, the Build Settings object is populated with chosen settings. The green checkmark next to the Build Settings object indicates it is now complete. Also, a Temperature result item is added under the Transient Thermal Solution object. Finally, the part body is assigned a Reference Temperature By Body and the Reference Temperature Value is set to the Process Temperature. This is an important update to the properties for the part.





See the following topics in the *DED Simulation Guide* for additional information:

- Define Build Settings

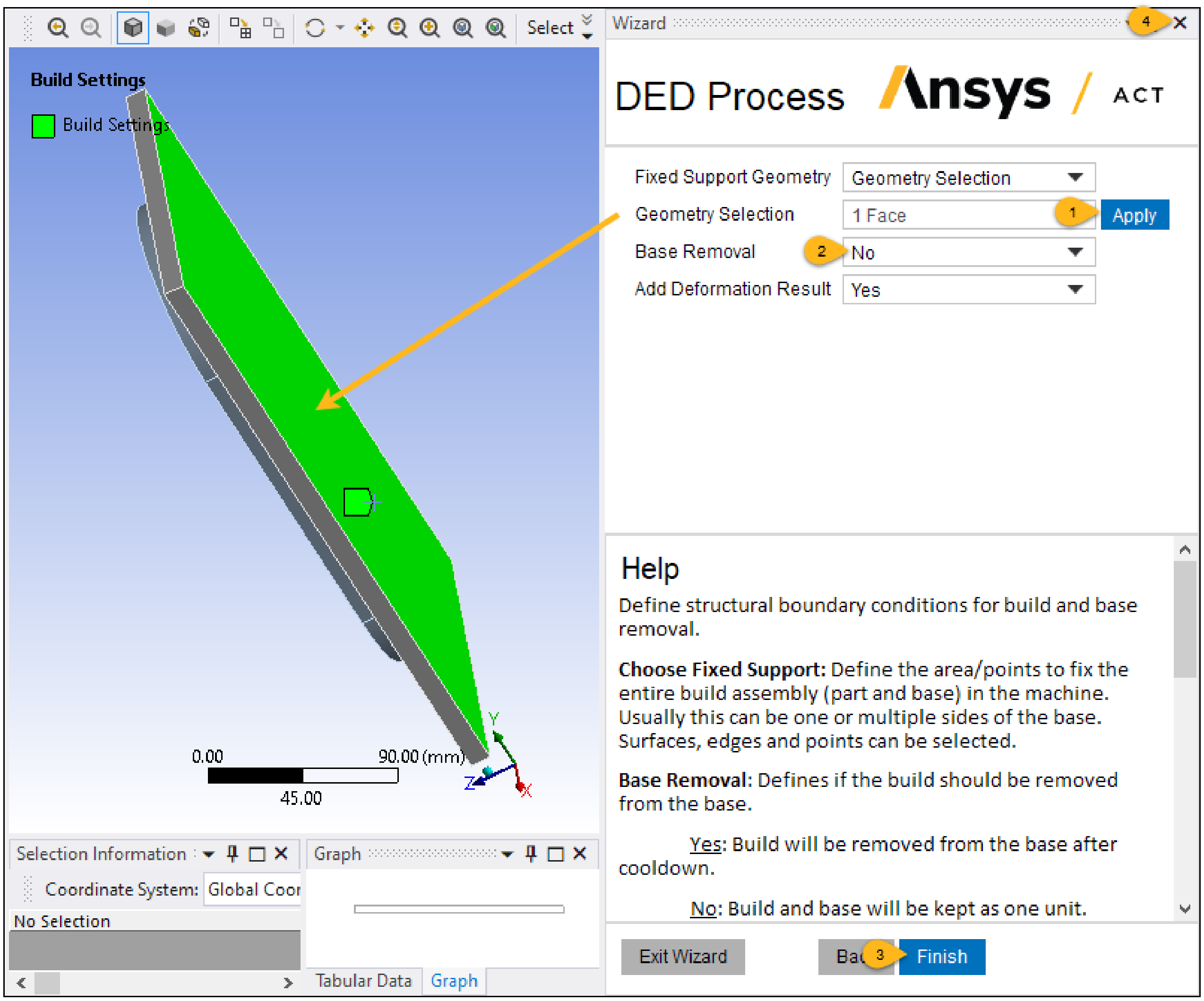
- Apply Thermal Boundary Conditions

4.10. Wizard Step 6 - Define Structural Boundary Conditions and Base Removal

Here we will apply a fixed support to the underside of the base plate. We will not simulate removal of the base plate for this example.

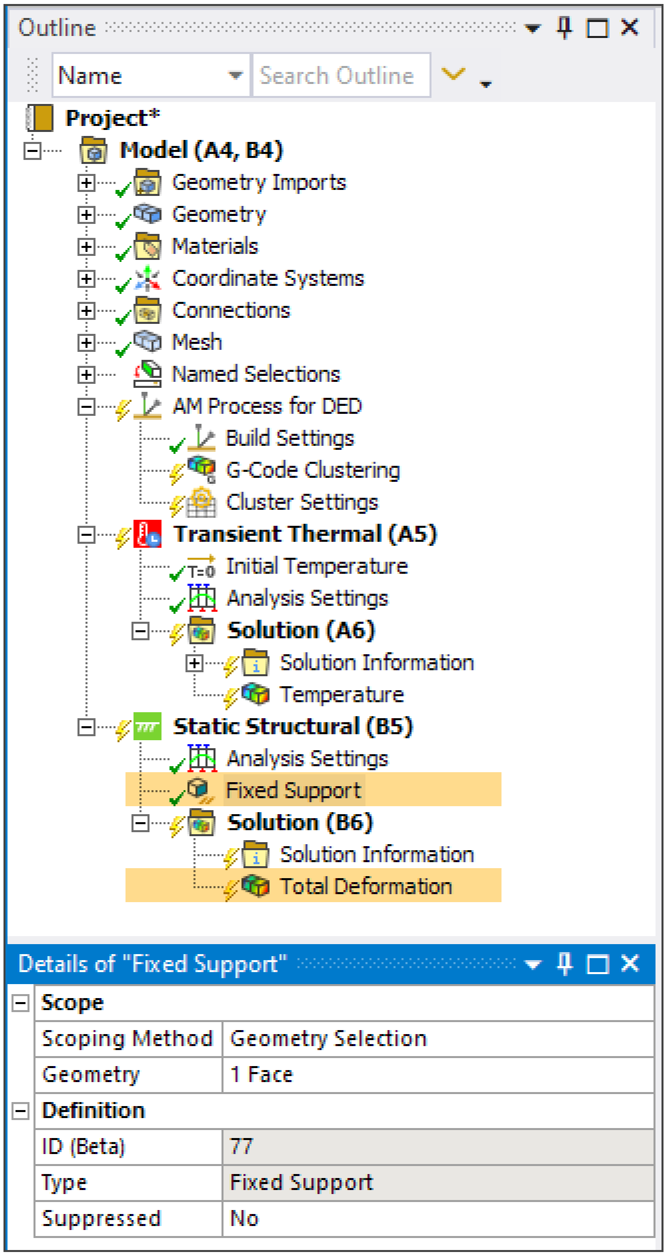
1. Select the underside surface of the base, and click **Apply** in the Geometry Selection field.
2. Choose **No** for Base Removal.

   Keep Yes (default) for Add Deformation Result. This is simply a convenience option related to the simulation and not related to anything in the printing process. A value of Yes adds a result item object into the project tree before solution so that the calculated displacements will be populated upon solution completion and viewable with one click.
3. Click **Finish** to complete the wizard.
4. Click **X** in the upper, right corner to close the wizard panel.



Upon completion of the wizard, a Fixed Support object has been added to the project tree, along with a Total Deformation result item under the Static Structural Solution object.

When completing any wizard, you should review the status icons next to each object in the project tree. Objects with green checkmarks are complete. Objects with yellow lightning bolts indicate an action is required, such as mesh, generate, or solve. For this tutorial, the yellow lightning bolt next to G-Code Clustering indicates we now need to generate element clusters.



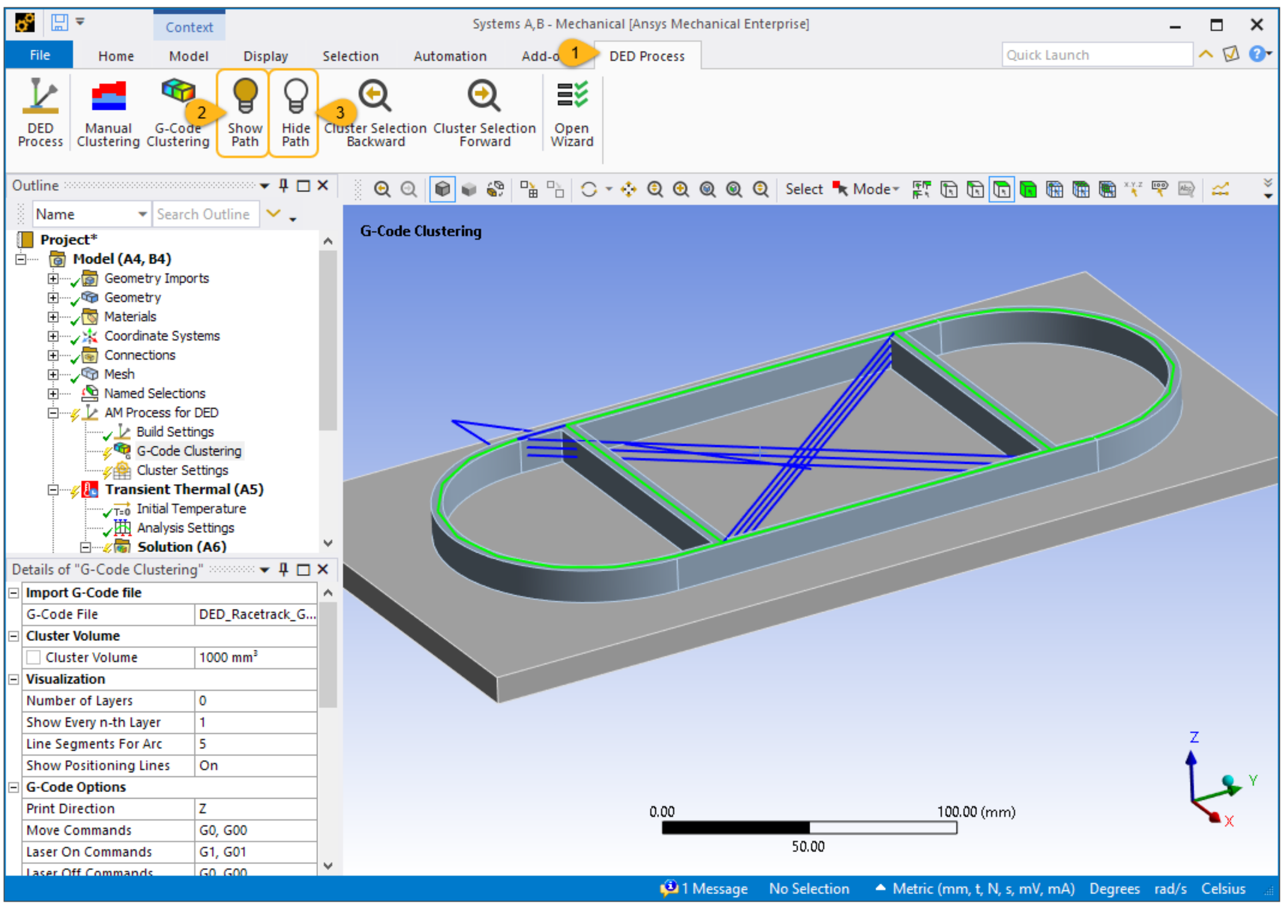
See the following topics in the *DED Simulation Guide* for additional information:

- Apply Structural Boundary Conditions

4.11. Perform Element Clustering

In the wizard, we identified the G-Code machine file. We should view and verify the tool path defined in the G-Code file before generating the element clusters.

1. Click the **DED Process** toolbar.
2. Click **Show Path**. Blue lines represent the tool path with no material deposition and green lines represent the tool path with material deposition. Visualization helps to determine the following:
   - Is it the correct G-Code file?
   - Is the location / rotation / orientation of the path correct with respect to the model?
   - Are the unit systems of the G-Code file and model matching?
   - Do I need to apply a Layer Offset to place the path on top of the layers?
   - Did I set the correct Laser On / Off commands?
3. Click the **Hide Path** button to turn off the tool path display.



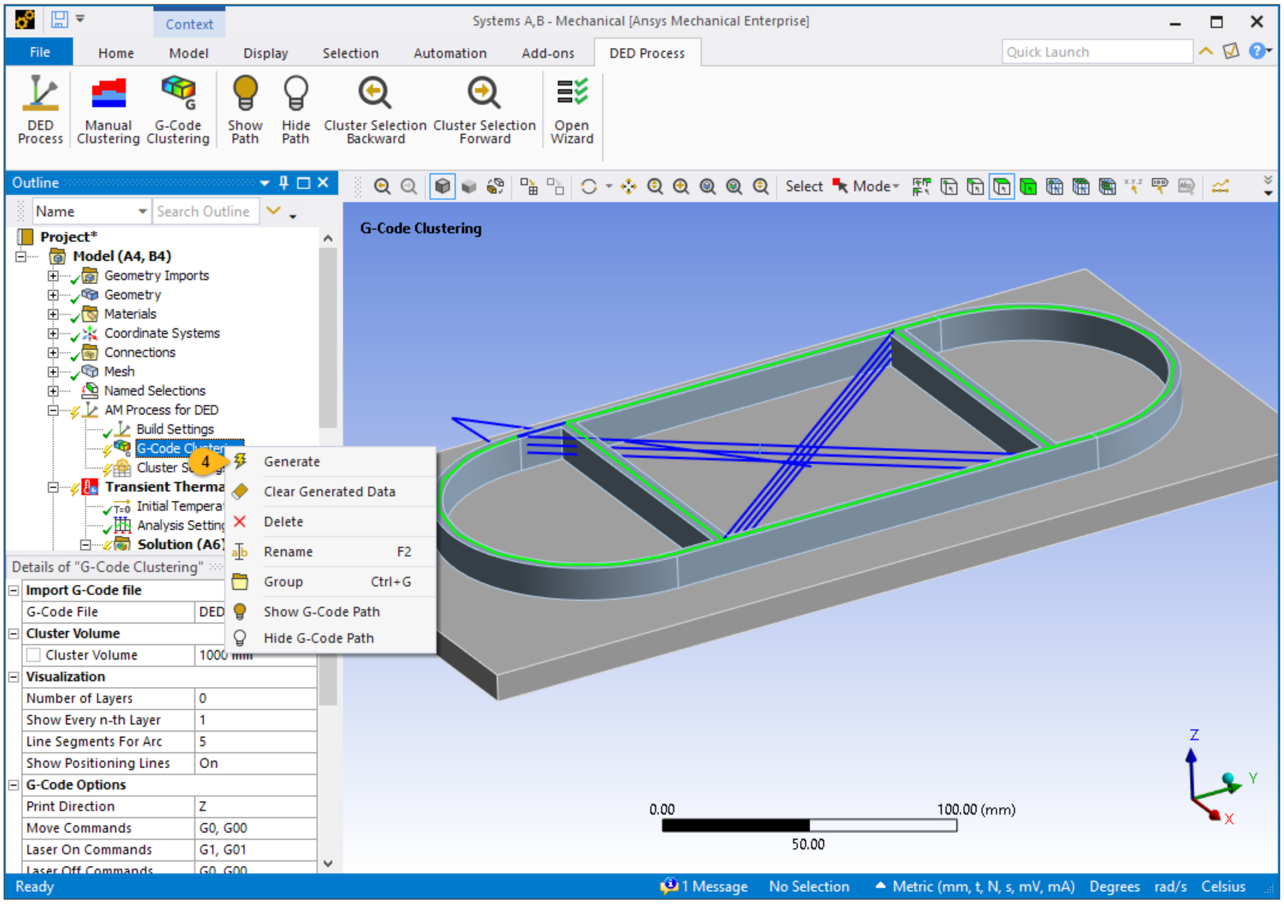
Once we have confirmed the path looks correct, we simply need to generate the element clusters.

Important:

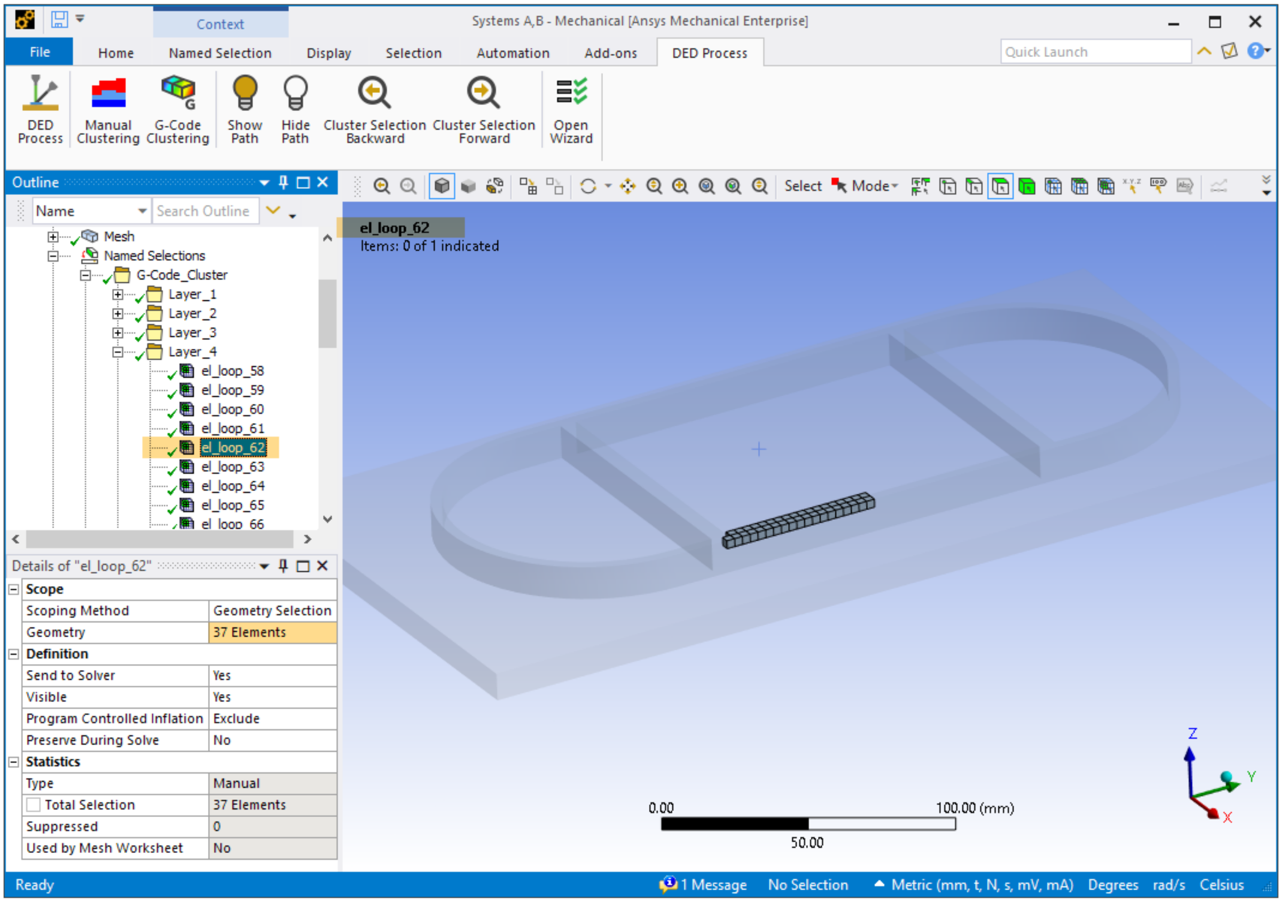
The cluster generation step requires an Ansys Additive Suite software license with Ansys Mechanical Enterprise or one of the multiphysics bundles. If you get a warning message at this step, check your software license.

---

4. Under the AM Process for DED object, right-click **G-Code Clustering** and select **Generate**.



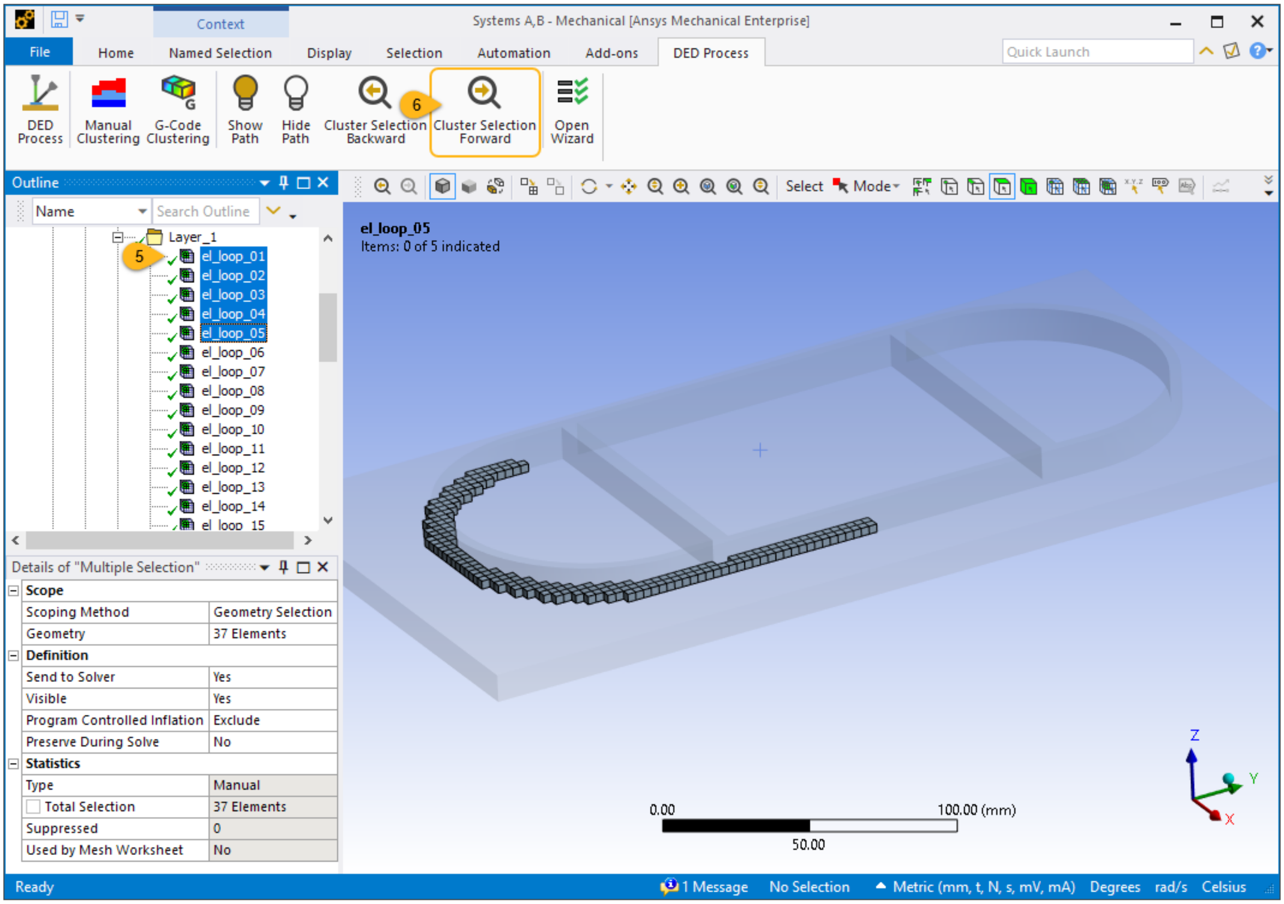
The clustering process may take a few to several minutes, depending on the size of your model. When completed, you will see Named Selections for all the clusters organized into folders, one folder per layer. Given the part geometry, the Build Element Size of 3 mm, and the Cluster Volume of 1000 mm^3, there are 76 total clusters for this model. The following image shows one example cluster consisting of 37 elements on layer 4.



