

Additive Manufacturing Tutorials



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

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

Copyright and Trademark Information

© 2024 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXIm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFID-ENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

What You Need to Know About AM Tutorials	v
1. Calibration - Double Arches Geometry	
1.1. Problem Description	1
1.2. Create the Analysis System	
1.3. Import Geometry and Launch Mechanical	5
1.4. Set Units and Load the LPBF Process Add-on	7
1.5. Wizard Step 1 - Identify Geometries	9
1.6. Wizard Step 2 - Generate Mesh	10
1.7. Wizard Step 3 - Assign Materials	11
1.8. Wizard Step 4 - Define Build Settings	12
1.9. Wizard Step 5 - Define Boundary Conditions and Constraints	13
1.10. Wizard Step 6 - Set Up Calibration	14
1.11. Run Calibration Iterations	15
1.12. Obtain the Optimum Calibration TSSF	16
2. Additive Print Calibration - 4 Pillars Geometry	19
2.1. Problem Description	19
2.2. Import Geometry	21
2.3. Set Up Simulation	22
2.4. Select Geometry and Supports	23
2.5. Configure Materials	25
2.6. Select Outputs and Solve	26
2.7. Obtain the Distortion	27
2.8. Use Calibration Spreadsheet	34
2.9. Run Subsequent Calibration Iterations	35
2.10. Save Calibrated Material	37
3. LPBF Simulation - Bracket	39
3.1. Problem Description	40
3.2. Create the Analysis System	
3.3. Import Geometry and Launch Mechanical	42
3.4. Load the LPBF Process Add-on and Open the LPBF Setup Wizard	
3.5. Complete Wizard Step 1 - Model Setup	
3.6. Complete Wizard Step 2 - Build Settings	
3.7. Complete Wizard Step 3 - Postprocessing Options	
3.8. SOIVE	
3.9. Keview Results	
3.9.1. Thermal Results: Temperature and Hotspot	
3.9.2. Structural Results: Deformation, Recoater Interference, and High Strain	
4. DED SIMUlation - Racetrack	
4.1. Problem Description	
4.2. Create the Analysis System	
4.5. Import Geometry and Launch Mechanical	
4.4. Open the DED Flocess Wizard	
4.5. Wizard Step 7 - Identify Geometries	
4.7 Wizard Step 3 - Set 1 In for Element Clustering and Create Contact Connections	/ / Q1
4.8 Wizard Step 4 - Assian Materials	۱۵ مو
4.9 Wizard Step 5 - Define Ruild Settings and Thermal Roundary Conditions	
4.10 Wizard Step 6 - Define Structural Roundary Conditions and Rase Removal	۰۰۰۰ ۵/ ۵۸
4.11. Perform Element Clustering	
· · · · · · · · · · · · · · · · · · ·	· · · · · · · · · · · · · · · · · · ·

4.12. Solve the Transient Thermal Analysis	
4.13. Review Thermal Results	
4.14. Solve the Static Structural Analysis	
4.15. Review Structural Results	
5. Sintering Simulation - Printed Bridge	
5.1. Problem Description	
5.2. Create Analysis System	
5.3. Attach Geometry and Set Units	
5.4. Load Sintering Process Add-on	
5.5. Wizard Step 1 - Identify Geometries	
5.6. Wizard Step 2 - Define Contact	
5.7. Wizard Step 3 - Define Constraints	
5.8. Wizard Step 4 - Generate Mesh	
5.9. Wizard Step 5 - Define Gravity	
5.10. Wizard Step 6 - Define Sinter Material	
5.11.Wizard Step 7 - Define Sinter Schedule	
5.12. Wizard Step 8 - Define Results and Solver Settings	
5.13. Generate Sinter Schedule and Solve	
5.14. Review Results	

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.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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
- 1.2. Create the Analysis System
- 1.3. Import Geometry and Launch Mechanical
- 1.4. Set Units and Load the LPBF Process Add-on
- 1.5. Wizard Step 1 Identify Geometries
- 1.6. Wizard Step 2 Generate Mesh
- 1.7. Wizard Step 3 Assign Materials
- 1.8. Wizard Step 4 Define Build Settings
- 1.9. Wizard Step 5 Define Boundary Conditions and Constraints
- 1.10. Wizard Step 6 Set Up Calibration
- 1.11. Run Calibration Iterations
- 1.12. Obtain the Optimum Calibration TSSF

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

Ti-6AI-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**.
- 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.