You can view the progression of element clusters along the path.

5. Under Named Selections, expand the Layer 1 folder and click the first cluster, **el_loop_01** to see the first element cluster.

6. Click the **Cluster Selection Forward** button in the toolbar to show the next cluster. Continue clicking the Cluster Selection Forward button to see consecutive clusters displayed along the tool path.



Once the element clusters are generated, a green checkmark appears next to the AM Process for DED object. Our simulation is ready to be solved.

See the following topics in the *DED Simulation Guide* for additional information:

- Perform Element Clustering

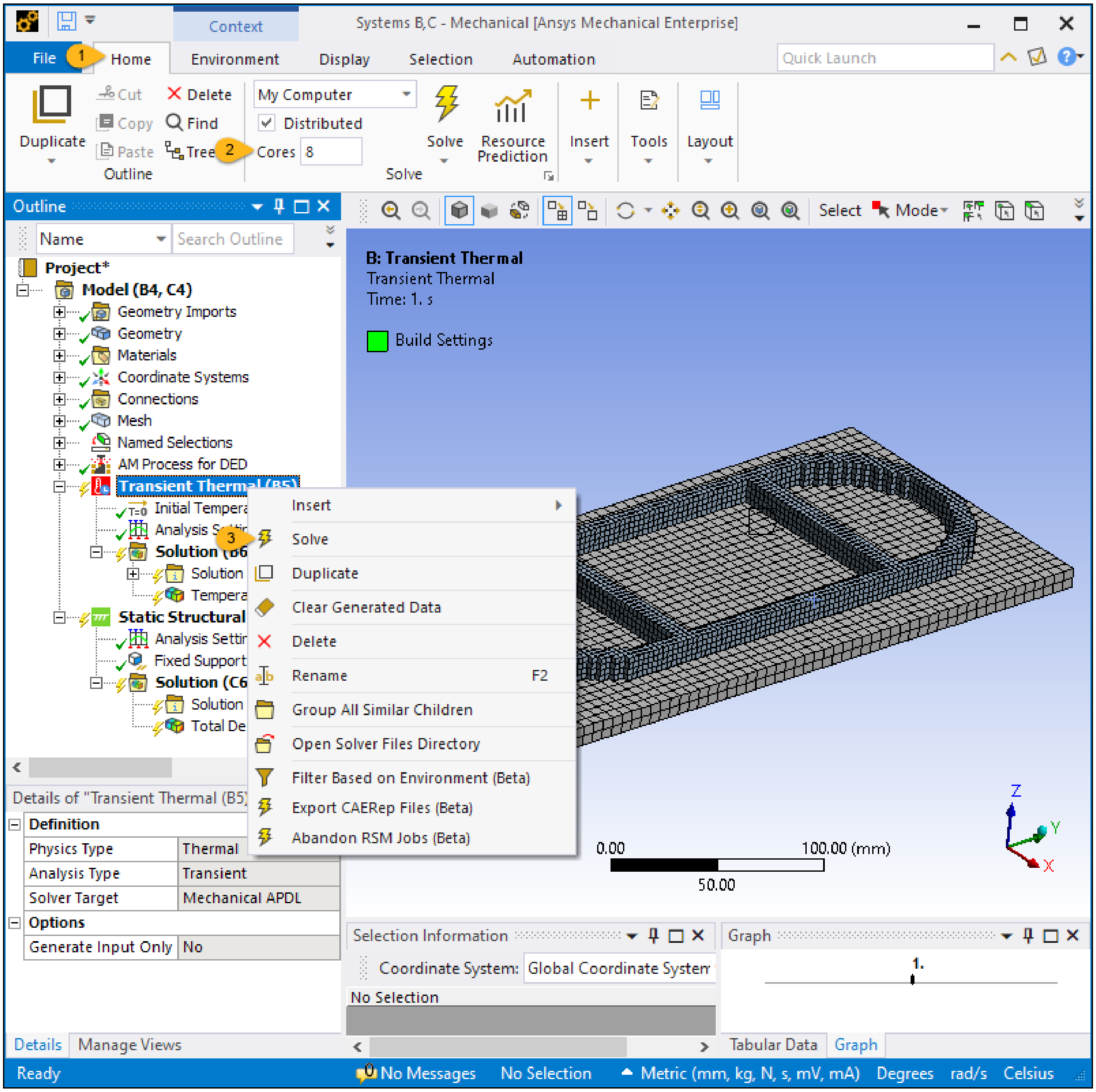
4.12. Solve the Transient Thermal Analysis

We will solve the Transient Thermal system first and view results before proceeding to the Static Structural solution. First we should set the number of cores we want to use.

1. Click the **Home** tab in the ribbon.

2. In the Solve group, set the number of **Cores** to solve with for this simulation, considering your machine capabilities and licensing for HPC. This example uses 8 cores.

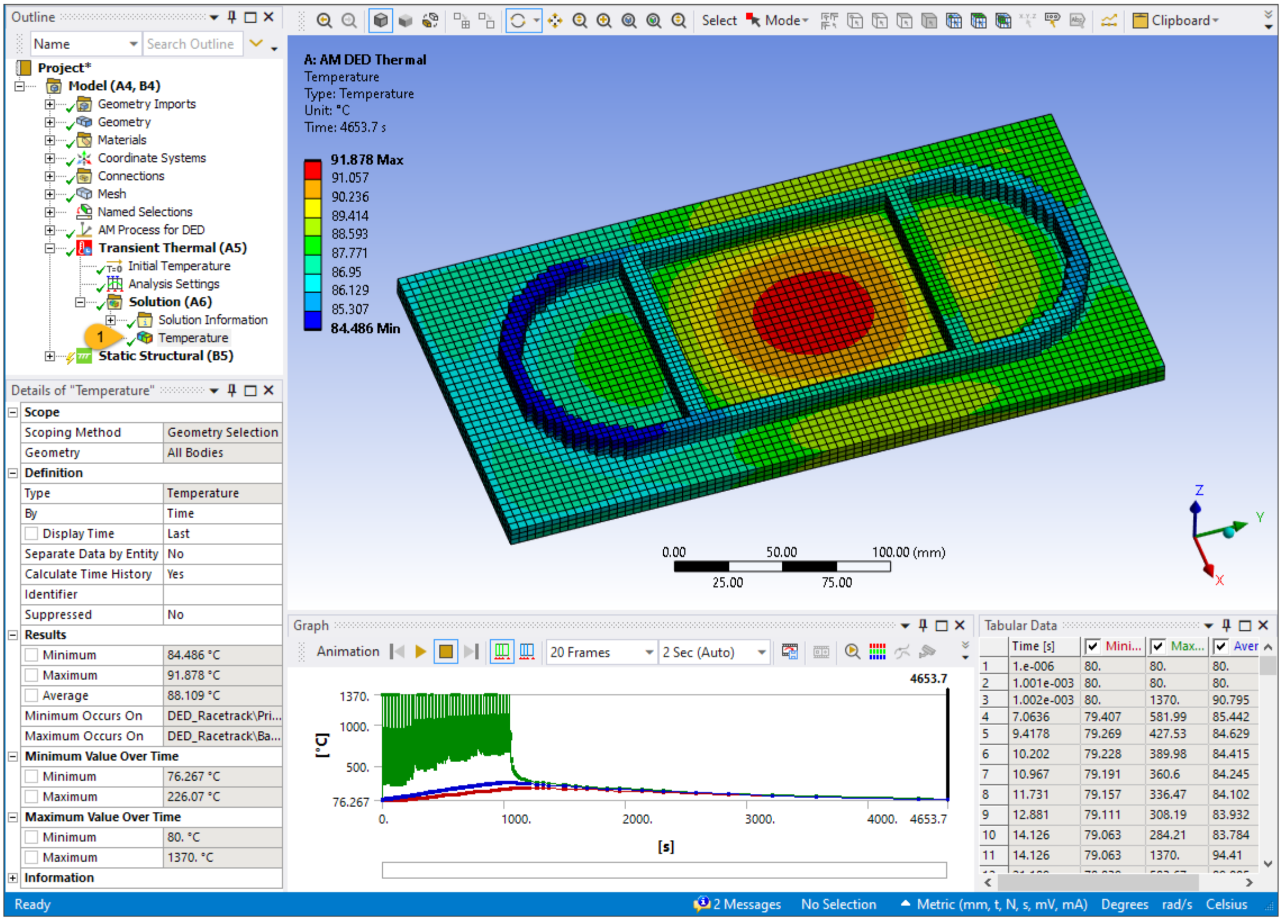
3. Right-click the Transient Thermal object and select **Solve**.



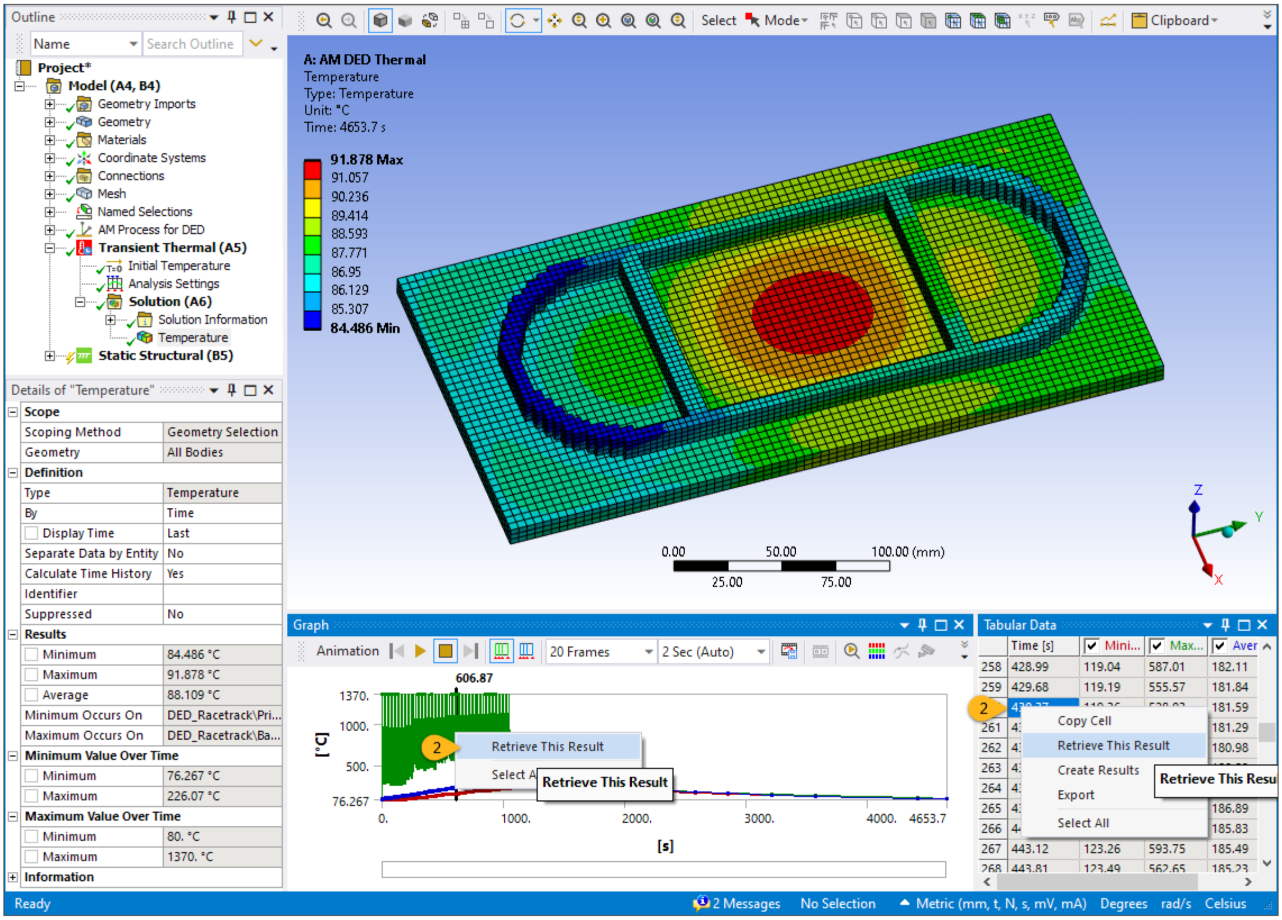
It is safe to ignore the warning message: *"The model has no temperature, convection, or radiation conditions specified..."*, as the DED Process Add-on applies the appropriate boundary conditions automatically with information from the wizard.

4.13. Review Thermal Results

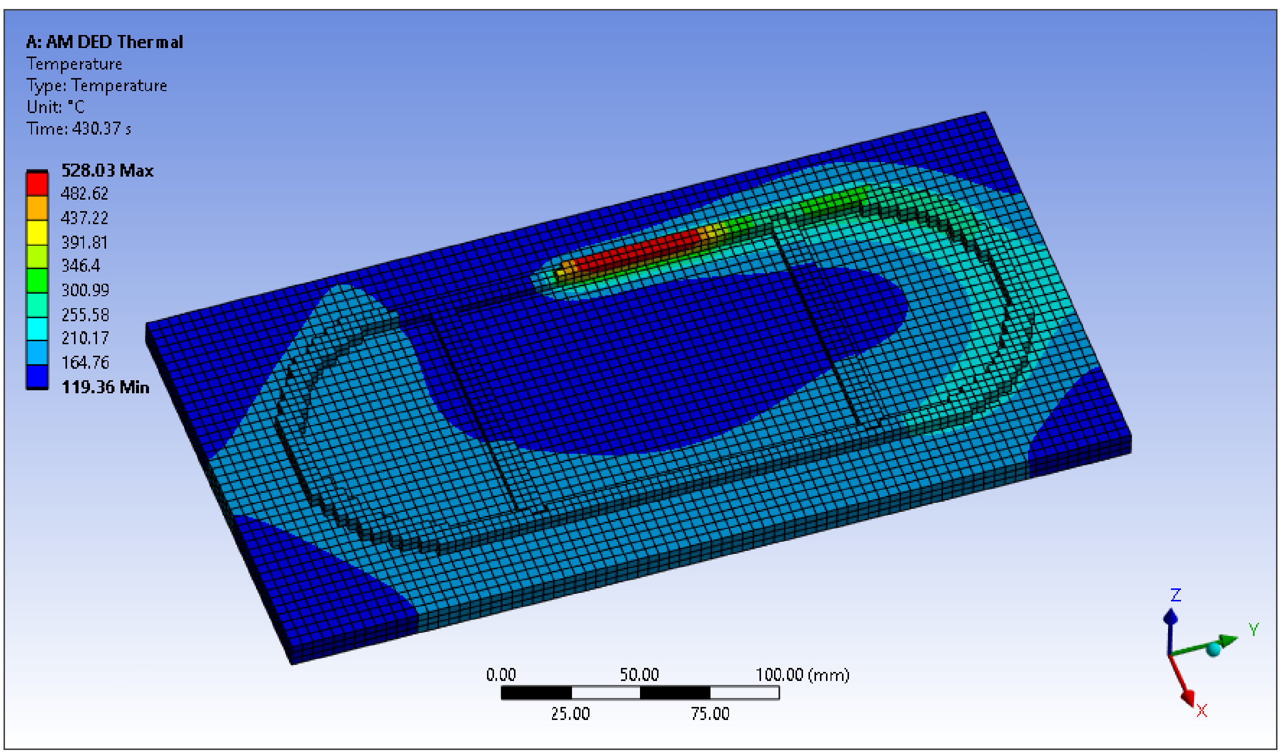
1. When the thermal solution is finished, click the **Temperature** result object under Transient Thermal > Solution to see a plot of overall temperatures at the end of cooldown.

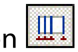
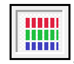


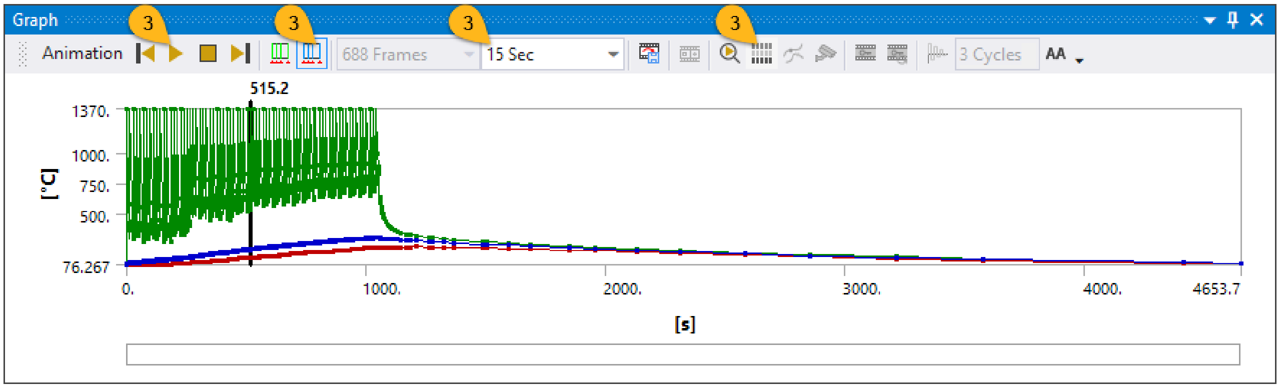
2. Right-click anywhere within the temperature-time graph located at the bottom, left corner of the screen, or at any time in the tabular data at the bottom right, and choose **Retrieve This Result** to see temperatures at that time point.



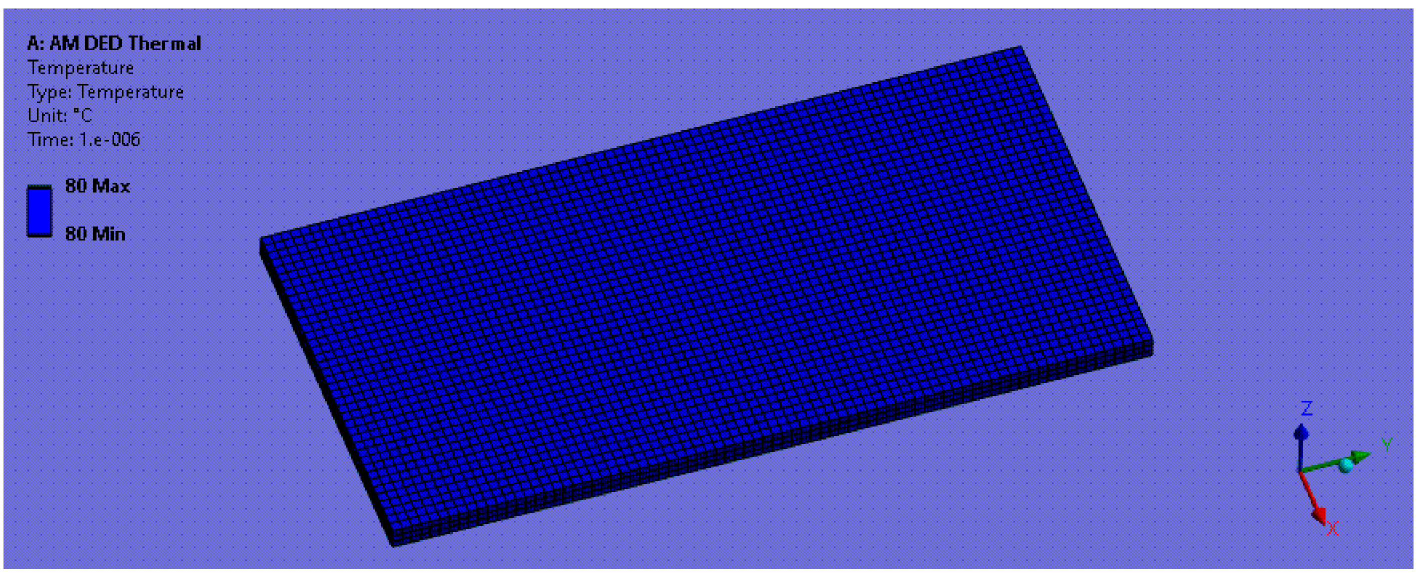
Here is the result at time=430.37 seconds.



3. Use the animation controls at the top of the Graph window. Click the **Result Sets** button  and also the **Update Contour Range at Each Animation Frame** button . Adjust the number of seconds for the animation and click **Play**. See Animation in the *Mechanical User's Guide* for more information about animation controls.

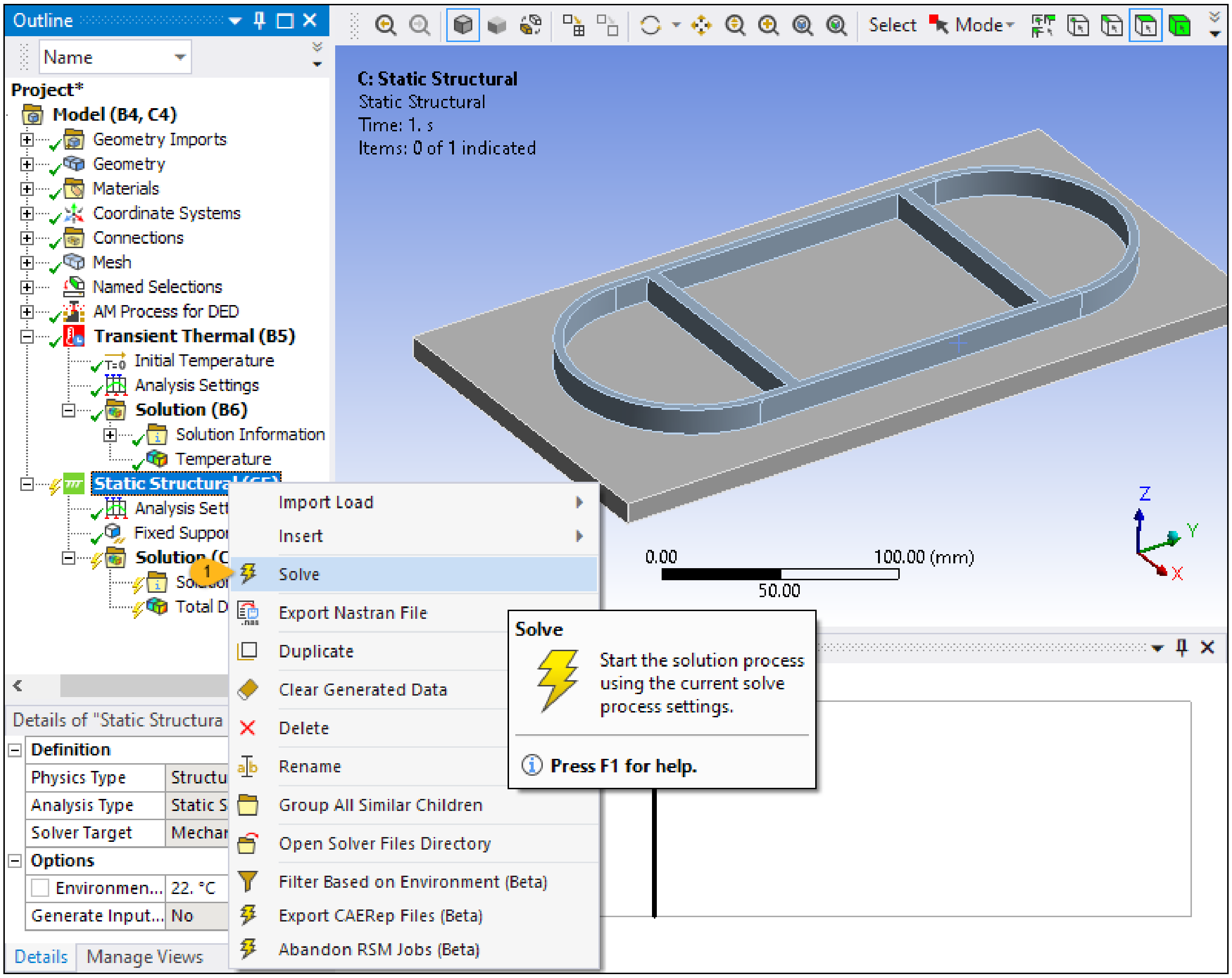
   

   The following is an animated gif. Refresh the page to refresh the animation. The animation is not viewable in PDF format.

4.14. Solve the Static Structural Analysis

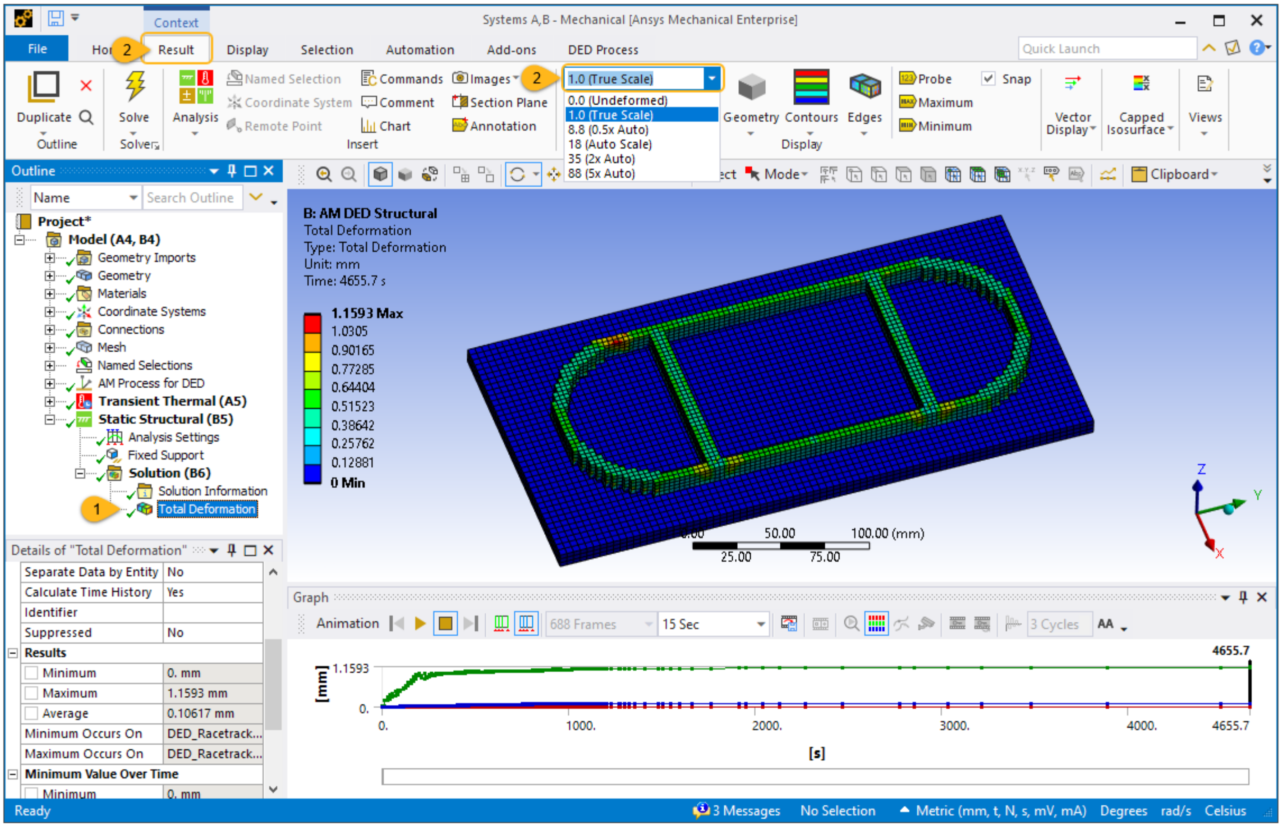
1. Right-click the Static Structural object and select **Solve**.



4.15. Review Structural Results

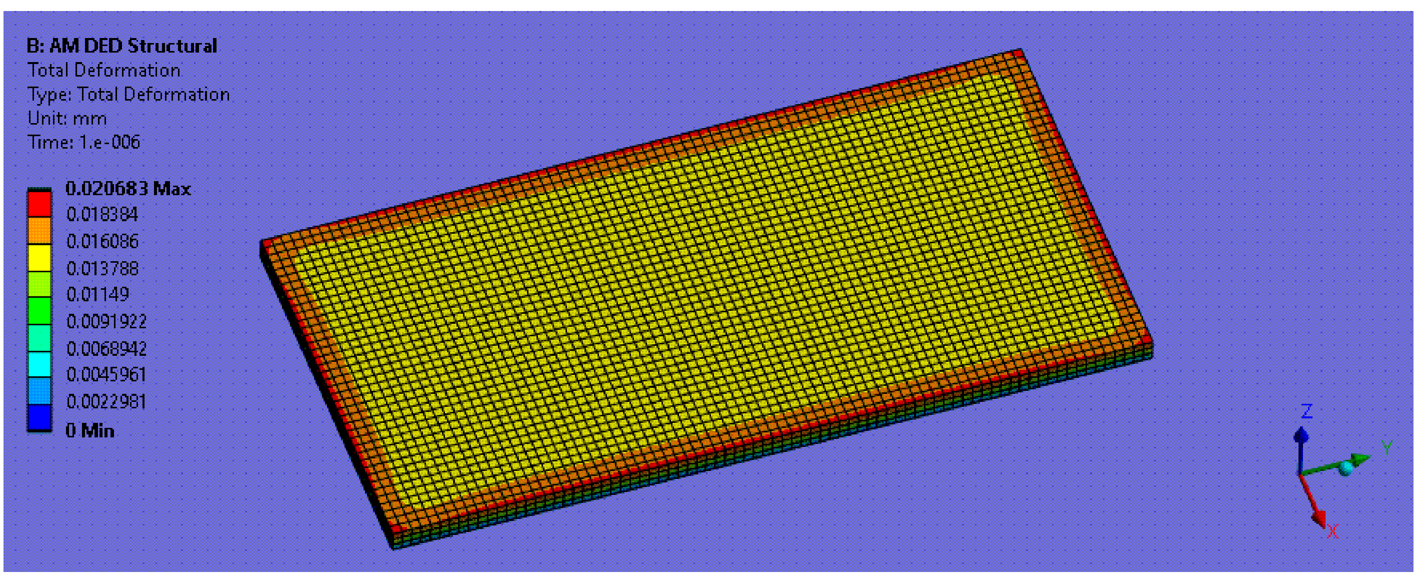
1. When the structural solution is finished, click the **Total Deformation** result object under Static Structural > Solution to see a plot of overall displacements at the end of cooldown.

2. Change the display scale to **1.0 (True Scale)** in the ribbon (**Result** tab) for a better view of deformation. (By default, the scale is set for an exaggerated display.)



3. To view different time steps, right-click different steps in tabular data or on the results graph and select **Retrieve This Result**.

   Just as we viewed an animation of thermal results, we can view an animation of the structural results. *The following is an animated gif. Refresh the page to refresh the animation. The animation is not viewable in PDF format.*



Congratulations! You have completed the tutorial.

Chapter 5: Sintering Simulation - Printed Bridge

This tutorial demonstrates how to perform a sintering process simulation using the Sintering Process Add-on in Mechanical.

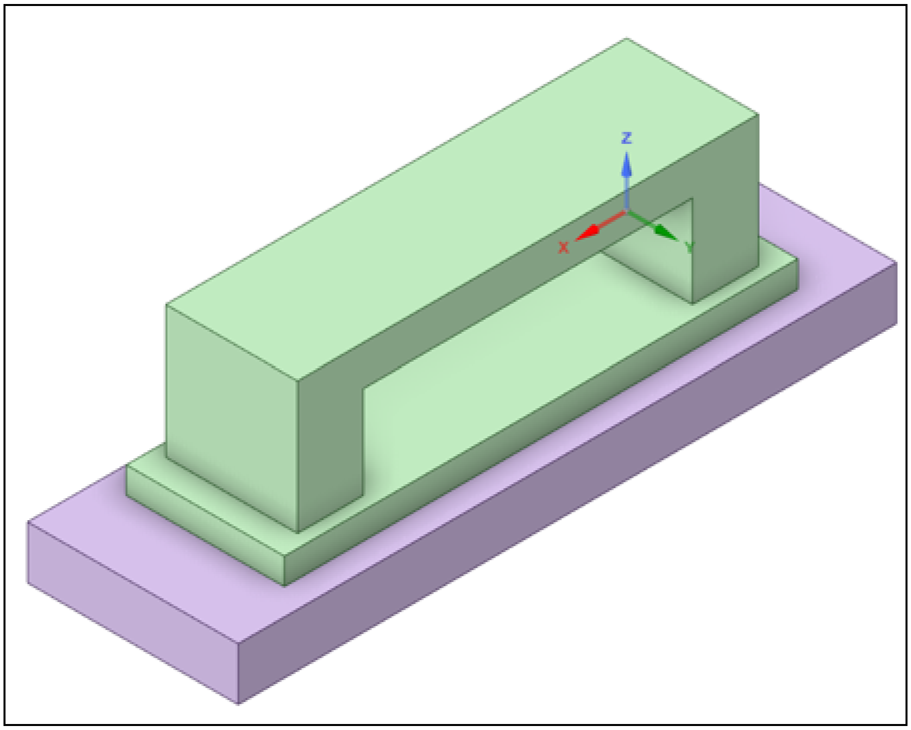
The following features and capabilities are used:

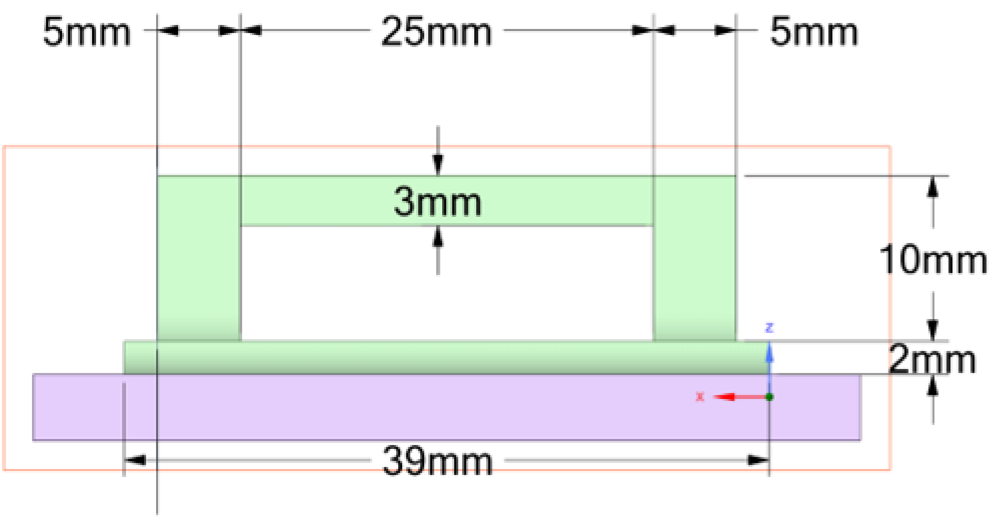
- Sintering Process Wizard (included with the add-on)
- Ansys-predefined sintering material 316L (PIM)
- Sintering result items: Relative Density and Sinter Stress

Tutorial steps:

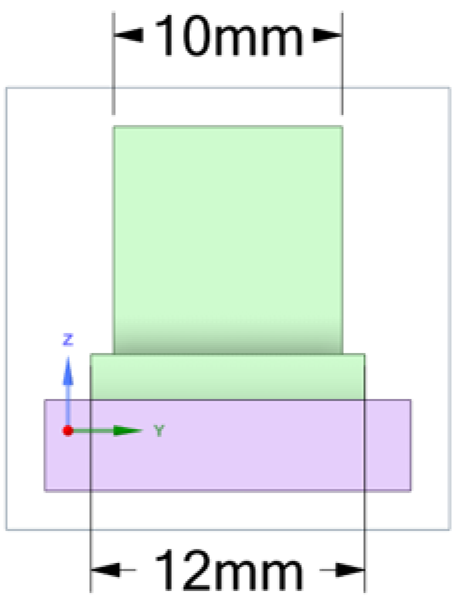
5.1. Problem Description

We will simulate the sintering process of this span bridge model, shown here on a planar baseplate. The model is composed of 316L stainless steel in a green state after it has been printed. Our simulation goals are to examine the densification during sintering and the final shape after shrinkage.


Z
X
Y

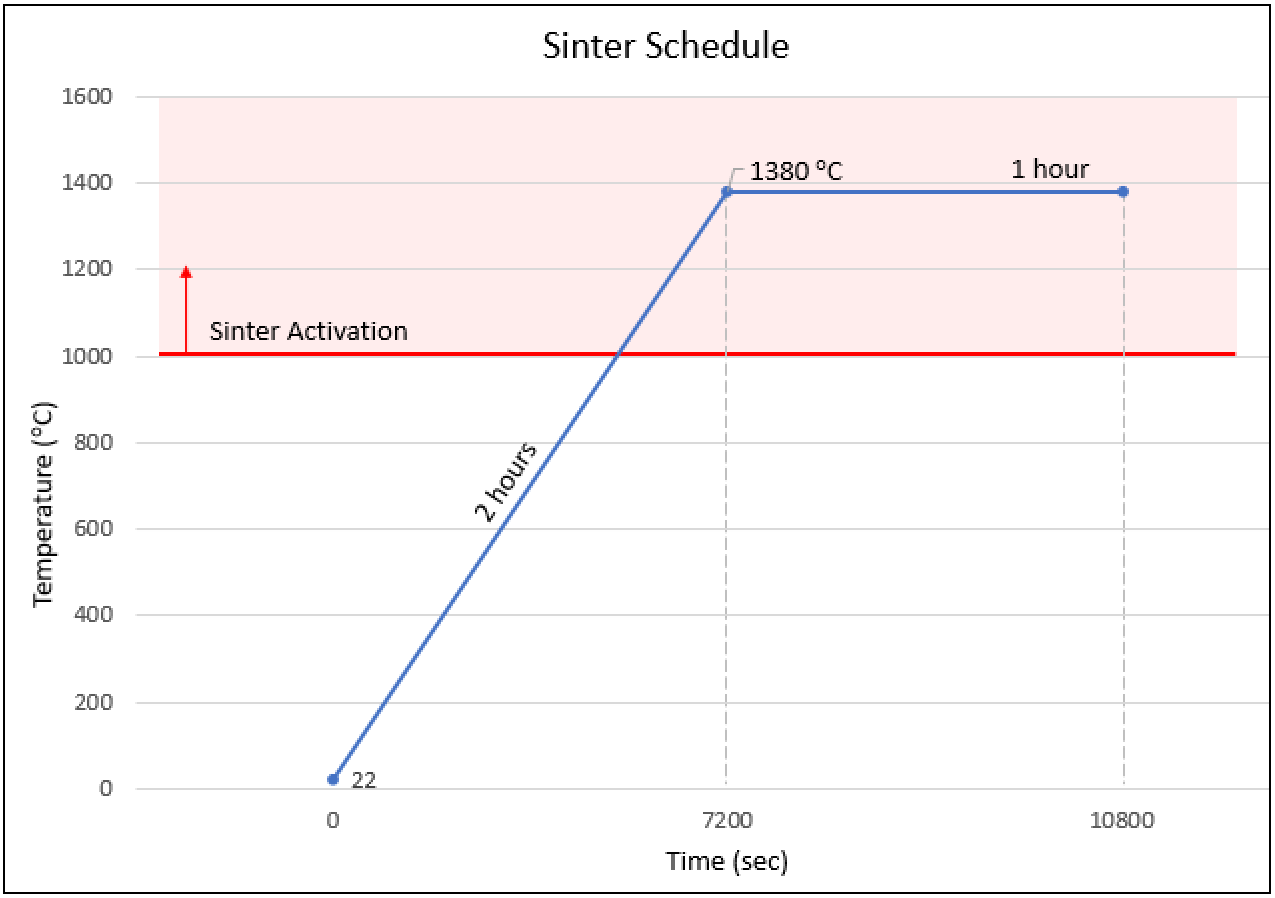


(a) Front view of the span bridge part



(b) Side view of span bridge part

The sintering process is performed in a single heating cycle in a batch furnace. For simplicity, the process has one short isothermal hold, as shown here:



Tutorial Files

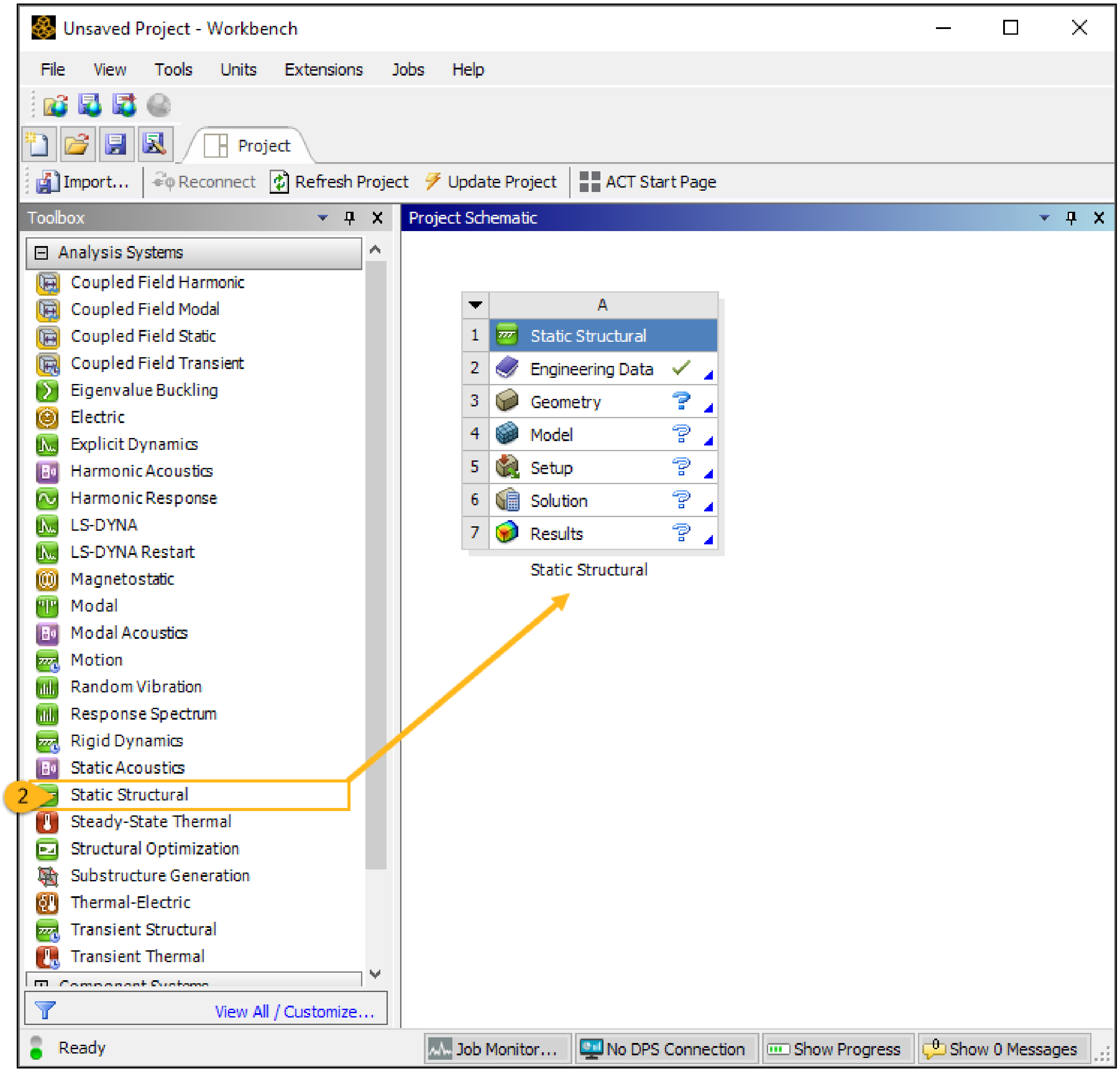
Click here to download the following:

`bridge_with_base.scdoc` — Geometry file of the span bridge part and a base plate, saved as an Ansys SpaceClaim document.