8 🖫 ₹		Context			Syst	ems A,B -	Mechanical [Ansys Mec	hanical Enterpri	se]				-		×
File 2 Ha	ome	Environment	Display	Selection	Automa	ation	Add-ons				Qu	ick Launch		<u>^</u>	1 🕜
Duplicate	ut) opy C aste ဌ	× Delete Q Find ⊈ Tree 2 Core	Computer Distributed s 8	Solve	Resource Prediction	Analysis	▲ Named Selection ☆ Coordinate System ▲ Remote Point Ir	C Commands Comment Comment Sert	Images ▼ Images ▼ Images ▼ Images ▼ Images ▼ Images ■ </th <th>E Tools</th> <th>Layout</th> <th></th> <th></th> <th></th> <th></th>	E Tools	Layout				

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.

8 🗄 ≠		Context			Systems A,B	Mechanical [/	Ansys Med	hanical B	Enterprise]					-		×
File	Home	Environment	Display	Selection	Automati 3	Add-ons						Quick Lau	nch		<u>~</u> 2	1 🕜
D DesignLife	NVH Toolkit	Forced Response	Drop Test	Bolt Statisti Tools Struct	kures Hydrodyna Pressure	mic Offshore	LPBF Process	DED Process	Sintering Process	Distortion Compensation	Variable Load	Left Motion Load Transfer	Add-ons Help			

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.

°	₹	Context			Systems A, E	3 - Mechanical [Ansys Mechanica	l Enterprise]		-		×
File	Home	Environment	Display	Selection	Automation	Add-o. 4	LPBF Process		Quick Launch		^ 🛛	8.
LPBF Hotspot	LPBF Hotspot Correction (t Time LPBF Recoat Beta) Interferenc	er LPBF High e Strain	LPBF Calibrati Wizard	LPBF Setup 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 **b** 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**.

୍ଲି ପ୍ ପ୍ 📦 କ୍ରି 🖫 🕒 🔾 ବ୍ ବି ପ୍ ପ୍ ପ୍ ପ୍ ପ୍ ପ୍ ପ୍ Select 🎽	LPBF Setup Wizard	- ⋕ × / аст
	Define Build Settings	
	Inherent Strain No Build Settings Input Enter Manually	^
	Machine Settings Deposition Thicknes 1 0.06 Hatch Spacing 2 0.12 Scan Speed 3 800 Time Between Layers 10 Dwell Time Multiplier 1 Number of Heat Sources 1	mm mms s
	Build Conditions Preheat Temperature Gas/Powder Temperature Use Preheat Temperature Gas Convection Coefficient TE-05 Powder Convection Coefficient TE-05 Powder Property Factor 0.01	*C W/mm ² W/mm ²
	Cooldown Conditions Room Temperature Gas/Powder Temperature Use Room Temperature Gas Convection Coefficient 1E-05 Powder Convection Coefficient 1E-05 Removal Settings Heat Treat No	°C W/mm ² W/mm ²
	Base Removal Type Off	~

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.

 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.
- 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.

 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.

🚳 Un	saved Project - Workbench					_		×
File	Edit View Tools Unit	ts Extensions	Jobs	Help				
1 na 1	a 🖬 📾							
- -		Ch Chultimi	nation N	5				
: <u> </u>								
🕐 Update 🖉 Clear Generated Data 👔 Refresh 📲 Approve Generated Data 🛛 🐢 Send Study to DPS Project								
Outline	of Schematic C2: Optimization	тфх	Table o	f Schematic C2: C	Optimization		1	▼ ₽ X
		A		A	В	С		
1			1	Name 💌	P1 - Build Settings Thermal Strain Scaling Factor	P2 - Calibration Deformation Maximum (mm)	4	
2	Optimization		2	1 DP 0	1	0.56182	4	
3	Objectives and Constr	raints	3	2 DP 1	0.002	0.0017135	-	
4	Seek P2 = 0.4	49 mm	4	3 DP 2	0.799	0.46742		
5	Domain		5	4 DP 3	0.767	0.4517	J	
6	AM LPBF Them	nal (A1)			\sim			
7	lip P1-Buik	d Settings Therma						
8	Parameter Relation	iships						
9	2 V Raw Optimizat	tion Data			N Contraction of the second se			
10	Convergence	Criteria						
11	Results							
12	Candidate P	Points						
13								
14	✓ NR Samples							
<		>						
Properti	es of Outline A9: Raw Optimiza	atic 🔻 🕂 🗙	Chart: I	No data				▼ ₽ X
	A	В						
1	Property	Value						
2	 General 							
3	Number of Columns	2						
4	Number of Rows	4						
🚦 Rea	dy				🏊 Job Monitor	No DPS Connection 📖 Show Progress 🟓 S	how 0 Mes	sages 🔡

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
- 2.2. Import Geometry
- 2.3. Set Up Simulation
- 2.4. Select Geometry and Supports
- 2.5. Configure Materials
- 2.6. Select Outputs and Solve
- 2.7. Obtain the Distortion
- 2.8. Use Calibration Spreadsheet
- 2.9. Run Subsequent Calibration Iterations
- 2.10. Save Calibrated Material

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-6AI-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.