5.2. Create Analysis System

1. Open Ansys Workbench.

2. In Workbench, from the Analysis Systems listed on the left, drag a Static Structural analysis into the Project Schematic.

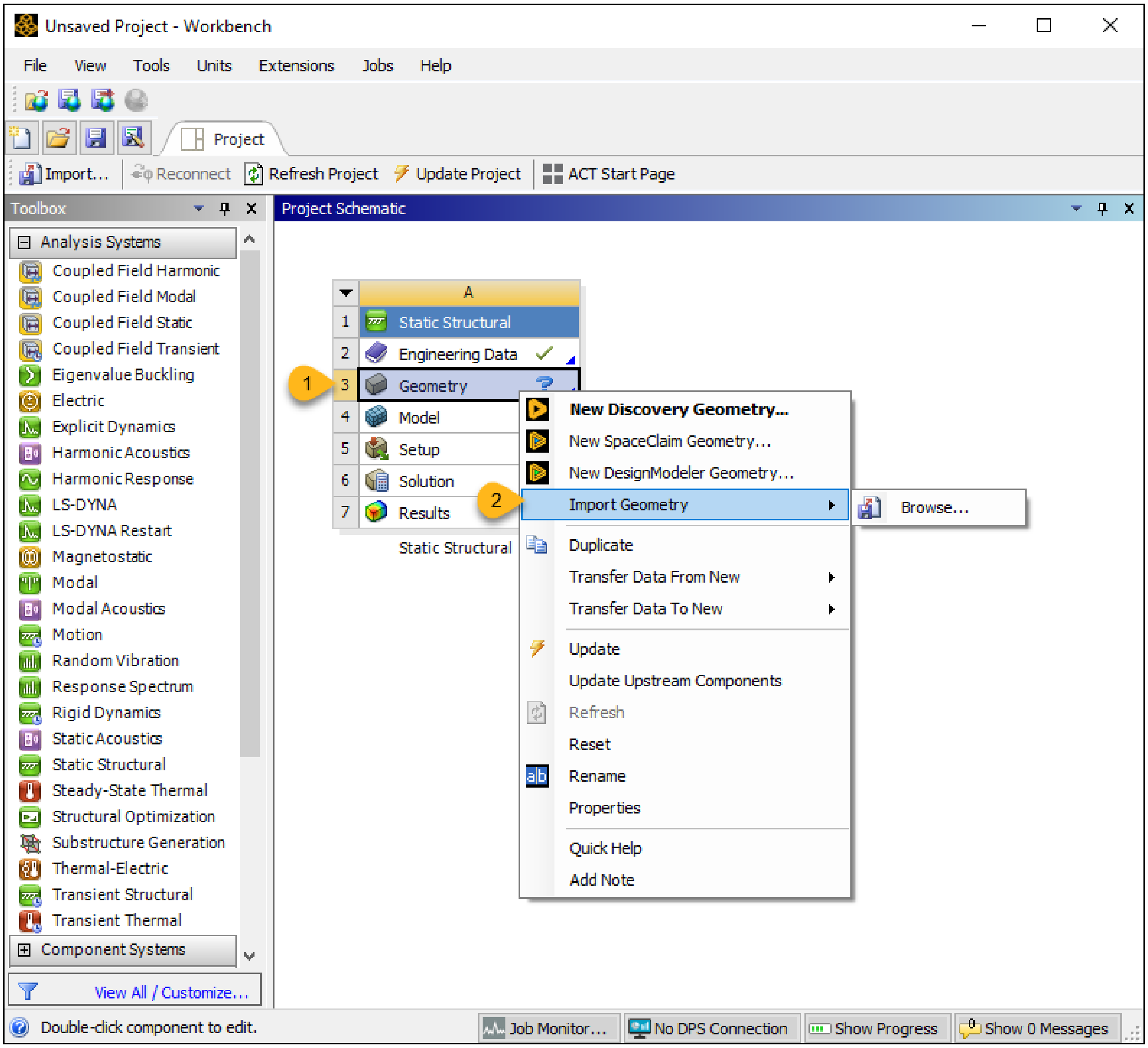


5.3. Attach Geometry and Set Units

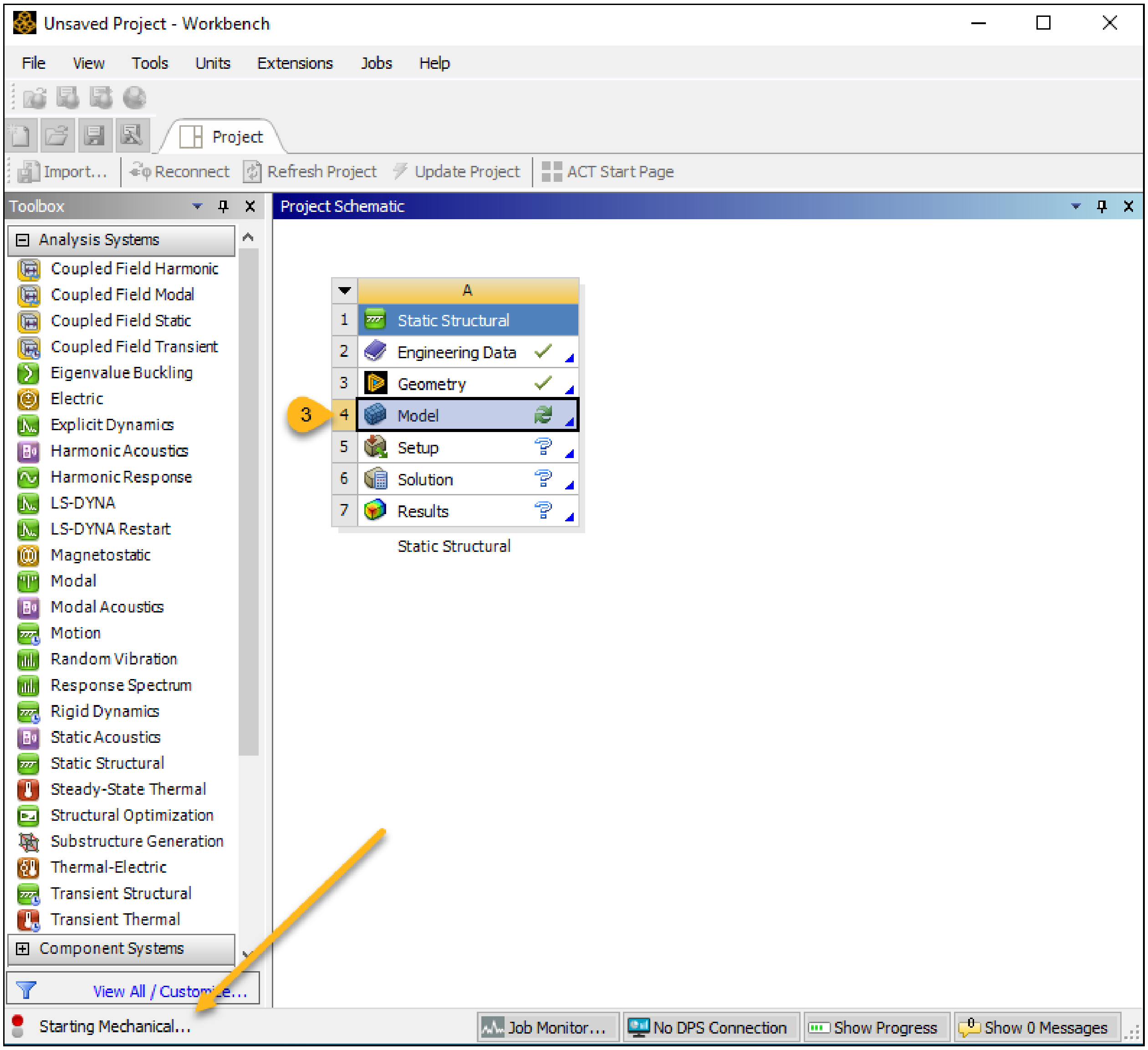
In this step, we attach the geometry file in Workbench and then open the Mechanical application. Once in Mechanical, the first thing we will do is set units.

1. Right-click the **Geometry** cell in the Static Structural system and select **Import Geometry** > **Browse**.

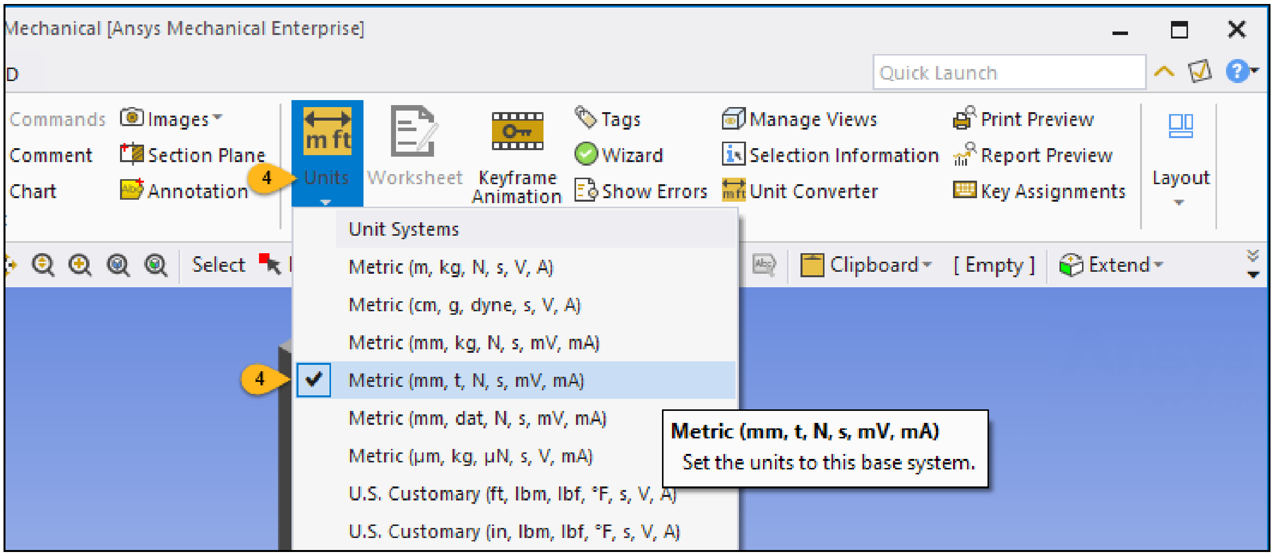
2. Navigate to the bridge with baseplate geometry file, select it, and click **Open** to add it to the analysis. A green check mark appears next to the Geometry cell in the Project Schematic when the geometry is added.



3. Double-click the **Model** cell to launch the Mechanical application. A message "Starting Mechanical" will show up in the status bar in the bottom, left corner. It may take a couple minutes for the application to open and attach the geometry.



4. Once the Mechanical application is open and you see a "Ready" message in the status bar, click the **Home** tab. Select **Units** from the ribbon and then select **Metric** (mm, t, N, s, mV, mA).

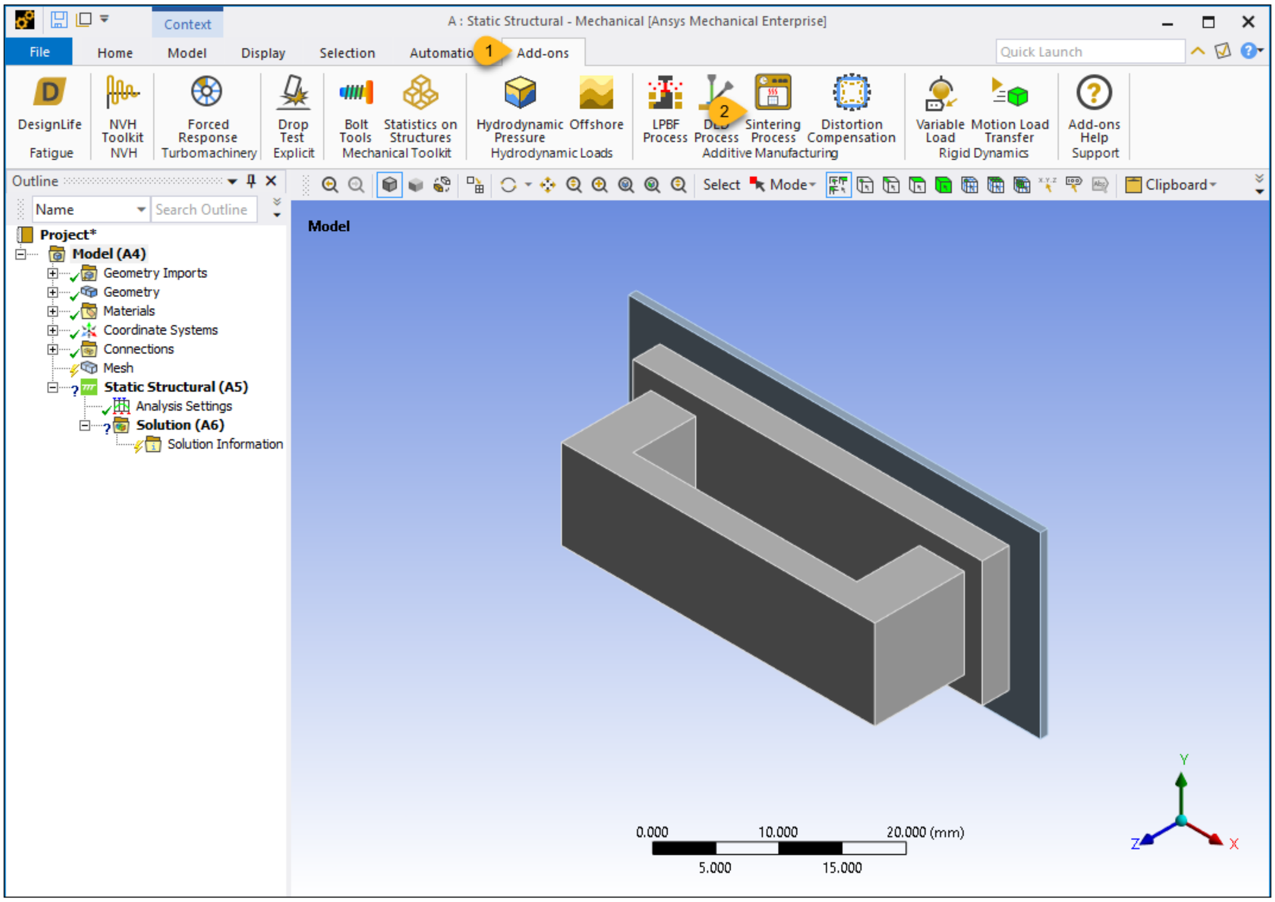


5.4. Load Sintering Process Add-on

This step includes the loading of the Sintering Process Add-on and opening of the Sintering Process Wizard.

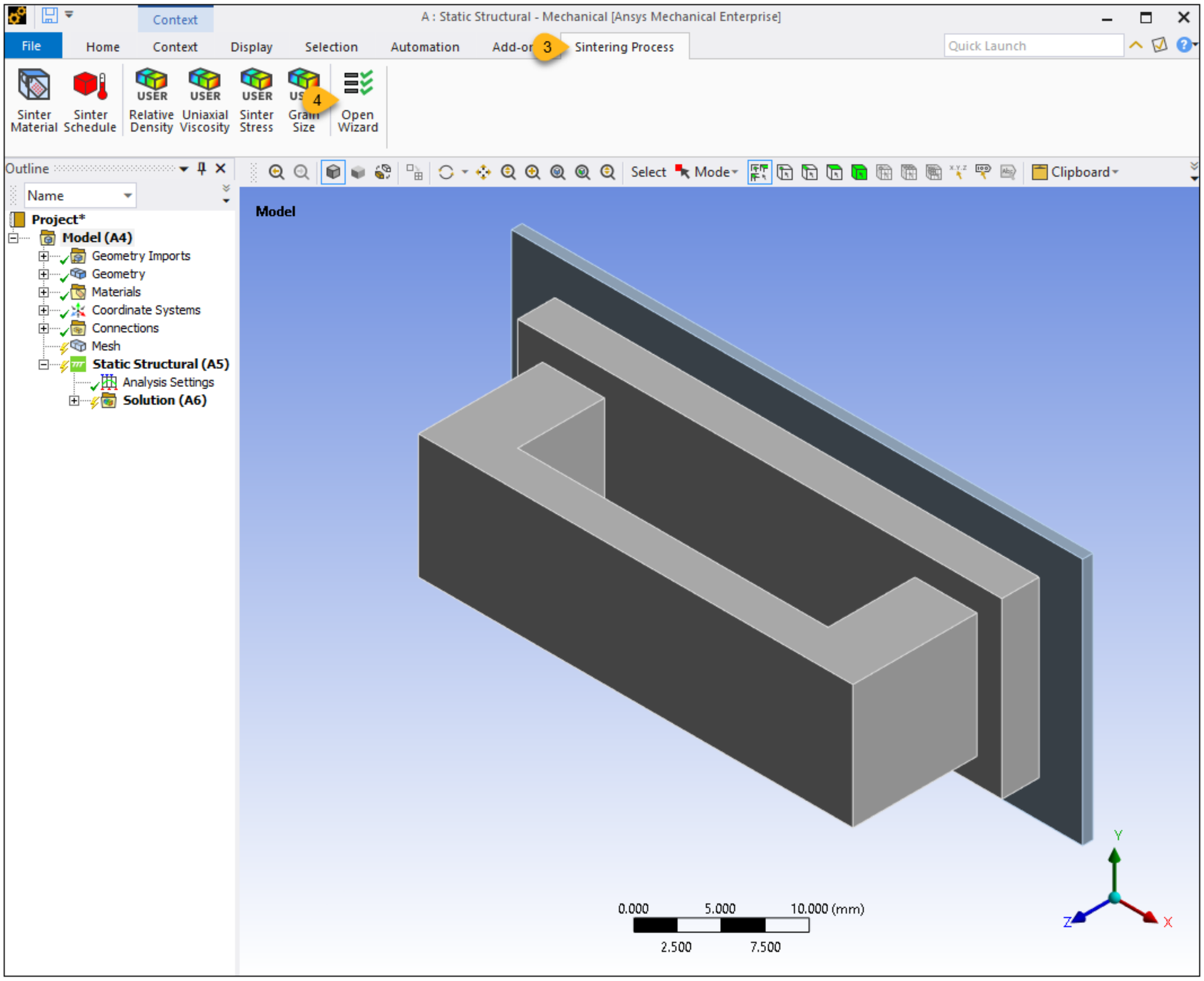
1. Click the **Add-ons** tab in the ribbon.

2. In the Additive Manufacturing group, click **Sintering Process** to load the add-on.



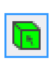
3. Click the **Sintering Process** tab in the ribbon.

4. Click **Open Wizard** from the ribbon.

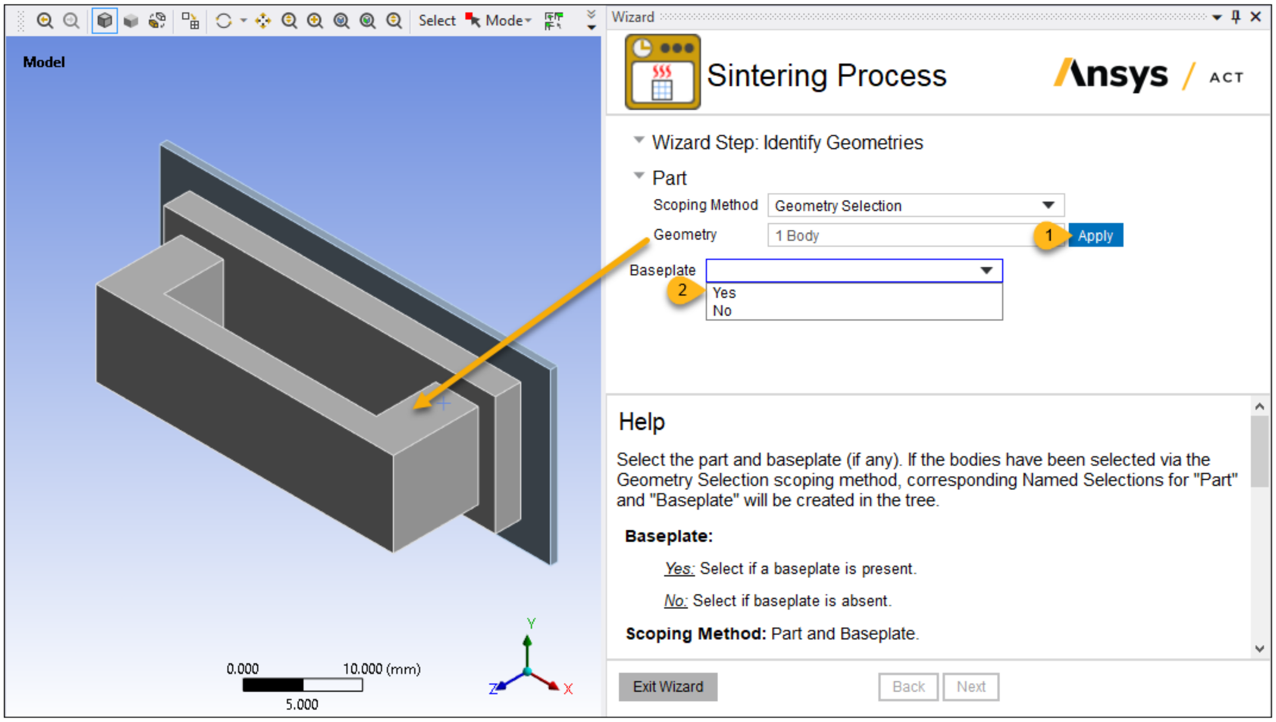


5.5. Wizard Step 1 - Identify Geometries

Identify which body is the green part and which body is the base.

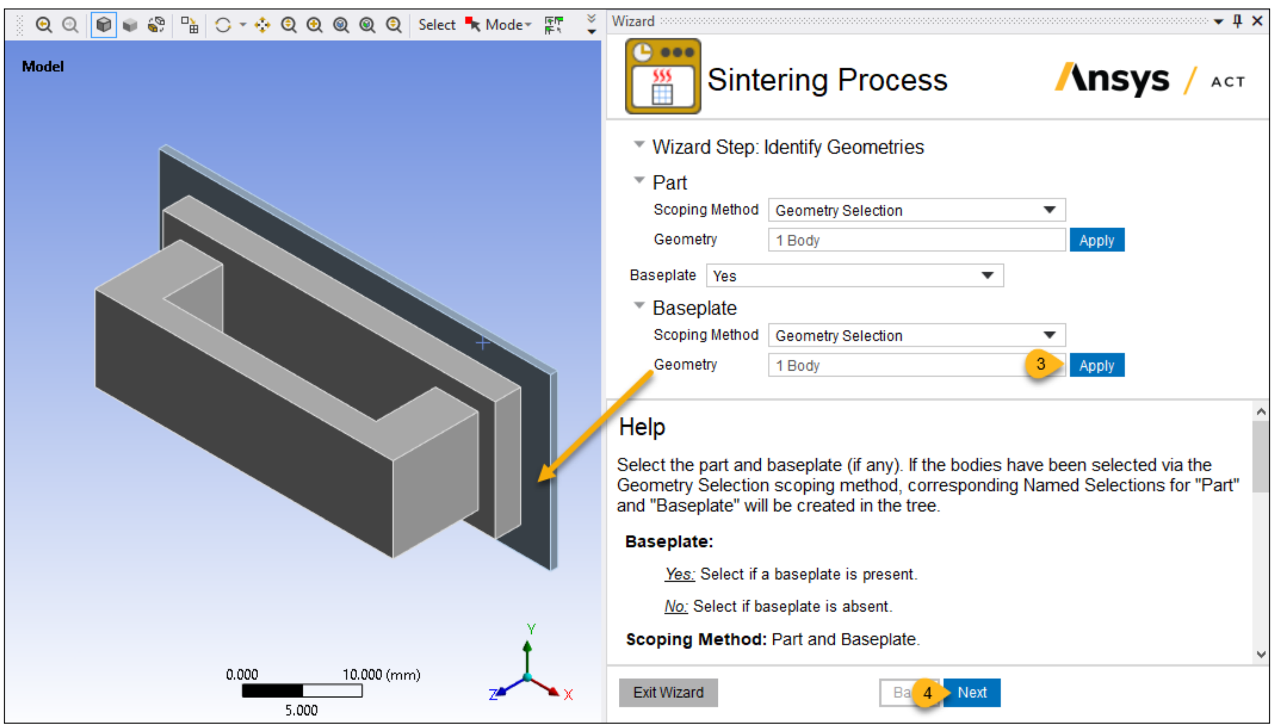
1. Use the Body picker  to select the bridge body and click **Apply** in the Geometry field under Part.

2. Click **Yes** in the Baseplate field because this model has a baseplate.

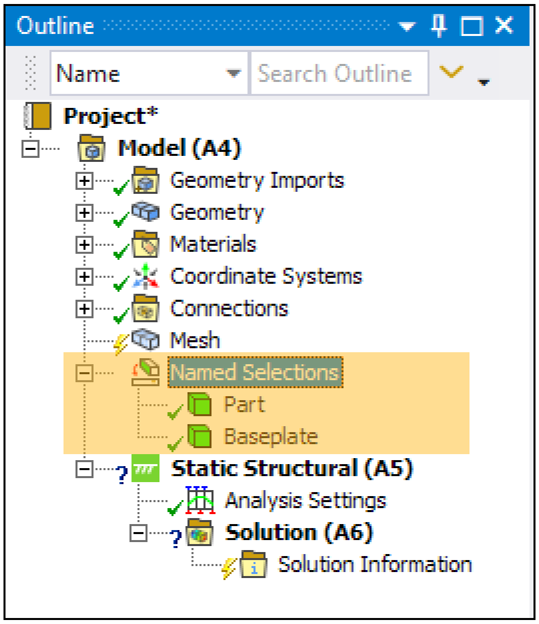


3. Select the base body and click **Apply** in the Geometry field under Baseplate.

4. Click **Next** to move to the next step.



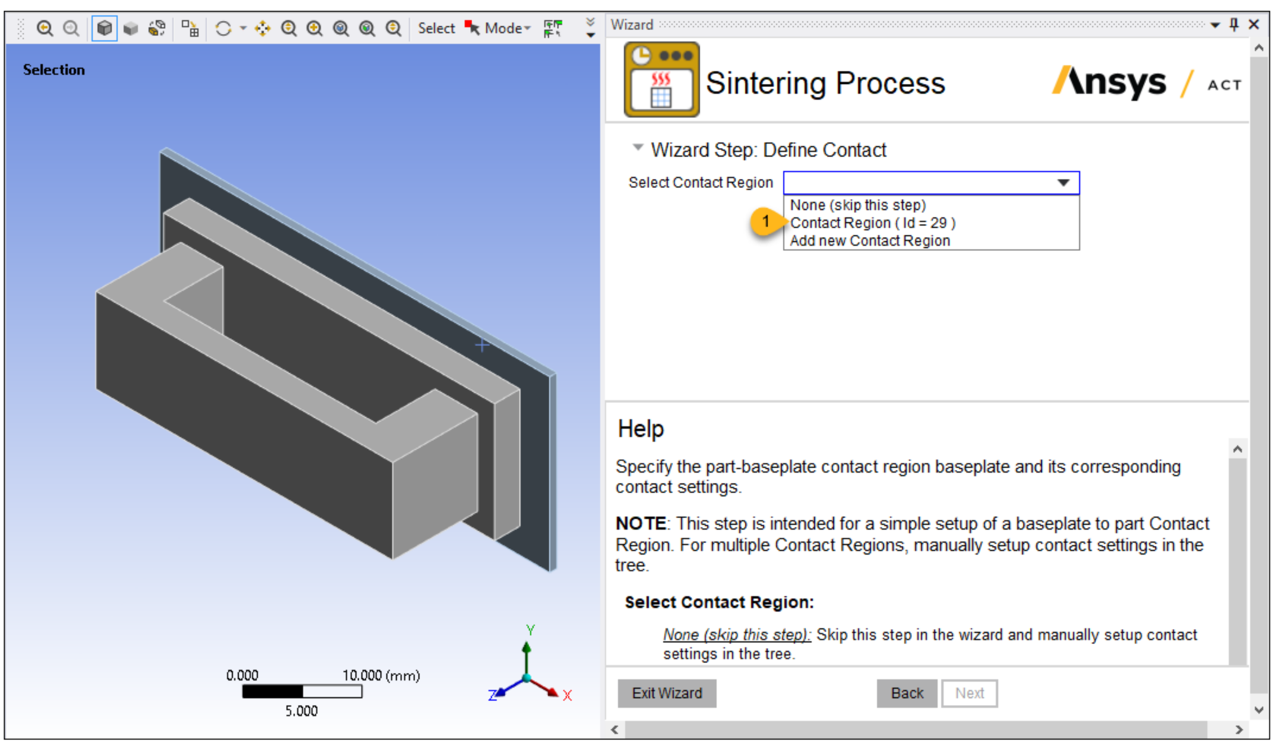
When this step is completed, Named Selections for the part body (Part) and base body (Baseplate) have been added to the Project tree. These will be used later for the creation of other Named Selections.



5.6. Wizard Step 2 - Define Contact

Contact between the part and baseplate is automatically detected by default in Mechanical because the bodies touch one another (they are mutually tangent). The wizard lets you use this automatically detected contact region, which is indicated by a specific contact region ID number, or you can specify a new contact region or skip the step and define contact later. We will use the automatically detected contact region.

1. Choose **Contact Region (Id = 29)** for Selected Contact Region.



2. We will use the recommended default settings for the contact interface between the part and the baseplate. The baseplate is large and stiff enough that we can ignore flexibility and assume a rigid baseplate. This assumption speeds up the simulation time. We expect the part to slide on the base as the part shrinks during sintering so we will be sure Small Sliding option is off.

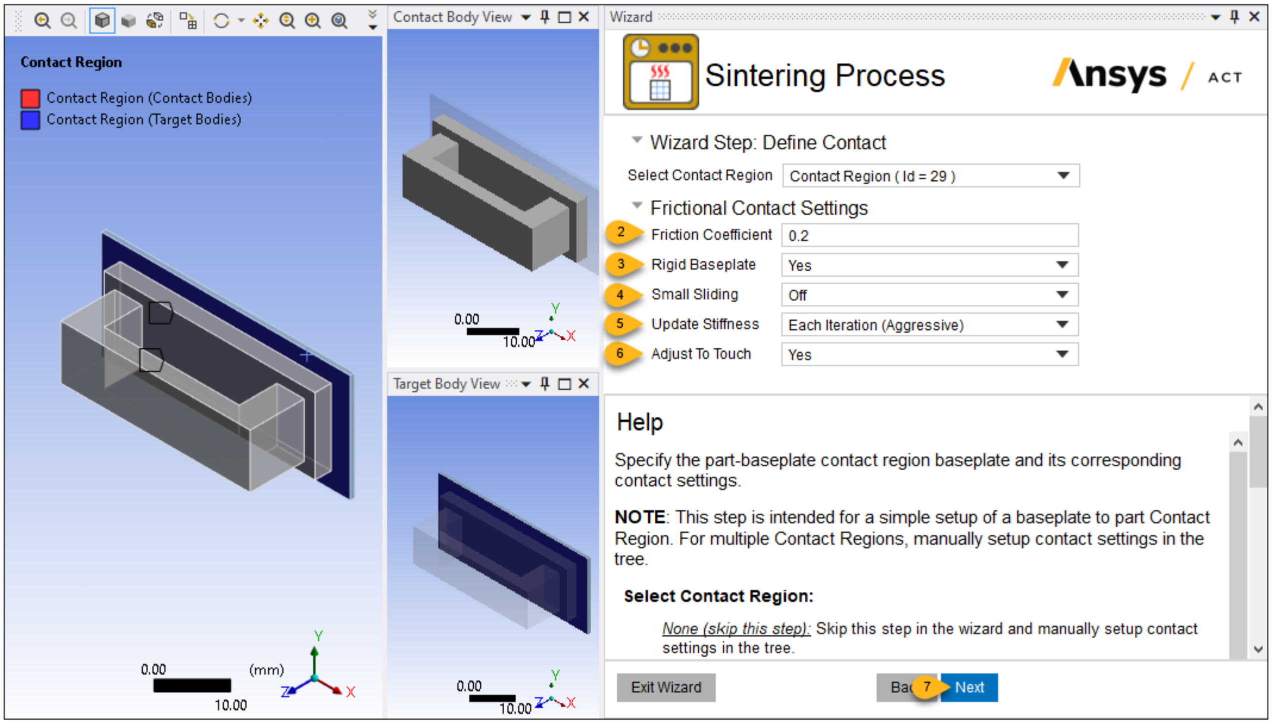
   Keep **0.2** (default) for Friction Coefficient.

3. Keep **Yes** (default) for Rigid Baseplate.

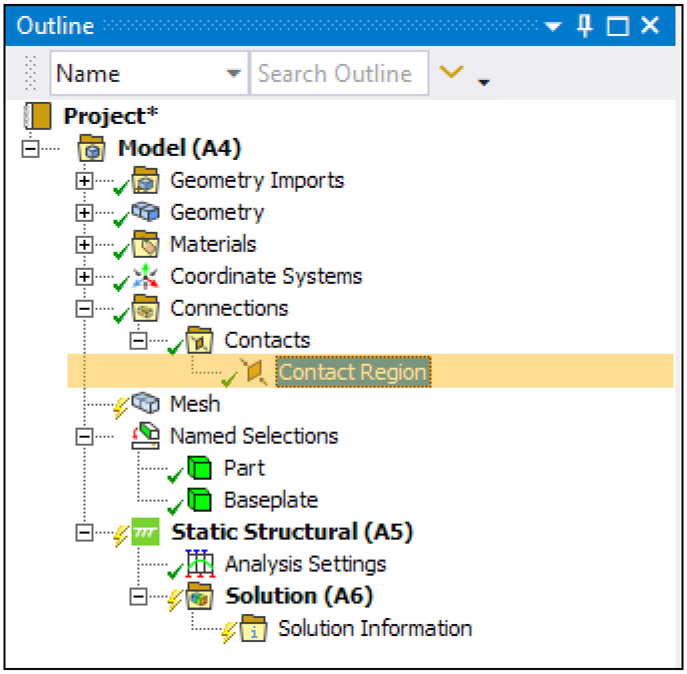
4. Keep **Off** (default) for Small Sliding.

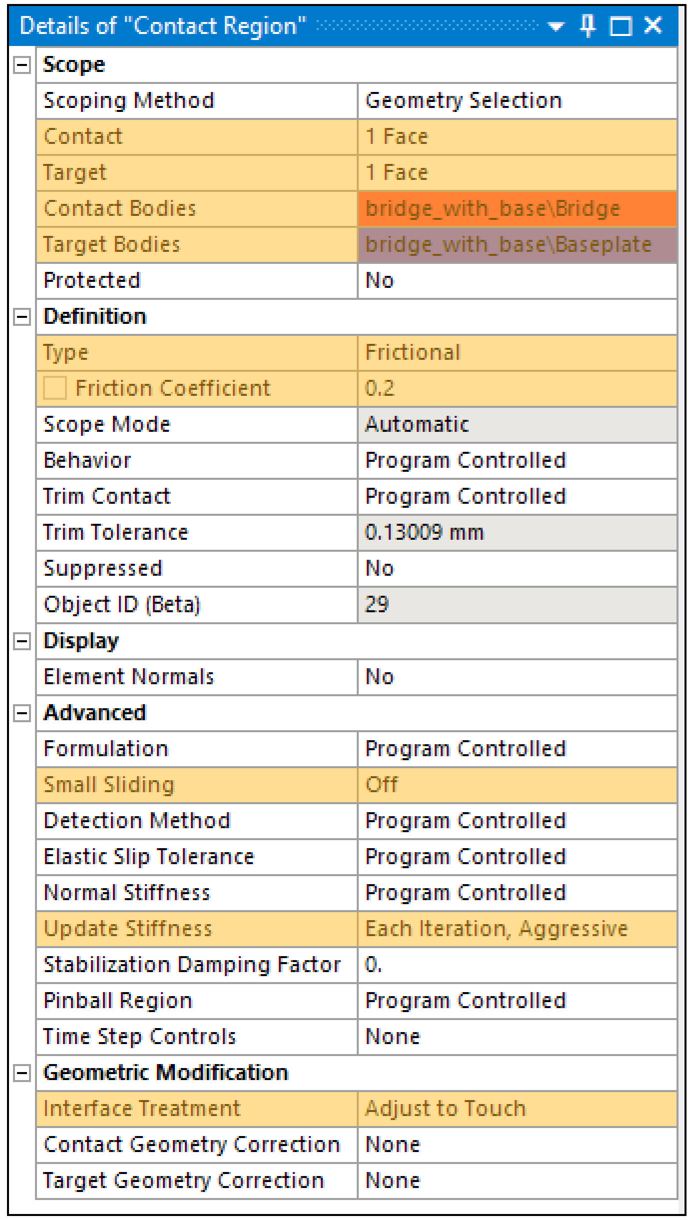
5. Keep **Each Iteration (Aggressive)** (default) for Update Stiffness.

6. Keep **Yes** (default) for Adjust To Touch.



When this step is completed, a Contact Region object has been added to the Project tree.

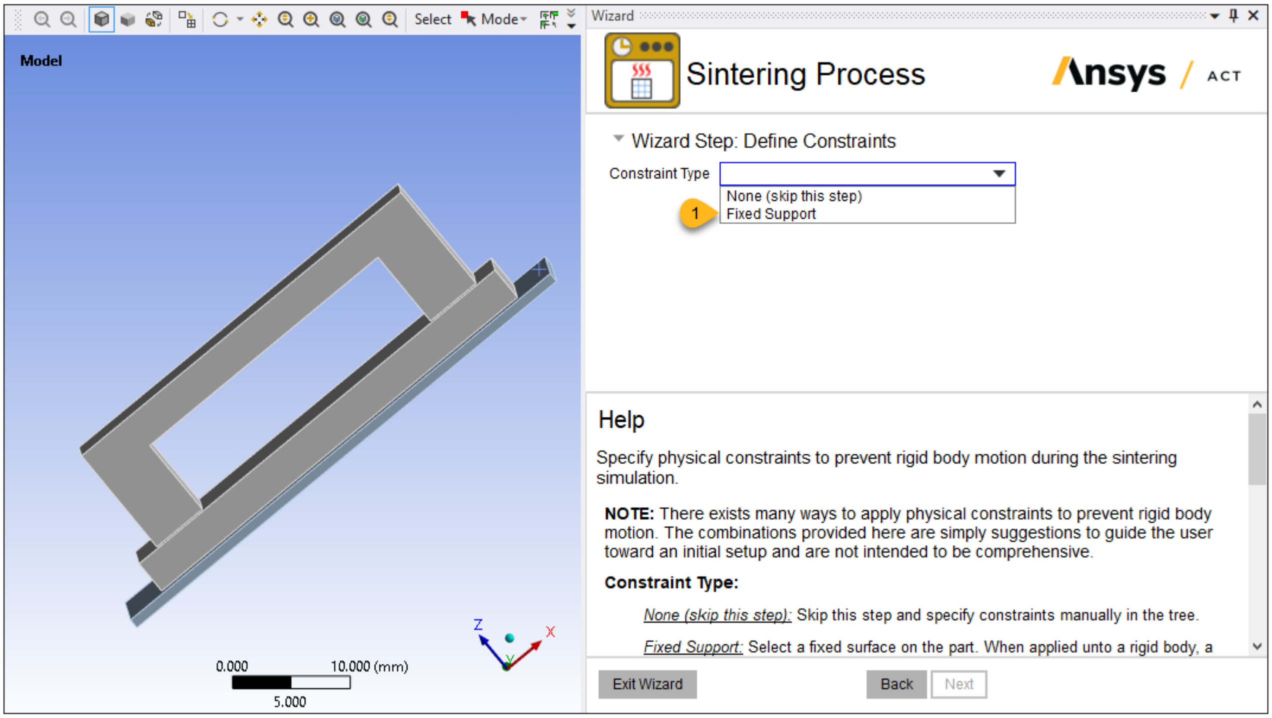




5.7. Wizard Step 3 - Define Constraints

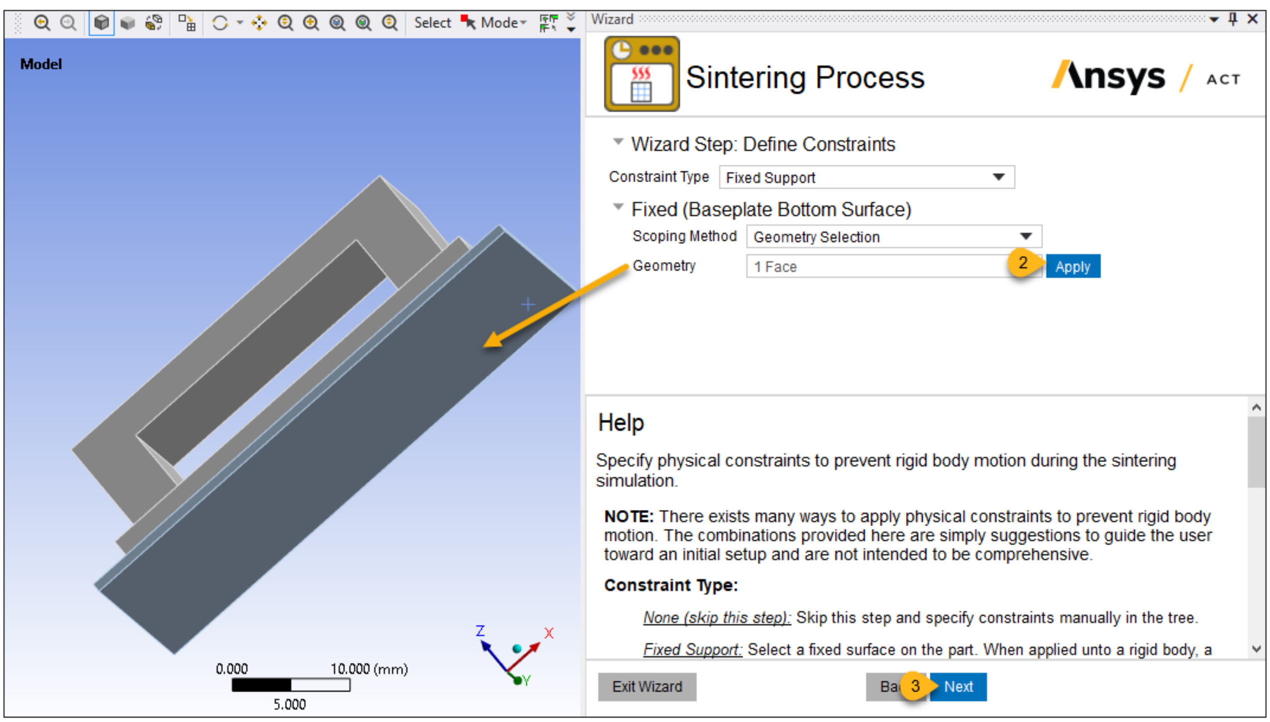
In this step we'll apply a fixed support to the underside of the base plate.

1. Click **Fixed Support** in the Constraint Type field.

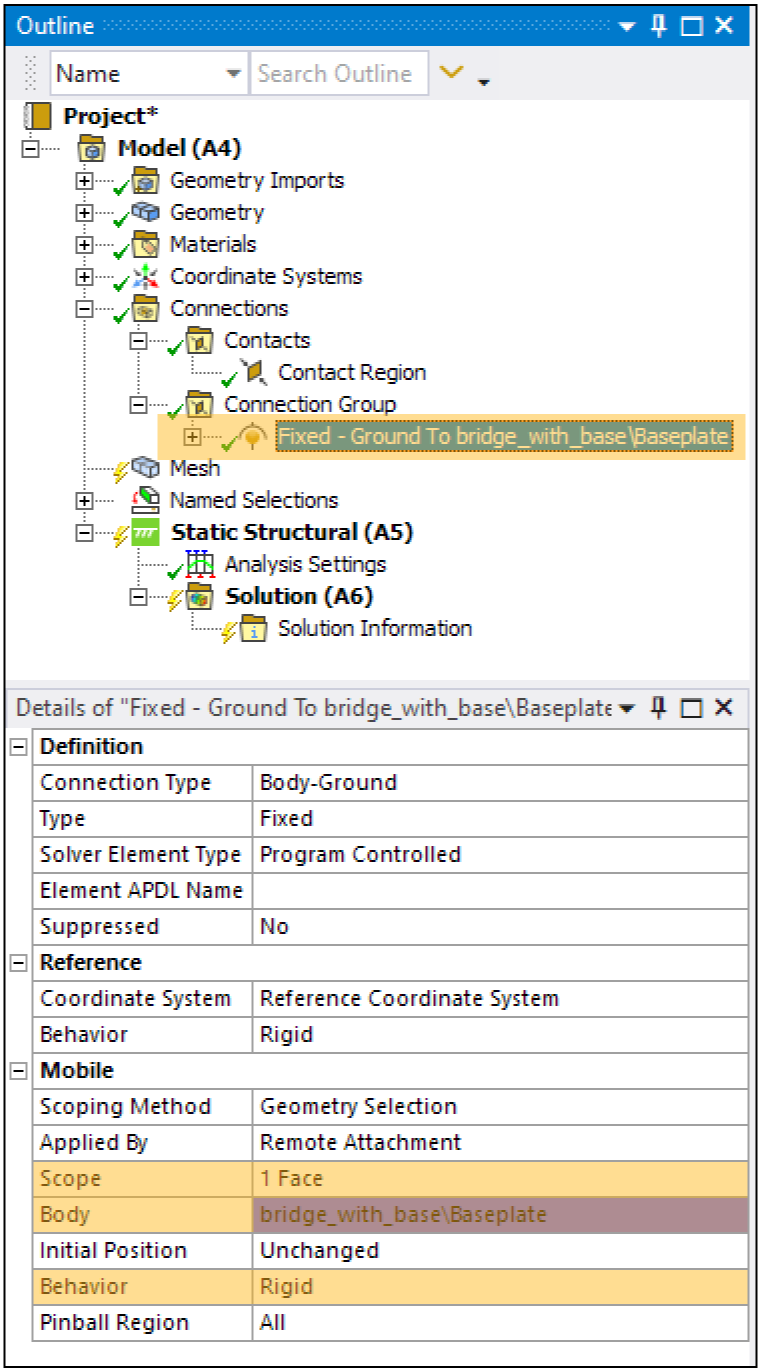


2. Rotate the model in the graphics window, use the Face picker to select the underside surface of the base, and click **Apply** in the Geometry field.