Ans	ys						
Dashboard	Parts					Import Part	
Parts	Search						
D Build Files	Name		Status	Upload Date	Description	Volume (mm ³)	
*	Gear selector for		Available	5/23/20, 4:21 PM		29,692.34	
Materials	³ /\nsy	S					
	G				and the first state of the first state of the		
	Dashboard	Impor	t Part				
	Parts	Select	Part				
	D Build Files	Choose F		/			
		Valid STL f	ile selected. Part will import on sav	/e.			
	Materials						
		Provide	Part Description				
		Name 4 pillars o	alibration part				
		Tags					
		calibrat	ion × + Tag				
		Description	1				
		4 pillars p	part does not include supports				
		Save	ancel				

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.

<mark>/</mark> \nsy	Ansys								
Dashboard	Draft Simulations	Running Simulations	Completed Simulations						
Parts	Additive Print Assumed Strain AISi10M Scan Pattern Thermal Strain Additive Science Additive Science	No running simulations	Microstructure showing porosity AISi10Mg - 2021 R1 Sim 568 - Success 21 days ago						
Materials	Microstr AISi10M Thermal History Reta Modified: De								
	Gear Selector Fork Modified: Apr 28, 2020								

- 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.

Ansy	/S		
Dashboard	Configure Assumed Uniform Strain Simulation		^
@ Parts	Details		
D Build Files	Simulation Title * 4 Pillars calibration - Assumed Strain, Linear Elastic, Ti64		
Materials	Tags		
	Calibration × + Tag		
	Tag your project for efficient filtering, grouping, and reporting Description		
	Calibration simulation to determine SSF for Assumed Strain, Linear Elastic simulation with Ti64 material - 1st iteration	li li	
	Number of Cores		
	8	T	

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**.

Ansys	S			
Dashboard	Geometry Selection			•
@ Parts	Parts Build F	Files		
D Build Files	Available Parts			
=±	Search			
Materials	Name	Status	Upload Date	
	4 pillars calibration part	Available	12/30/20, 1:40 PM	Add
	Gear selector fork	Available	5/23/20, 4:21 PM	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.

Ansys		
Dashboard	Geometry Selection	4
e Parts	Parts Build Files	
D Build Files	4 pillars calibration part Change	
Materials	Status: Available Size (x, y, z): 20.00 mm, 20.00 mm, 25.00 mm	
	Voxel Size (mm)	
	Sestimated Memory Usage: 0.28 GB	
	Voxel Sample Rate (sample rate ³)	
	5	

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

<mark>/</mark> \nsy	S	
Dashboard	Supports	*
Parts	Simulate With Supports	
D	Support Type	
Build Files	Automatic	
Materials	Minimum Overhang Angle (°)* 👔	
	45	
	Minimum Support Height (mm) *	
	0	
	Support Yield Strength Ratio * 👔	
	0.4375	

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.

/\nsy	S
Dashboard	Material Configuration
Parts	Material * Ti64
Build Files	Stress Mode Linear Elastic
Materials	Elastic Modulus (GPa) *
	110
	Poisson Ratio *
	0.3
	Yield Strength (MPa) *
	1100
	Strain Scaling Factor * 1

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.

/\nsy	/s	
Dashboard	Outputs	*
e Parts	On-plate residual stress/distortion 1	
D Build Filos		
Materials	Eayer by Tayler Suessidiation for T Files for transfer to Ansys Mechanical® Detect potential blade crash due to distortion	
	High strain areas	
	Save Start Duplicate Export Delete	

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.

<mark>/\</mark> nsy	S									
ashboard							Exp	port Simulation	Cancel D	uplicate
@ Parts	Overv	iew	1	Part			Out	tputs Selec	cted	
				Name	4 pillars c	alibration part	On-pl	late residual stress	distortion	\checkmark
ild Files	ID	0/0		File Name	4_pillars.s	tl	Disto	rtion compensated	STL file	×
	Status	In Progress		Description	4 pillars p include su	art does not ipports				~
	Title	4 Pillars calit ain, Linear E	pration - Assumed Str lastic, Ti64	Tags	calibration	1	Supp	orts residual stress	alstortion	×
	Description	Calibration s	imulation to determin	Volume	5,100.0 (r	nm ³)	Displ	acement after cuto	ff	\times
		e SSF for As Elastic simul al - 1st iterati	ssumed Strain, Linear Ilation with Ti64 materi Ition	Size (x,y,z)	20.00 mm mm	, 20.00 mm, 25.00	Disto (after	rtion compensated cutoff)	STL file	×
	Tags	Calibration		Triangle Count	84		Layer	r by layer stress/dis	tortion	×
	Requested	a few second 1/7/21, 1:55	is ago PM	Voxel Size	0.5 mm		Ansy	s mechanical files		×
	Simulation Type	Assumed Un	iform Strain	Voxel Sample Rate	5		Detec	ct potential blade ci rtion	ash due to	×
	Number of Cores	8					High	strain areas		×
				Materia	I					~
	Activit	y Statu	s	Material		Ti64	Out	tput Files		
	Voxelization – Finished Mechanics Solver – Running			Stress Mode		Linear Elastic				
				Yield Strengt	h	1,100 MPa	Solve	er Voxel Input	Expo	ort
	VTK to AVZ	Conversion -	Requested	Elastic Modu	lus	110 GPa	Posit	ioned Part	Expo	ort
				Poisson Rati	•	0.3				
	Logs		Strain Scalin	g Factor	1					
	Da	te v	Message	Suppor	ts					
	1/7/2	1, 1:56 PM	Starting element layer 36	Simulate Wit	h Supports	×				
	1/7/2	1, 1:56 PM	Starting element layer 35							

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.

Ansy	S							
ashboard						Export Simulation	Save Logs Restart Duplicate	Delete
ë Parts	Overvi	ew	1	Part			Outputs Selected	
	0 incode tie m			Name	4 pillars ca	alibration part	On-plate residual stress/distortion	1 🗸
ild Files	ID	5/5		File Name	4_pillars.s	ti	Distortion compensated STL file	×
	Status	Success		Description	4 pillars p include su	art does not pports		~
terials	Title	4 Pillars calibi ain, Linear Ela	ration - Assumed Str astic, Ti64	Tags	calibratior	1	Supports residual stress/distortio	n X
	Description	Calibration sir	mulation to determin	Volume	5,100.0 (n	nm ³)	Displacement after cutoff	×
		e SSF for Assumed Strain, Linear Elastic simulation with Ti64 materi al - 1st iteration		Size (x,y,z)	20.00 mm mm	, 20.00 mm, 25.00	Distortion compensated STL file (after cutoff)	×
	Tags	Calibration		Triangle Count	84		Layer by layer stress/distortion	×
	Requested	a few second: 1/7/21, 1:55 F	s ago PM	Voxel Size	0.5 mm		Ansys mechanical files	×
	Simulation Type	Assumed Uni	form Strain	Voxel Sample Rate	5		Detect potential blade crash due t distortion	° ×
	Number of Cores	8		Material			High strain areas	×
	Activity	/ Status	6	Material		Ti64	Output Files	
	Voxelization - Einished			Stress Mode		Linear Elastic		
	Mechanics Solver – Finished		Yield Strength	I	1,100 MPa	Solver Voxel Input	Export	
	VIK to AV2 C	Sonversion - Finished		Elastic Modul	us	110 GPa	Positioned Part	Export
	Logs			Poisson Ratio		0.3	On plate strass/displacement	Export
	0			Strain Scaling	Factor	1	Solver Voxel Input	Export
	Date	~	Message				(avz)	CAPUIT
	1/7/21, 1:57 PM Simu		Simulation 575	Support	S		On plate View stress/displacement (avz)	Export
	1/7/21	, 1:57 PM	completed. Activity vtk-to-	Simulate With	Supports	×		

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.
| Ansys | 5 | | | | | | | | |
|-------------------|--------------------|--|--|----------------------|--------------------------|-------------------------|---|----------------|--------|
| Dashboard | | | | | | Export Simulation | Save Logs Restart | Duplicate | Delete |
| @
Parts | Overv | iew | 1 | Part | | | Outputs Sel | ected | |
| D | Cimulation | 575 | | Name | 4 pillars o | alibration part | On-plate residual stre | ess/distortion | ~ |
| Build Files | ID | 575 | | File Name | 4_pillars. | stl | Distortion compensa | ted STL file | × |
| Materials | Status | Success | | Description | 4 pillars p
include s | art does not
upports | Supports residual str | ress/distortio | |
| | Title | 4 Pillars calibr
ain, Linear Ela | 4 Pillars calibration - Assumed Str
ain, Linear Elastic, Ti64 | | calibratio | n | oupporto residual sa | 000/0101010101 | · X |
| | Description | Calibration sin | nulation to determin | Volume | 5,100.0 (| mm ³) | Displacement after c | utoff | × |
| | | e SSF for Ass
Elastic simula
al - 1st iteratio | umed Strain, Linear
tion with Ti64 materi
n | Size (x,y,z) | 20.00 mn
mm | n, 20.00 mm, 25.00 | Distortion compensa
(after cutoff) | ted STL file | × |
| | Tags | Calibration | | Triangle
Count | 84 | | Layer by layer stress | /distortion | × |
| | Requested | a few seconds
1/7/21, 1:55 P | s ago
M | Voxel Size | 0.5 mm | | Ansys mechanical fil | es | × |
| | Simulation
Type | Assumed Unif | orm Strain | Voxel
Sample Rate | 5 | | Detect potential blad | e crash due te | ° × |
| | Number of
Cores | 8 | | | | | High strain areas | | ~ |
| | | | | Materia | ıl | | | | ^ |
| | Activit | y Status | ; | Material | | Ti64 | Output Files | ; | |
| | Voxelization | - Finished | | Stress Mode | • | Linear Elastic | a har been been to be a first | | |
| | Mechanics S | Solver – Finishe | ed
Finished | Yield Streng | th | 1,100 MPa | Solver voxel input | | Export |
| | VIR to AV2 | Conversion - | Inisited | Elastic Mode | ulus | 110 GPa | Positioned Part | | Export |
| | Loas | | | Poisson Rat | io | 0.3 | On plate | | Export |
| - | =-9- | | | Strain Scalir | ng Factor | 1 | Stress/displacement | | |
| | Dat | ie ~ | Message | | | | (avz) | View | Export |
| | 1/7/2 | 1, 1:57 PM | Simulation 575 | Suppor | ts | | On plate
stress/displacement
(avz) | View | Export |
| | | completed. | | Simulate Wi | th Supports | × | | | |
| | 1/7/2 | 1, 1:57 PM | Activity vtk-to-
avz completed. | Canada M | | X | | | |