3. Click **Next** to move to the next step.



When applied to a rigid body, a body-to-ground fixed joint will be created instead of the Fixed Support boundary condition, as shown in the Project tree.



5.8. Wizard Step 4 - Generate Mesh

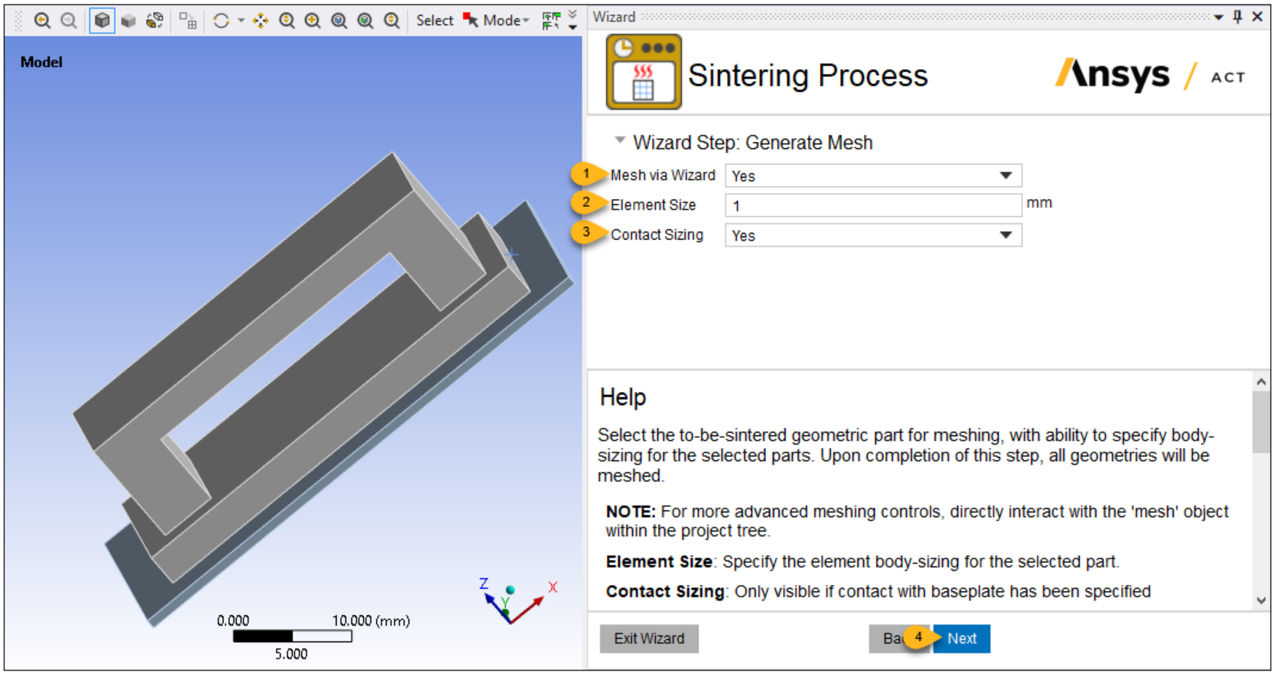
We'll specify a mesh size of 1 mm so that there are at least three elements through the height of the bridge span.

1. Keep **Yes** (default) for Mesh via Wizard.

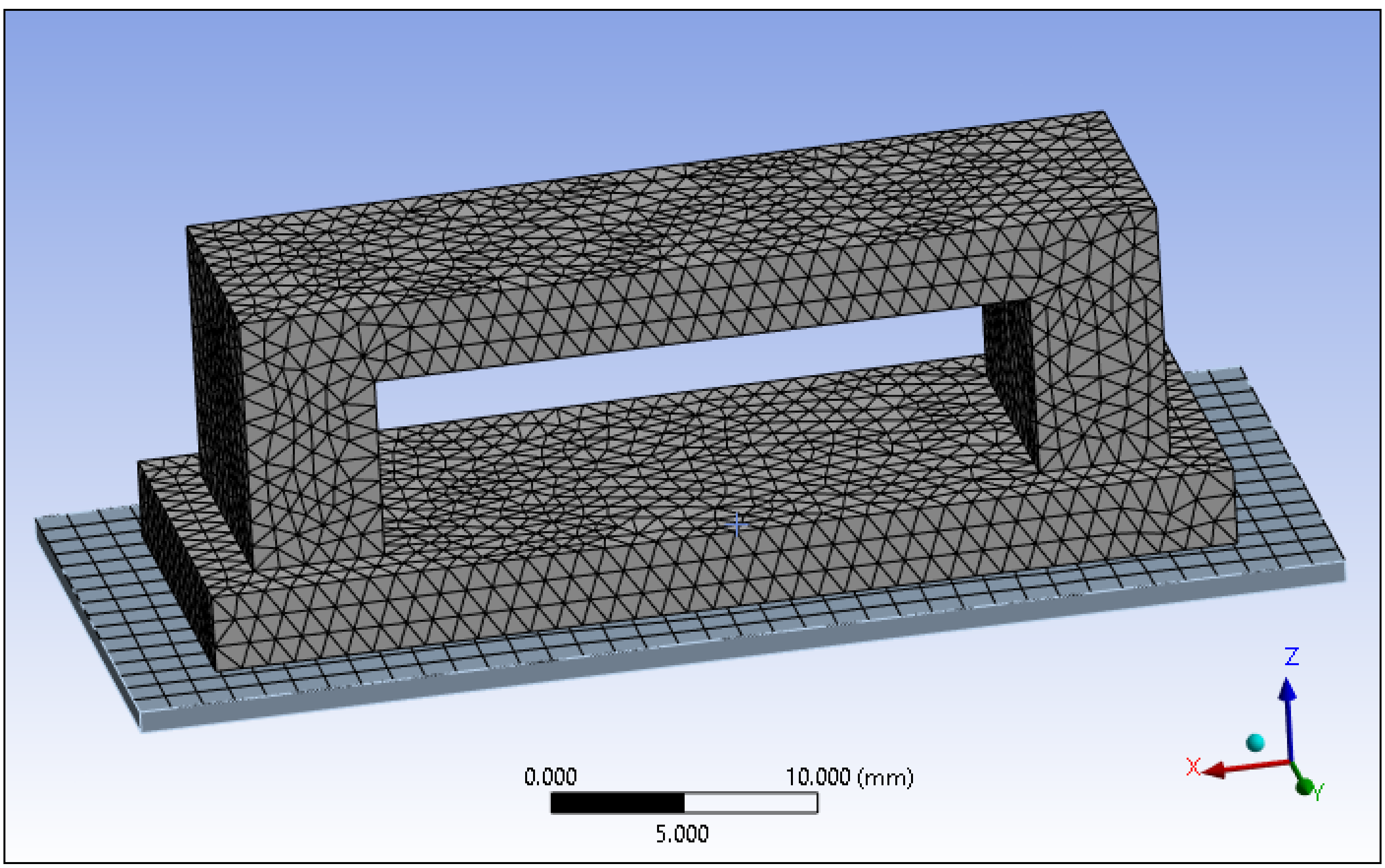
2. Enter **1** for Element Size.

3. Keep **Yes** (default) for Contact Sizing (recommended).

4. Click **Next** to move onto the next step.



When this step is completed, mesh objects have been added to the Project tree: a Body Sizing object for the part and baseplate, and Contact Sizing for improved contact traction. The resulting mesh is shown here.

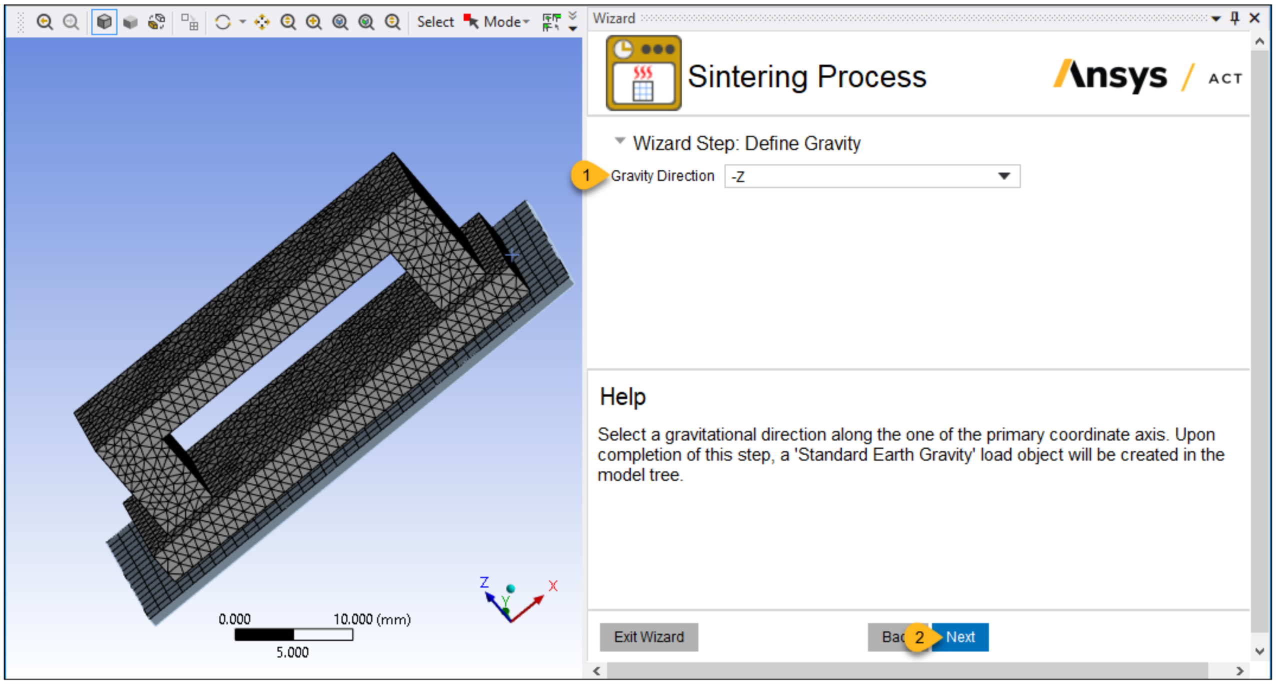


5.9. Wizard Step 5 - Define Gravity

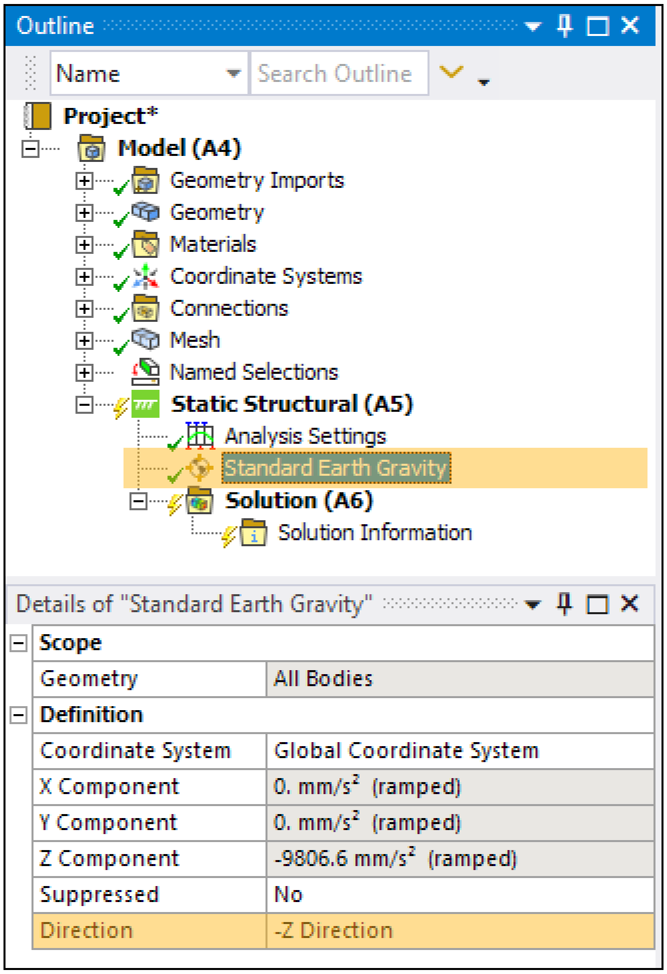
A gravitational acceleration is applied so that frictional forces between the bridge and the baseplate prevent rigid body motion.

1. Keep **-Z** (default) for Gravity Direction.

2. Click **Next** to move to the next step.



A Standard Earth Gravity object has been added to the Project tree.



5.10. Wizard Step 6 - Define Sinter Material

Use an Ansys-defined sinter material or a user-defined one. We'll use stainless steel 316L (PIM) from our library. See Sintering in the *Material Reference* for more information.

1. Choose **316L (PIM)** for Material.

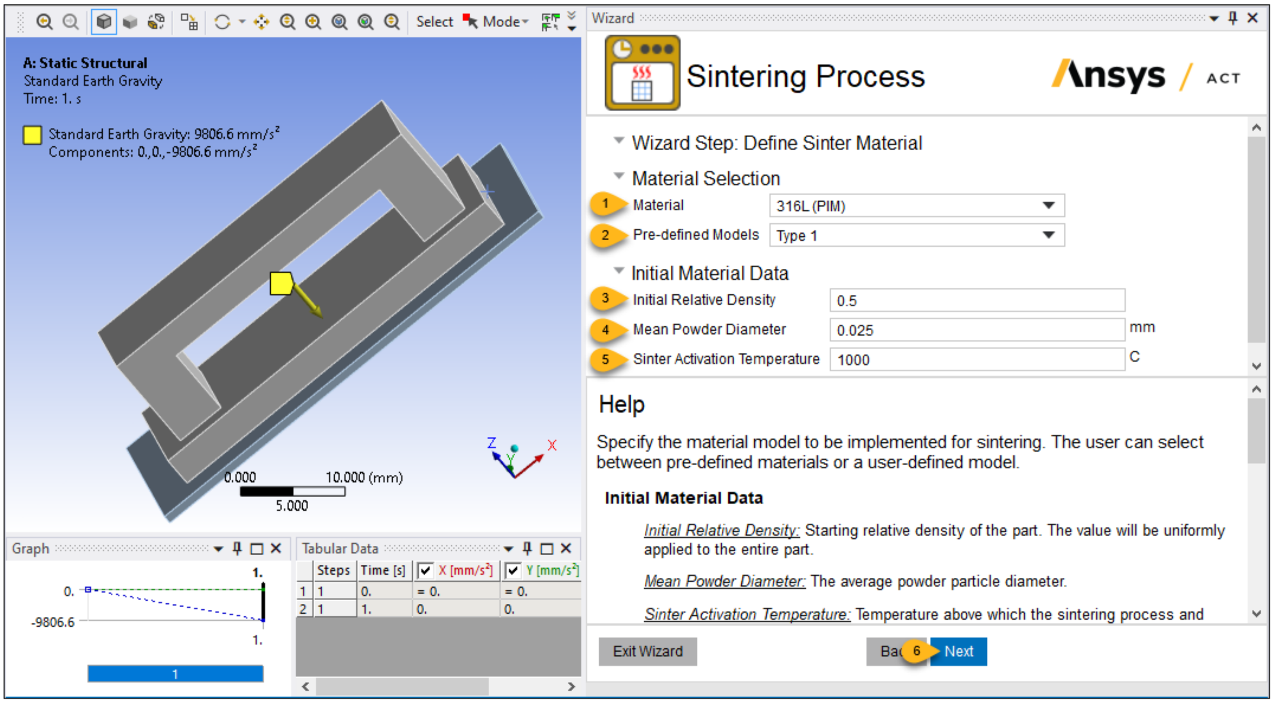
2. Choose **Type 1** for Pre-defined Models.

3. Keep **0.5** (default) for Initial Relative Density. This value—50% dense—will be uniformly applied to the entire part.

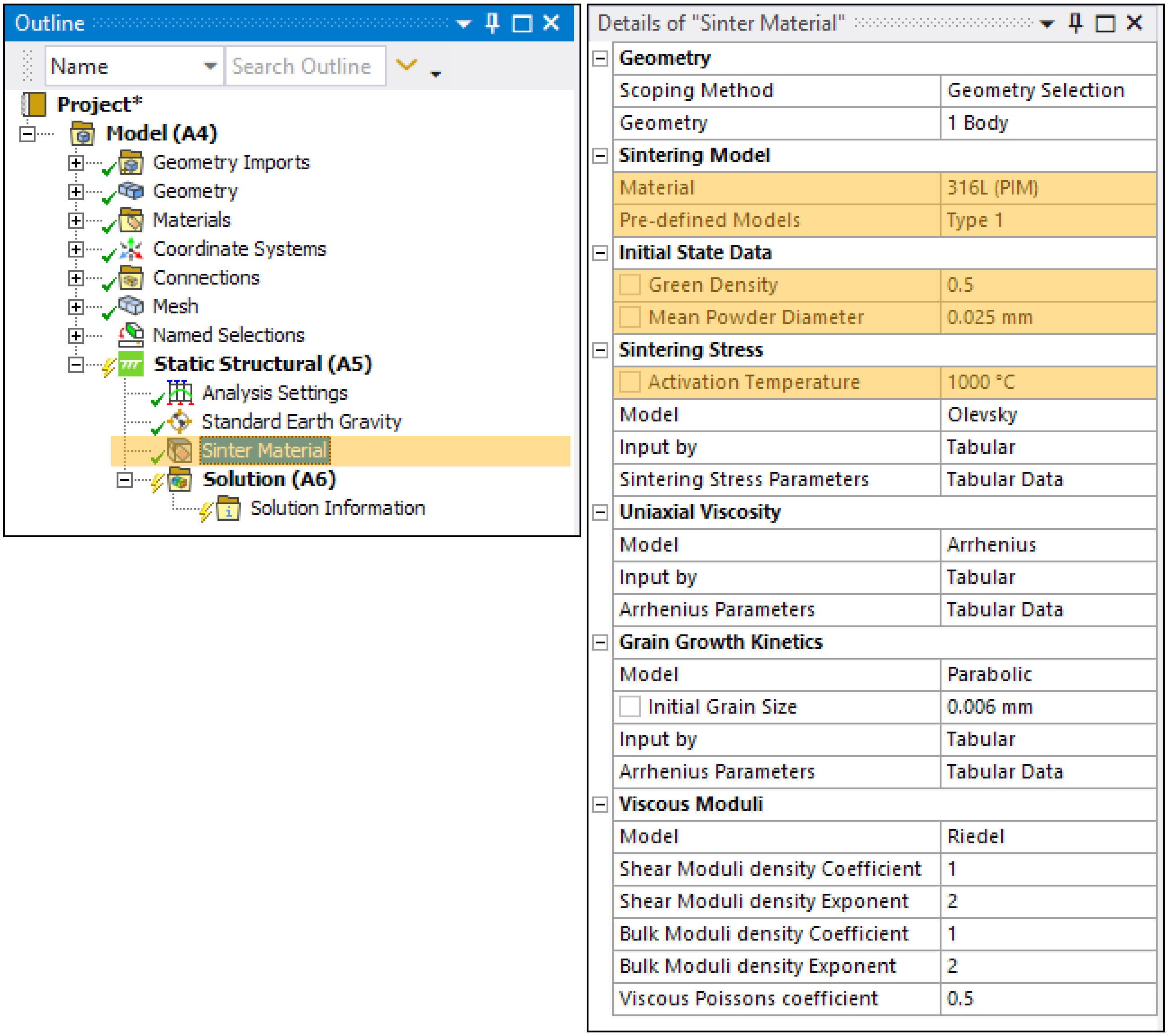
4. Keep **0.025** (default) for Mean Powder Diameter. This is the average powder diameter used for building of the part.

5. Enter **1000** for Sinter Activation Temperature. This is the temperature above which sintering stress is non-zero and shrinkage can occur.

6. Click **Next** to move to the next step.



When this step is completed, a Sinter Material object has been added in the Project tree with the required coefficients and exponents and other material model inputs populated based upon the model type selected in the wizard.

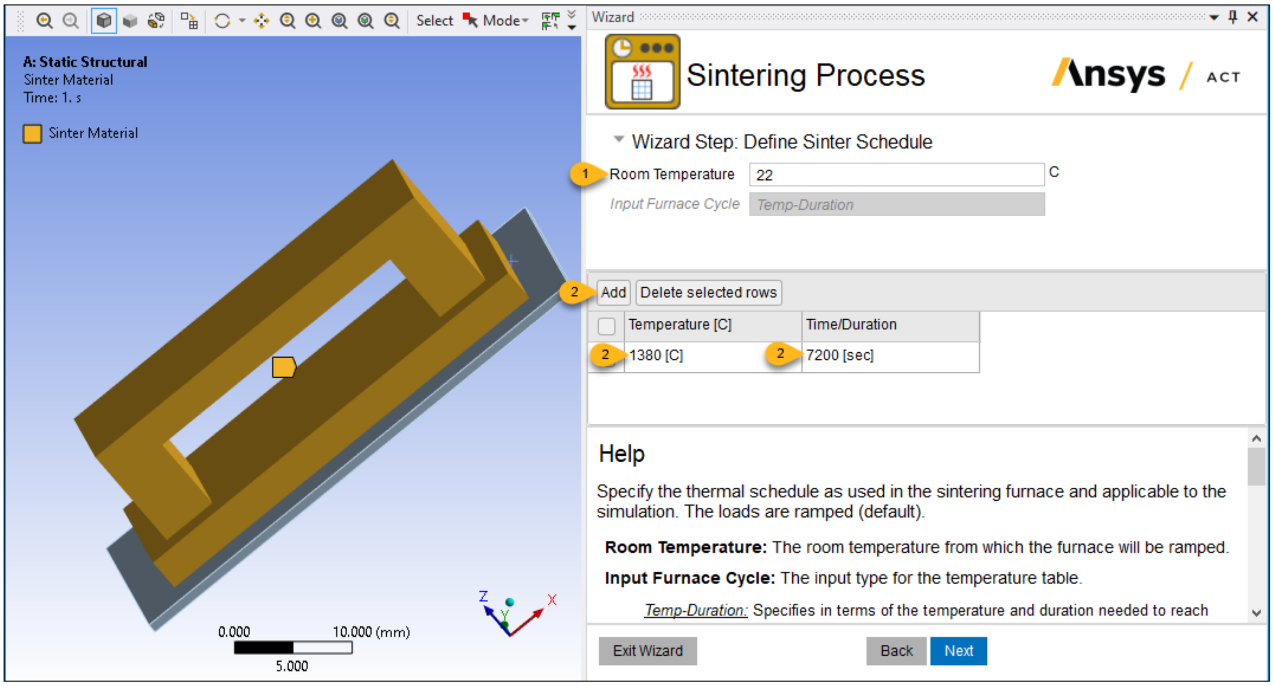


5.11. Wizard Step 7 - Define Sinter Schedule

The sintering furnace schedule is defined in this step. As shown in the problem description (p. 103), the temperature ramps up to 1380°C in two hours and holds at that temperature for one more hour.

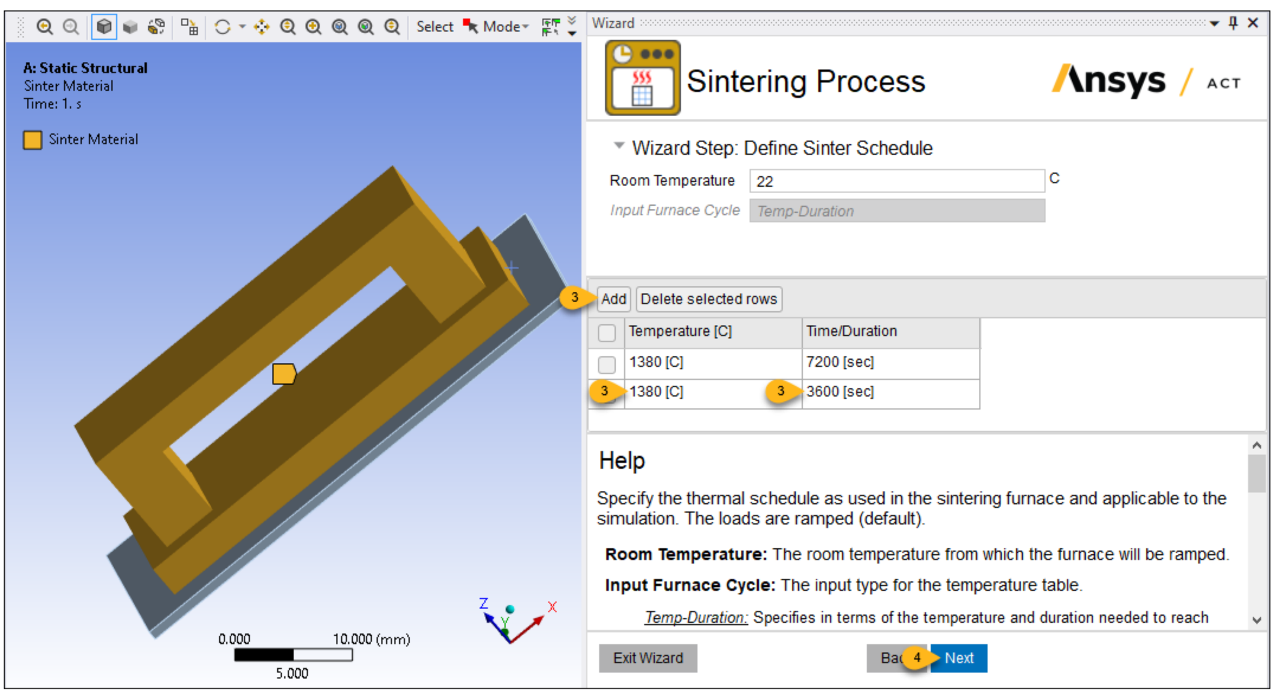
1. Keep **22** (default) for Room Temperature.

2. Click **Add**, and enter **1380** and **7200** for Temperature [C] and Time/Duration, respectively.

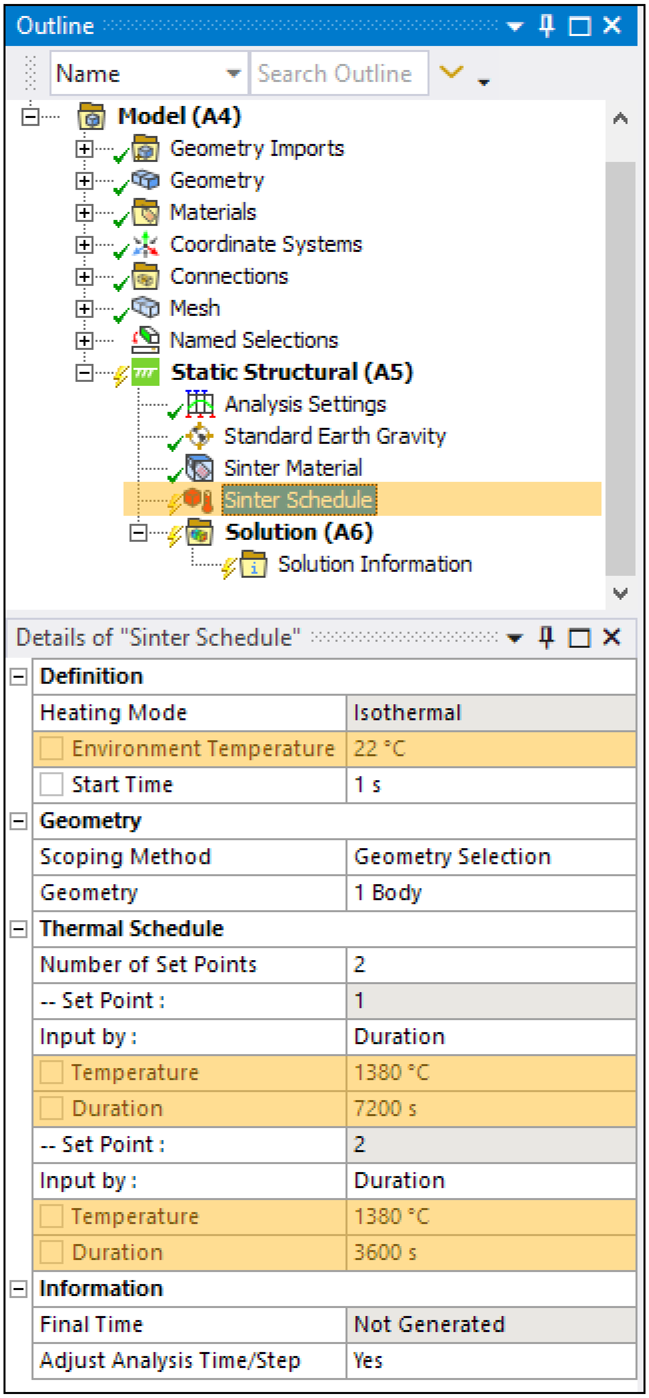


3. Click **Add** to insert a second row, and enter **1380** and **3600** for Temperature [C] and Time/Duration, respectively.

4. Click **Next** to move to the next step.



When this step is completed, a Sinter Schedule object has been added in the Project tree. The yellow lightening bolt next to the object indicates that there is an action required, which is the generation of the sinter schedule itself. We will do this after completing the wizard.



5.12. Wizard Step 8 - Define Results and Solver Settings

The final wizard step allows us to customize results and solver settings. We will use the recommended defaults.

1. Keep **Yes** (default) for Generate Result Objects.

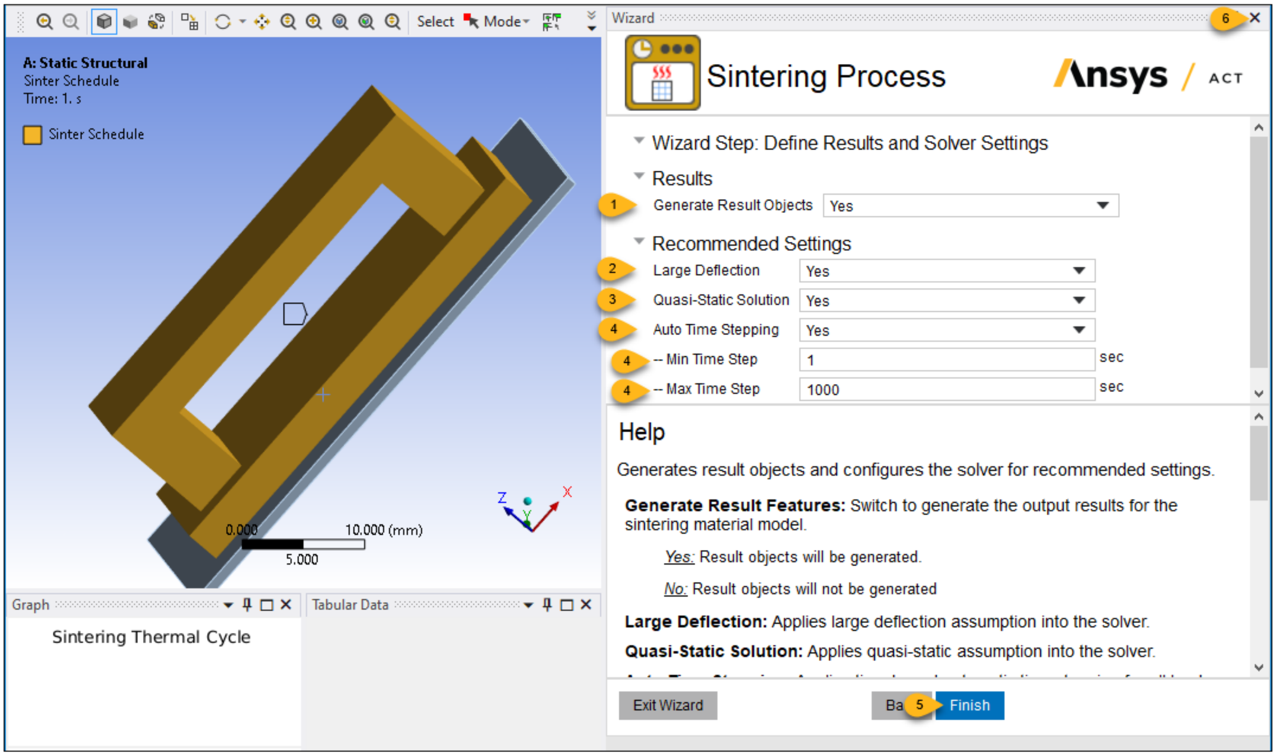
2. Keep **Yes** (default) for Large Deflection.

3. Keep **Yes** (default) for Quasi-Static Solution.

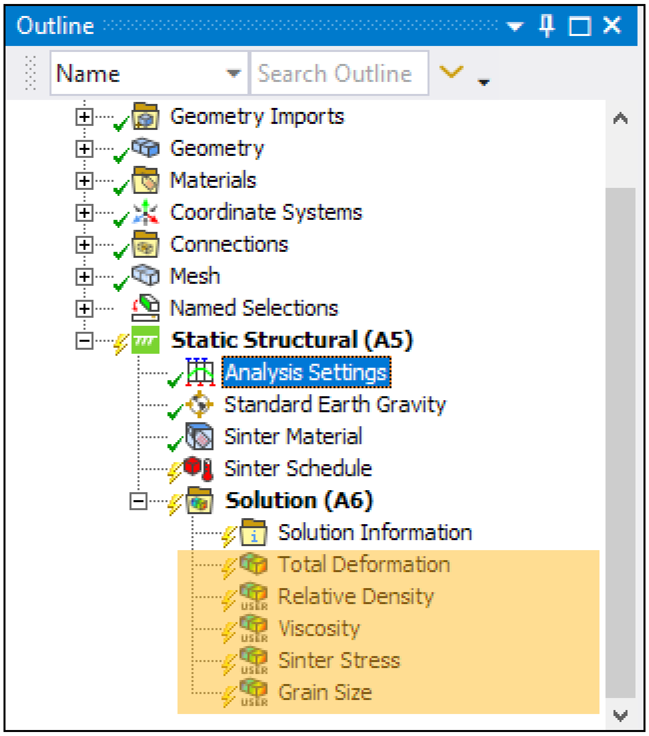
4. Keep **Yes** (default) for Auto Time Stepping, **1** (default) for Min Time Step, and **1000** (default) for Max Time Step.

5. Click **Finish** to complete the wizard.

6. Click **X** in the upper, right corner to close the wizard panel.



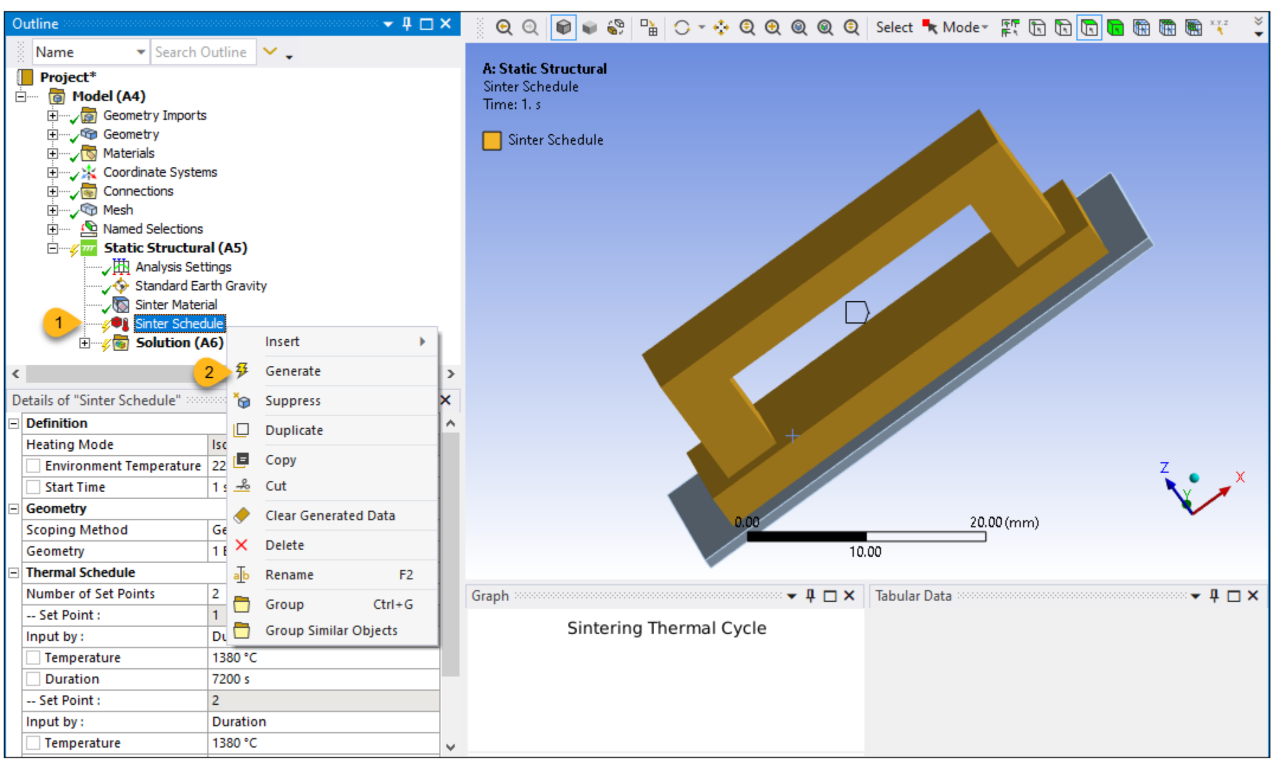
When this step is completed, several sintering-specific result items have been added in the Project tree. We will review those items once the solution is complete.



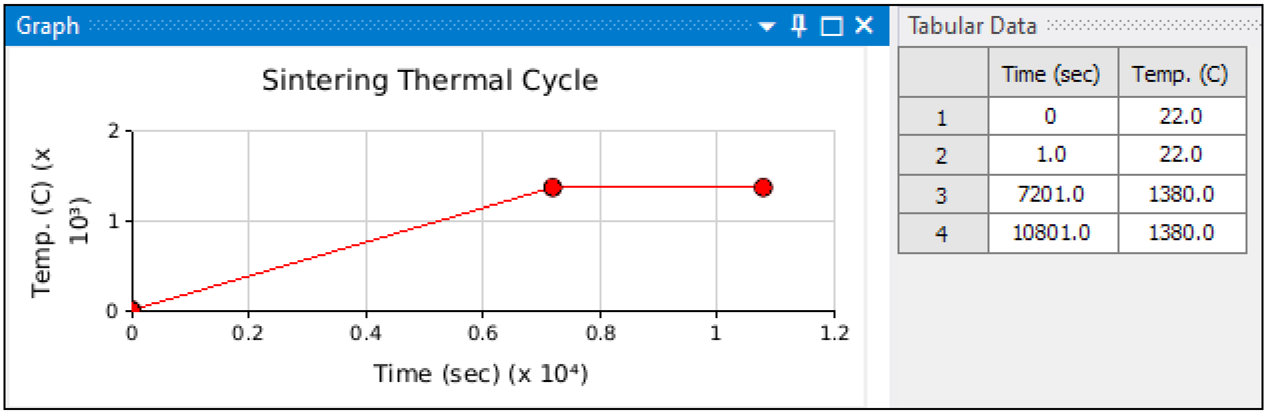
5.13. Generate Sinter Schedule and Solve

An optional step is to generate the sintering schedule before solving, perhaps to verify that the data is correct. Otherwise, generation of the schedule is done automatically when the solution is initiated.

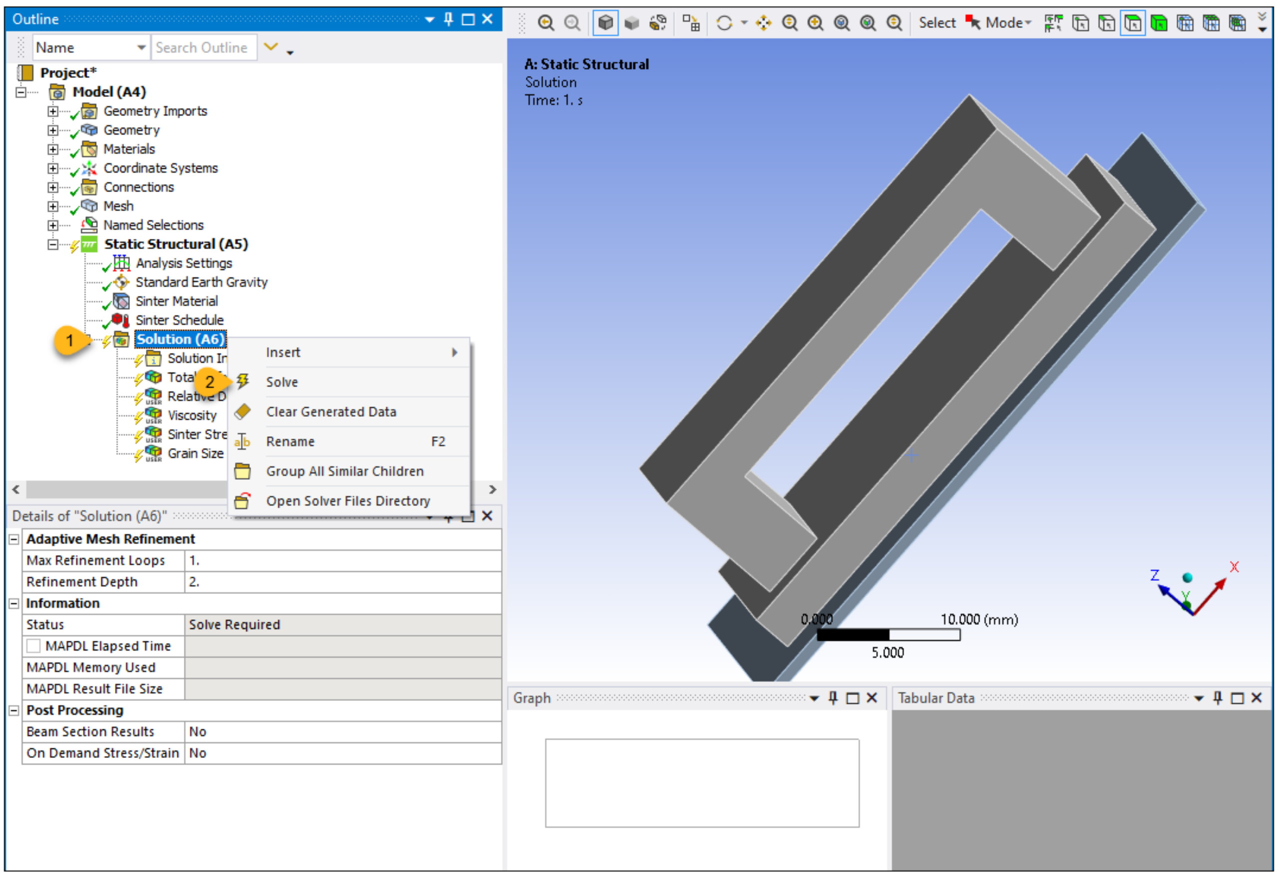
1. Under the Static Structural analysis, right-click **Sinter Schedule** and select **Generate**.