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

EnSight Standard 202	21 R1 :: (ples\AP Tutorial Models\4_pillars\sim575-on-plate-stress-displacement.vtk)			- 🗆 X
File Edit Create Qu	uery View Tools Window Case Reports Help			
🇱 🚍 📈 🎖	🖗 🔲 A 🛄 🎬 👑 🔎 🎸 🗲 🧃 🕷	i 👁 🍮 🚍 🕄 🎕 🛃 🗛 🚧 🚣 🌅	۶. 🖬 🕣 🖇	
Parts	8 ×			ANSYS
Name	Id Show Color Color by			2021 R1
 Case 1 sim575-on-plate-stre 	ess-displacement 1			
	Edit			
	Create >			
	Hide			
	Delete			
	Clone			
	Color by Select variable			
	Style			
	Part select Black			
	Clips > White			
	Contour Grey			
	Isosurface Random			
	Vector arrows Matching partnames in other of	case(s)		
	Show normals Make transparent			
	Particle trace Adjust transparency			
	Save to report			
Plots/queries	Load part & X			
Name Show X varia	iable Y Add group/move to group			
Queries				
Plotters	🛤 Choose a variable 🛛 🕹			
	Select a variable to use			
	Variable:			
	Name			
	Coordinates[] Coordinates[X]			
	Coordinates[Y]			
	Coordinates[Z]			
	> Scalars			
	✓ Vectors	Y .		
	disp mm []	4		
	disp_mm_[Y]			
	disp_mm_[Z]			
		×		
	Sort: 🔾 Alphabetically 🕐 Type 🔍 Tree			
	Show vector components			
	OK Cancel			
Variables Annotations	Plots/oueries Viewports States			
1		** 🔲 上 🖉 🖿 🔐	A ≤ ₹ A A.	
Constanting and	S 🖻 🖉 🖸	K → X Ø ₽° ØG	NA 2 R. 7	
Upen color dialog				

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.

EN Create/ed	EN Create/edit probe query							
Probe create	Display style							
Display ID								
Display expansio	on factor 0 💭 **** Will only expand if the picked part *** contains element labels.							
	_							
✓ Visible	Always on top							
R 0.00	G 0.00 B 0.00 Mix							
Format Floatin	ng point 🗸 Decimal places 🛛 🖨							
Marker								
Visible	Size 8.000000							
R 0.50	G 0.50 B 0.50 Mix							
	Create Close H	Help						

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.

	А	В	С	D	E	F	G	Н	1	J	K
1			SSF Cali	bration for	Assumed Strai						
3					Distortion (mm)]				
4			Geometr	/ Nominal		Extract distortion va	lue at the location of				
5			Measu	rement	0.242	models built with th	stortion				
6									-		
7		U	Simulation	Simulation		Enne r0(
8		asti	iteration	number	Distortion (mm)	SSF	SSF	Error%	Sim di	stortion	
9		ar El	1st	575	0.169	1.000	1.432	30.2%			
10	2nd			1.432	•		New S calibra	SF for ne tion iter	ext ation		
11			3rd								
12					·	·					
13			Simulation	Simulation	Distortion (mm)	Simulation settings	New settings	Error%			
14			iteration	number		SSF	SSF	E115115	-		
15		Ę	1st			1.000			4		
16		tic	2nd						4		
17		las	3rd						4		
18		<u>ь</u>	4th						-		
19		9	5th						-		
20			6th						4		
21			7th						1		
22 23											
	(• c	over Page	Results Summa	ry Calibration	for AS Calibration	for SP Fine tune	for SP Calibra	ation for TS	Fine tune	for TS

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 2^{nd} iteration with SSF = 1.432, which yields a 0.0% error.