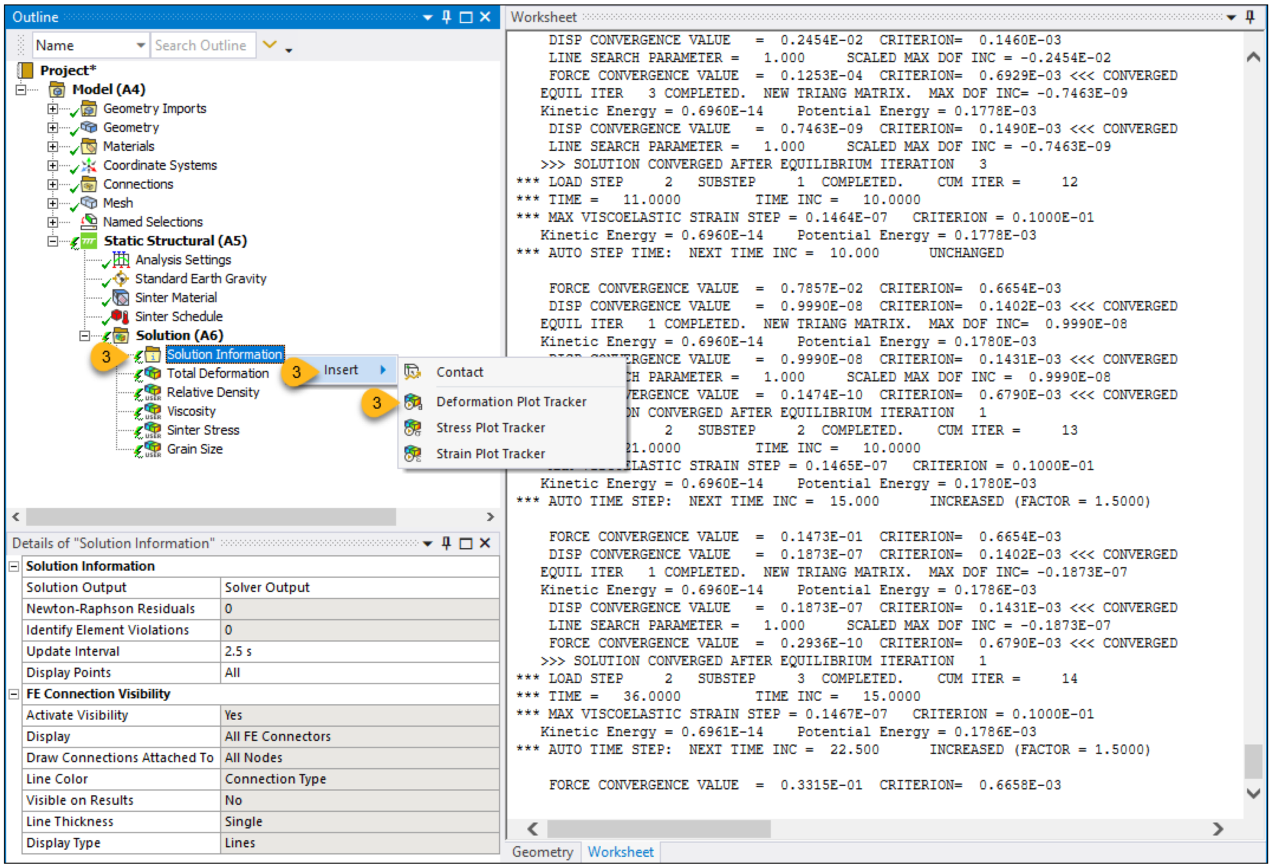
After a few seconds, a plot and table representing the sinter schedule appears below the graphics window. Note that a very small, insignificant step (1 second) is added at both the beginning and end of the furnace schedule to aid in simulation convergence.



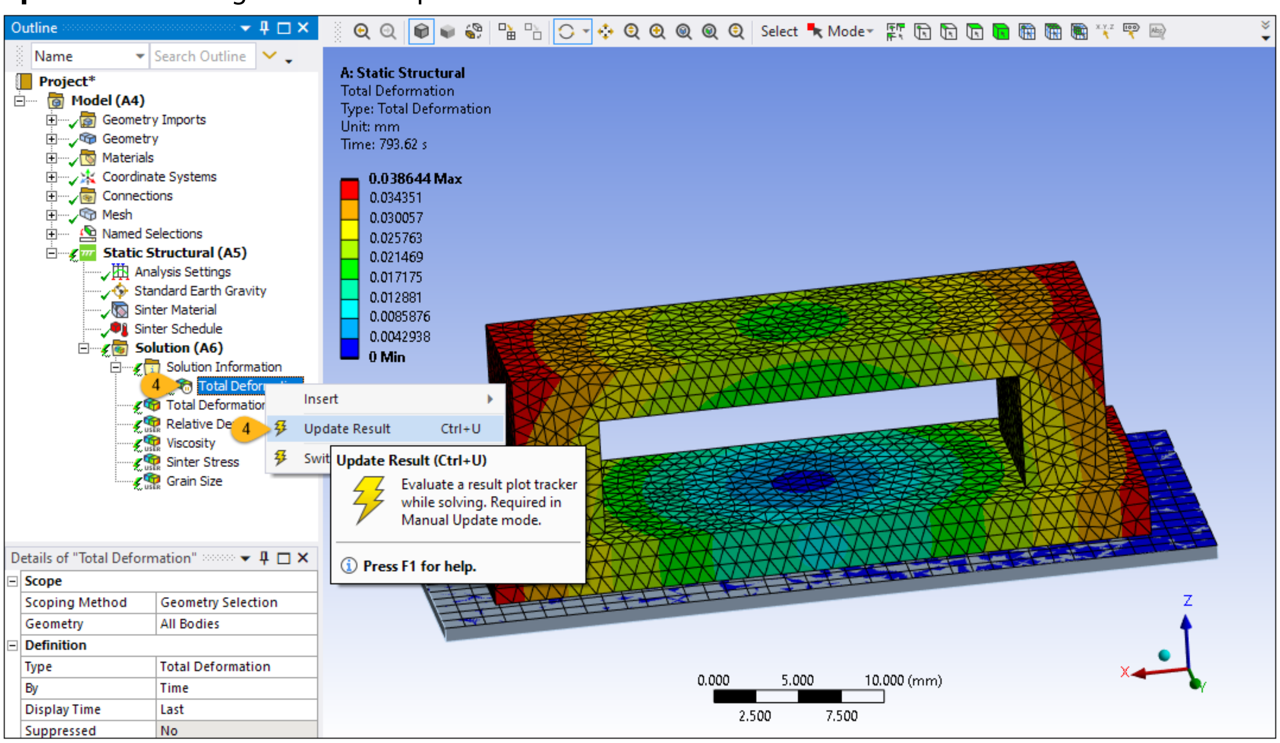
2. We are now ready to solve! Under the Static Structural analysis, right-click **Solution (A6)** and select **Solve** to initiate the solution.



3. While the solution is solving, right-click **Solution Information**, click **Insert**, and select **Deformation Plot Tracker** to see the deformation during solution.



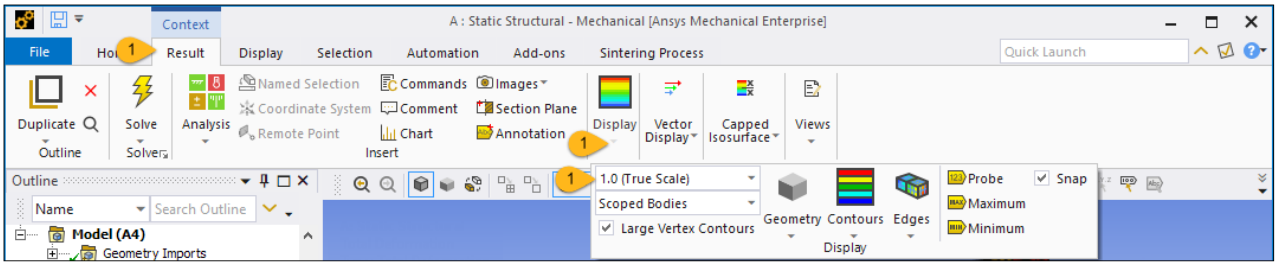
4. Occasionally during the solution, right-click the **Total Deformation** plot tracker and select **Update Result** to get a fresh snapshot of the deformation.



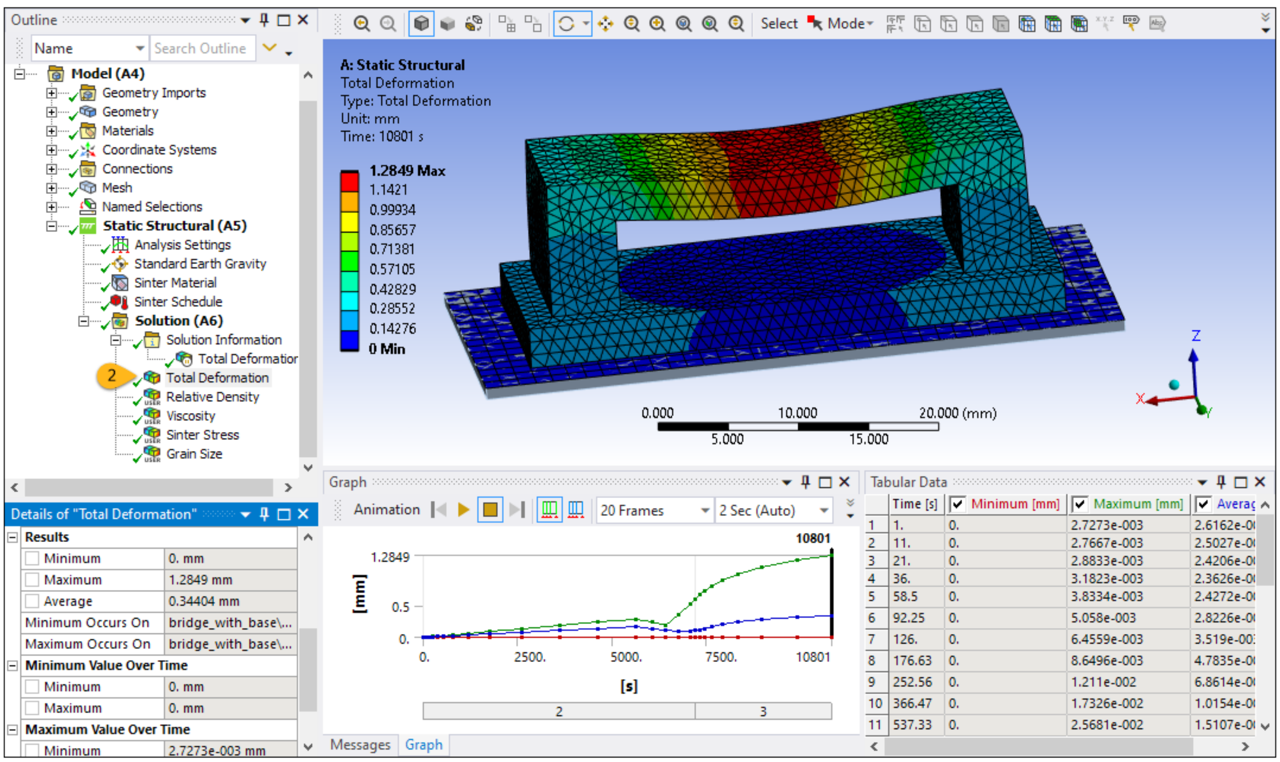
5.14. Review Results

Once the solution is complete, review results by clicking on the result items in the tree.

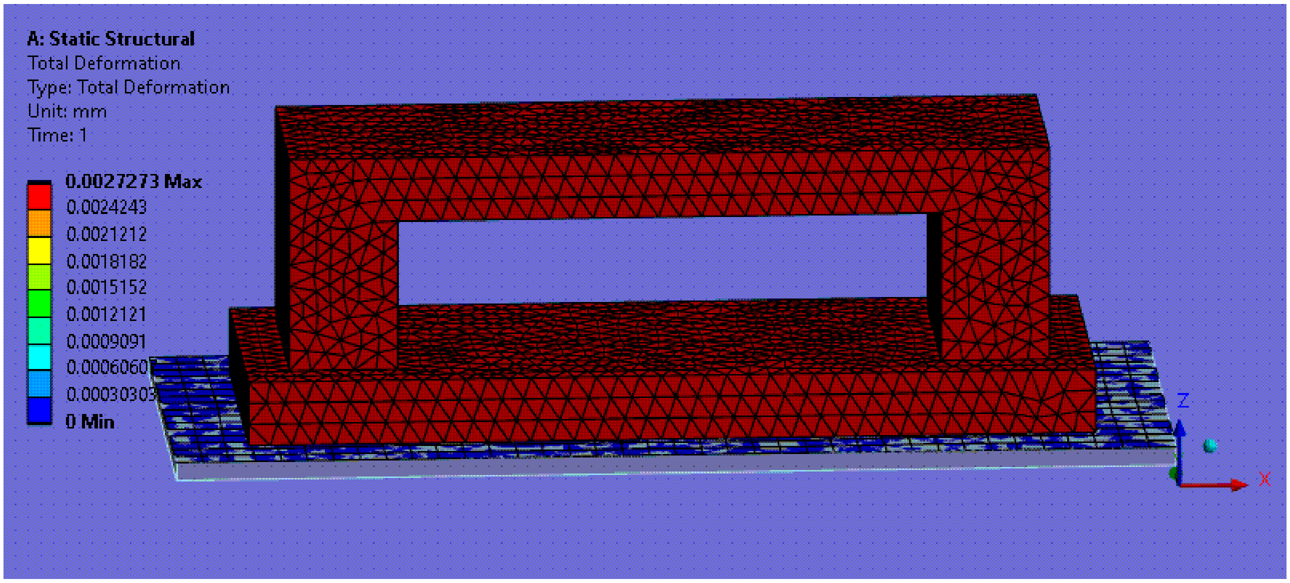
1. First, change the display scale to **1.0 (True Scale)** in the ribbon (**Result tab**) for a better view of deformation. (By default, the scale is set for an exaggerated display.)



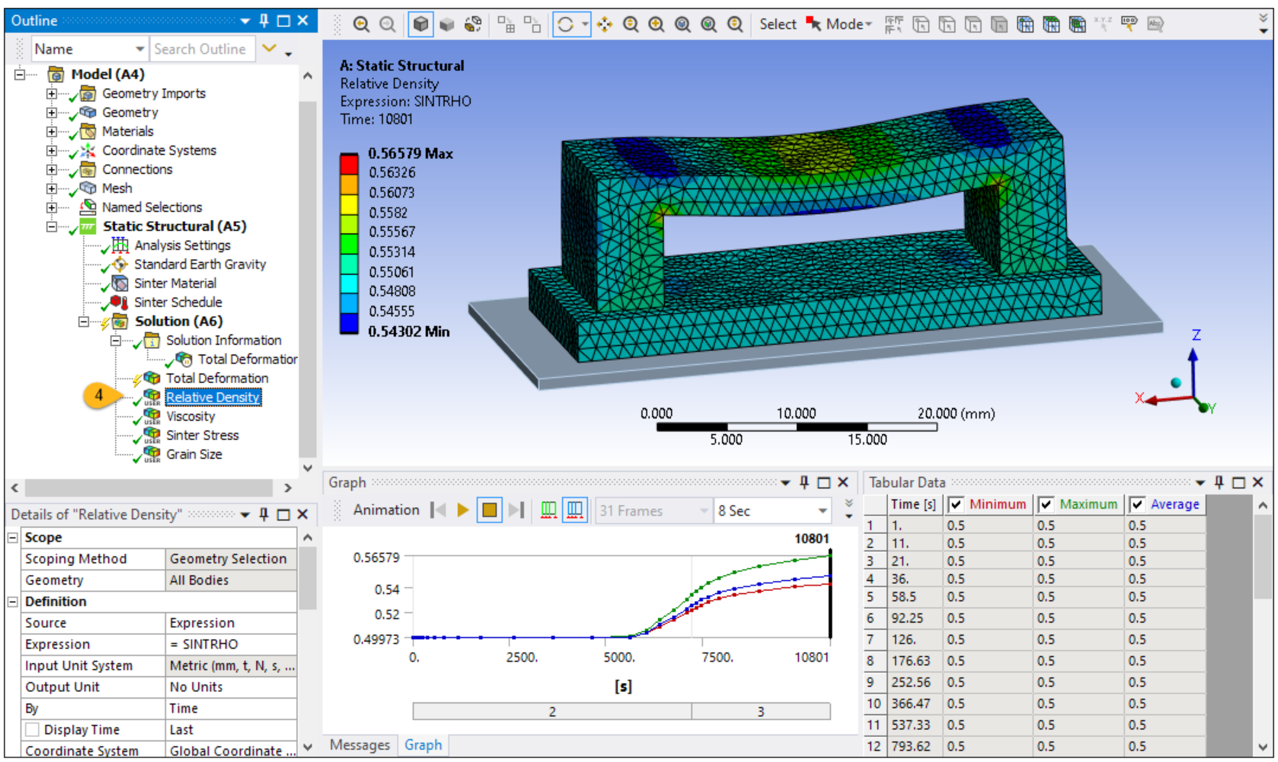
2. Next click **Total Deformation**. The result shows bending in the middle of the bridge span due to gravity because of the viscoplasticity of the material. View the behavior over time in the graph below the graphics window. There is an initial thermal expansion followed by shrinkage once the sintering activation temperature is reached.

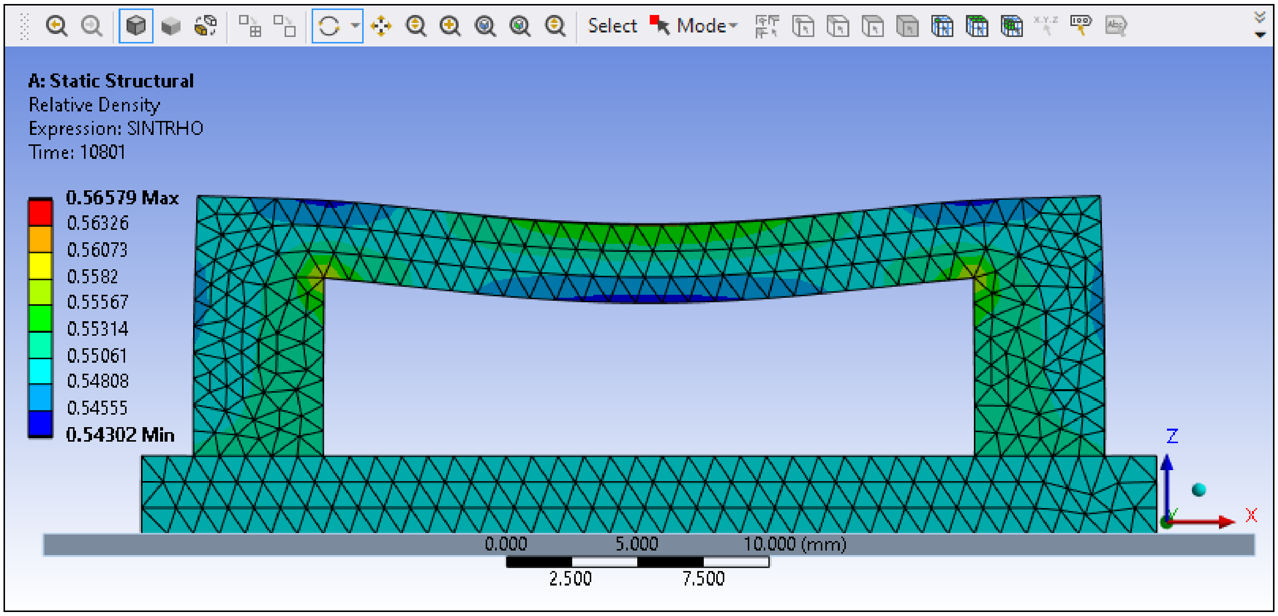


3. Use the animation controls to see an animated display of the deformation, as shown here. *The following is an animated gif. Refresh the page to refresh the animation. The animation is not viewable in PDF format.*

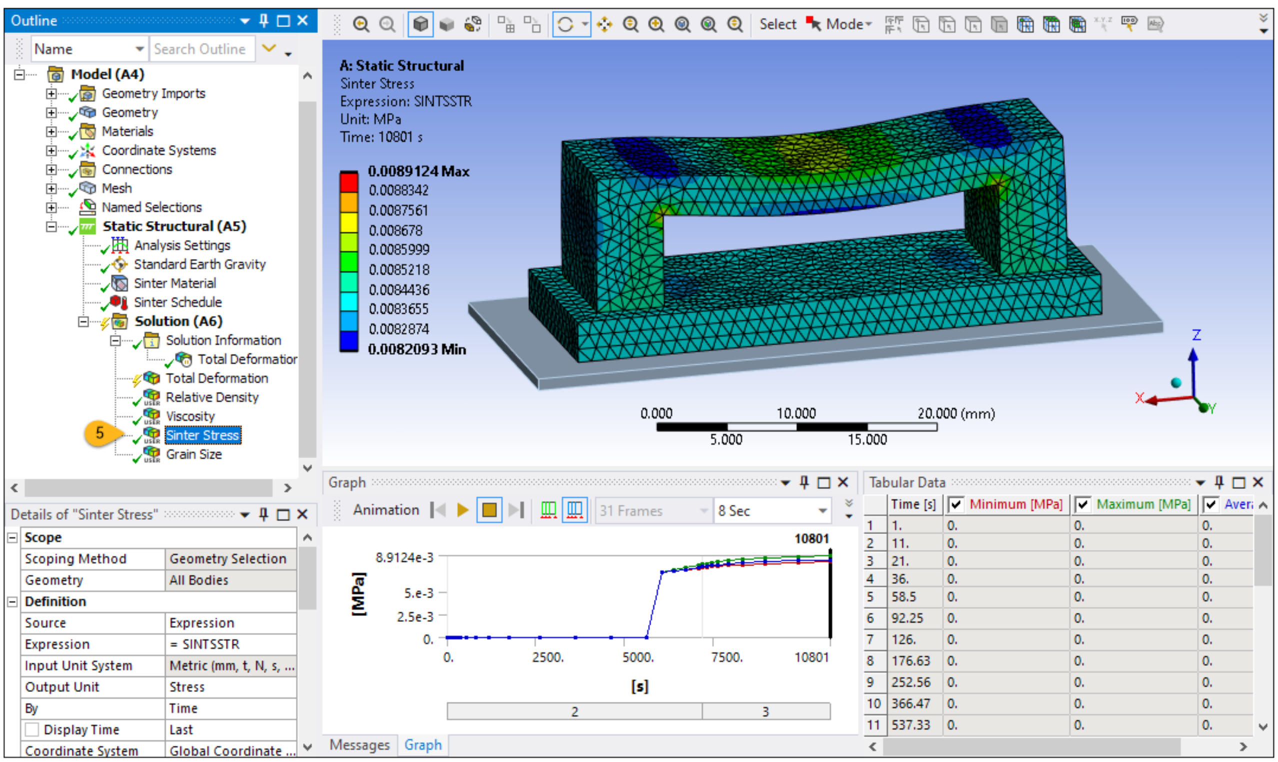


4. Click **Relative Density**. A maximum densification of 0.566, or about 56%, occurs at the top of the bridge span where compression of the material aids densification. The bottom of the span experiences tension and, therefore, less densification.





5. Click **Sinter Stress**. Sinter stress is the driving force that cause shrinkage. Once the temperature reaches the sintering activation temperature, the stress abruptly increases.



Congratulations! You have completed the tutorial.