	Constant	N				
	Geometry	Nominal		Extract distortion va	lue at the location of i	interest from
	Measu	rement	0.242	models built with th	ird scan pattern (rotat	ting stripe)
J	Simulation	Simulation	Distortion (mm)	Simulation settings	New settings	Error%
asti	iteration	number	Distortion (mm)	SSF	SSF	EITOT76
ar El	1st	575	0.169	1.000	1.432	30.2%
inea	2nd	576	0.242	1.432	1.432	0.0%
_	3rd			1.432		

SSF Calibration for Assumed Strain Simulations

Distortion (mm)

	Simulation	Simulation	Distortion (mm)	Simulation settings	New settings	Error®/
	iteration	number	Distortion (mm)	SSF	SSF	EITOT%
tγ	1st			1.000		
tici	2nd					
ast	3rd					
P	4th					
2	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-6AI-4V.

			,					
Material	Stress Mode	Assumed Strain		Scan P	attern	Thermal Strain		
			SSF		SSF			
	Lineas Floatio	CCT	1.432	ASC		ASC		
	Linear Elastic	55F		ASC 1		ASC 1		
24.61				ASC_z	1.000	ASC_z	1.000	
3101				SSF		SSF		
	12 Diacticity	SSF		ASC		ASC		
	JZ Plasticity			ASC 1		ASC 1		
				ASC_z	1.000	ASC_z	1.000	

Summary of Results

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
- 3.2. Create the Analysis System
- 3.3. Import Geometry and Launch Mechanical
- 3.4. Load the LPBF Process Add-on and Open the LPBF Setup Wizard

3.5. Complete Wizard Step 1 - Model Setup3.6. Complete Wizard Step 2 - Build Settings3.7. Complete Wizard Step 3 - Postprocessing Options

3.8. Solve

3.9. Review Results

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.

8 🗄 ≈		Context		Systems A, B - Mechanical [Ansys Mechanical Enterprise]							
File	Home	Mesh Dis	play Selectio	n Autom	Add-ons L	earning and Suppor	t LPB	F Process			
DesignLife		Forced Response	Drop Test Too	t Statistics on s Structures	Hydrodynamic Offsh Pressure	ore Aqwa Co-simulation	LPBF Process	DED Sinter Process Proc	ing Distortion	Variable Motion Load	(?) Add-ons Help
Fatigue	NVH	Turbomachinery	Explicit Me	chanical Toolkit	Hydrodyna	mic Loads		Additive Man	ufacturing	Rigid Dynamics	Support

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

8° 🗄	~	Context				Sys	ems A,B - Mechanical [/	Ansys Mechanical Enterprise]
File	Home	Mesh	Display S	election Auto	mation Add-	ons Learning and Sup	2 LPBF Process	
					Ξž			
LPBF Hotspot	LPBF Hotspoi Correction (t Time LPBF (Beta) Into	Recoater LPBF H erference Stra	ligh LPBF Calibrat in Wizard	ion LPBF Setup 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.

- 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.

Outline ::::::::::::::::::::::::::::::::::::	→ ‡ □ ×								
Name	🛛 Search Outline 💙 🖕								
Project*									
🗄 🖷 🐻 <u>Model (A4</u>	🗄 🔤 Model (A4, B4)								
🗄 🛶 🏹 Geome	📺 🛶 🙀 Geometry Imports								
🗄 🗸 🖓 Geome	⊡√∰ Geometry								
🕂 🗤 🗸 🔂 Materi									
🗄 🗸 🎇 Coordi	Coordinate Systems								
E									
□ □ □ · · · √ 🕅 (⊡√ Contacts								
	P Build To Base								
Mesh	M Bond - bracket (Part) Patch body21110 Support								
	Part Lavered Tetrabedral								
	ase Sizing								
AM Pro	ocess								
	Build Settings								
🖻 🖉 🖻 S	Support Group								
	🐖 STL Support								
🖻 \cdots 😫 Named	Selections								
	ART								
	NASE Wild Contact Flomont Eacon								
	ase Contact Element Faces								
	ient Thermal (A5)								
│	Analysis Settings								
	Solution (A6)								
	🕤 Solution Information								
⊡? <mark>‴</mark> Static	Structural (B5)								
· · · · · · · · · · · · · · · · · · ·	Analysis Settings								
	mported Load (A6)								
E	Solution (B6)								
/	Solution Information								
Details of "Mesh"	→ ‡ □ ×								
Display									
Display Style	Use Geometry Setting								
 Defaults 									
Physics Preference	Mechanical								
Element Order	Program Controlled								
Element Size	1.25 mm								
+ Sizing	Sizing								
Quality									
Inflation									
Batch Connections									
+ Advanced									
Statistics									

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_AlSilOMg.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.

LPBF Set	up Wizard	۸nsys	• Ф S / Аст
Model Setup	 Simulation Settings 		
Build Settings	LPBF Simulation Type T Calibration Settings	hermal - Structural	
Postprocessing Options	Thermal Strain Scaling Fa	ctor 1	
- F	Hatch Spacing	0.1	mm
Apply Changes	Deposition Thickness Scan Speed	0.05	mm mm/s
2 Read from Tree	Dwell Time Dwell Time Multiplier	10	S
Advanced Options	Number of Heat Sources Build Conditions	1	
Help	Preheat Temperature 10	0	°C
	Base Face Selection 1 F	ace	Apply
	Base Structural Bour Base Face Selection 1 F	dary Condition (Fixed Support)	Apply
Exit Wizard	Back Next		

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.

LPBF Setu	up Wizard	Ansys	AC
Model Setup	 Simulation Settings LPBF Simulation Type 	hermal - Structural	
Build Settings	 Calibration Settings 		
Postprocessing	Thermal Strain Scaling Fac	ctor 1	
Options	 Machine Settings Hatch Spacing 	0.175	mm
3 Apply Changes	Deposition Thickness Scan Speed	0.03	mm mm/s
Read from Tree	Dwell Time Dwell Time Multiplier	10	S
Advanced Options	Number of Heat Sources Build Conditions 	1	
Halo	Preheat Temperature 35		°C
Help	Base Thermal Bound	ary Conditions	
	Base Face Selection 1 F	ace	Apply
	Base Structural Bound	dary Condition (Fixed Support)	
	Base Face Selection 1 F	ace	Apply

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.
- 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.

🚰 🛄 🔻 Context						Systems A, B
File Home AM Process [Display Selection	Automation	Add-ons Lear	ning and Support	LPBF Process	
Duplicate Q Solvers	amed Selection Com pordinate System Com emote Point L Cha Insert	imands limages iment limages rt limages Annota	Plane tion AM Proces	Support Predefine Group Support Su	ed Generated	STL Crea Bas
Name Search Outline		s Sequence		r		
Project*	Transien	t Thermal	*	R	eset All	
Model (A4, B4)	Transie	nt Thermal		Static Structura	al	
Emery Geometry	Build St	ер		Build Step		
⊞,⁄ ™ Materials ⊞,⁄ ¼ Coordinate Systems	Build			Build		
	Cooldo	wn Step	×	Cooldown Step		
AM Process	Cooldov	vn		Cooldown		
Eulid Settings				Removal Step		×
⊡ Selections Image:				Base Unbolt		
				Removal Step		×
				Base		
				Removal Step		×
				STL Support		
				<u> </u>		
	Add Step)	-	Add Step		-

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 \mathbf{x} in the upper right corner to close the wizard panel.

LPBF Set	up Wizard	/\ns	ys / ac
Model Setup	 Postprocessing 	Steps	
Build Settings	Base Unbolting		
Dana Cottingo	Base Removal		
Postprocessing	Support Remova	I	
Options	Base Removal		
	Base Removal Type		
7 Apply Changes		Directional	
	Removal Step Size	10	mm
8 Add Results	Removal Direction	90	•
Dood from Trop	LPBF Results		
Read from free	🗹 Hot Spot		
Help	Recoater Interfer	ence	
	✓ High Strain		
	 Hot Spot Result 		
	Threshold Definition	○ None	
		User Defined	
	Warning Threshold	135	°C
	Critical Inreshold	235	
	 Recoater Interfe 	rence Result	
	Inreshold Definition	O None	
		Layer Thickness Based	
	Powder Packing Day	User Defined	
		.u.	
	 High Strain Residues 		
		None	
	Warning Threshold	User Defined	mm/mm
	Critical Threshold	0.2	mm/mm



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.

of 🗄 🗸	;	Context					Sj	ystems A, B - N	lechanical [Ans	ys Mechanical Ente	erprise]	
F 1	Home	Solution	n Display	Selection	Automation	Ado	d-ons	Learning an	d Support	LPBF Process		
	- Cut	× Delete	My Computer	- 4	<u></u>		Name	d Selection	Commands	Images ▼	E	
Duplicate	E Copy	Q Find	Cores 10	Solve	Resource	Analysis	米 Coord	inate System	다그 Comment	Annotation	Tools	Layout
•	Outline			Solve	Frediction 5	*	- griener	In	sert		•	•

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.

8 🖽 ≠	Context				Sy	stems A,B - Mech	anical [Ansys N	lechanical Ent
File Home	Solution Information	Display	Selection	Automation	Add-ons	Learning and S	upport LF	BF Process
Duplicate	X Delete My Computer Q Find ✓ Distributed Tag Tree ▼ Cores	4 Solve	Resource Prediction	Analysis	Named Selection Coordinate Systen Remote Point	Commands Comment		ane un Units
Outline Name	Search Outline 🗸 🗸	· ↓ [Solve	e (F5) Z Solve th	e simulation		<u>፦</u> ତ୍ ତ୍ ଭ୍	🔍 🤤 Selec	t 🦎 Moder
Project*	B4) ry Imports ry	7 ① F	vising th solve ha	e selected ndler. elp.				

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.

° 🗔	Ŧ	Context					Systems A, B - N	dechanical [Ansys Mech	hanical E
File	Home	Result	Display	Selection	Automatio	n Add-ons	Learning and Support	LPBF Process	
	×		Named 🖓	Selection	Commands	Images 7 7 Section Plane	1.0 (True Scale)	• =	
Duplicate	Q Solv	e Analys	is 🖉 Remote	Point	Liii Chart	Annotation	✓ Large Vertex Contours	Geometry Contours	Edges
Outline	e Solv	era		In	sert			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
- 3.9.2. Structural Results: Deformation, Recoater Interference, and High Strain

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**.

Graph	- ₽ □ ×	Tab	ular Data 🤉					×
Animation 🖌 🕨 🔲 🔛 📖 20 Frames 🗸	2 Sec (Auto) 👻 🖼 🎟 🔍 🗰 🦟 🎽		Time [s]	Minimum [°C	Maximum [°C]	Average [°C]	Identifier	^
		64	2148.6	96.767	100.	98.477	Build Step	
2149.1		65	2148.6	97.264	570.	159.83	Build Step	
5 ^{70.39}		66	2110.0	07.070	570.	166.39	Build Step	
I 🕮 NNNNNNNNNNNN 🗤 🗸 Reti	rieve This Result	67	2 Co	opy Cell	06.02	102.77	Build Step	
26.67		2	Re	etrieve This Result	00.	98.395	Build Step	
Sele	Retrieve This Result 182.6	69	2		70	168.29	Build Step	
[-]		70	2 Cr	eate Results	etrieve This Resul	t 74.58	Build Step	
[5]		71	2 Ex	port	05.00	02.77	Build Step	
Build Step	Cooldo	72	2		00.	98.26	Build Step	
		73	2 Se	ect All	70.	168.06	Build Step	
Graph Messages		74	2431.1	97.03	570.	175.5	Build Step	×
	😐 6 M	lessag	es Se	elect All	nm, kg, N, s, m\	, mA) Degrees	rad/s Celsius	.d

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
- 4.2. Create the Analysis System
- 4.3. Import Geometry and Launch Mechanical
- 4.4. Open the DED Process Wizard
- 4.5. Wizard Step 1 Identify Geometries
- 4.6. Wizard Step 2 Generate Mesh
- 4.7. Wizard Step 3 Set Up for Element Clustering and Create Contact Connections
- 4.8. Wizard Step 4 Assign Materials
- 4.9. Wizard Step 5 Define Build Settings and Thermal Boundary Conditions
- 4.10. Wizard Step 6 Define Structural Boundary Conditions and Base Removal
- 4.11. Perform Element Clustering
- 4.12. Solve the Transient Thermal Analysis

- 4.13. Review Thermal Results
- 4.14. Solve the Static Structural Analysis
- 4.15. Review Structural Results

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.

0)utline	▼ ‡ □ ×	(Dutline	▼ ‡ □ ×
	Name - Search	n Outline 🗸 🗸		Name	▼ Search Outline V
[Project* Model (A4, B4) Model (A4, B4) Geometry Impo Geometry Materials Coordinate Syst Connections Mesh Mesh Mesh Materials Mesh Mesh Materials Mesh Mesh Materials Mesh	rts tems ed Cartesian g ns v		Project* Model (A4 	4, B4) eetry Imports eetry ials dinate Systems ections Body Fitted Cartesian Body Sizing d Selections
C	etails of "Body Fitted Cartes	ian" - Method 👓 🔻 🖡 🗖 🗙	1	Details of "Body Sizi	ng" - Sizing 👻 🕈 🗖 🗙
E	Scope		E	Scope	
	Scoping Method	Named Selection		Scoping Method	Named Selection
	Named Selection	print_part		Named Selection	base_plate
E	Definition		E	Definition	
	Suppressed	No		Suppressed	No
	Method	Cartesian		Туре	Element Size
	Element Order	Use Global Setting		Element Size	4.0 mm
	Туре	Element Size	E	Advanced	
	Element Size	3.0 mm		Defeature Size	Default
	Spacing Option	Uniform		Behavior	Soft
Ε	Advanced		∥∟		
	Projection Factor	0			
	Project in constant Z-Plane	No			
	Stretch Factor in X	1.0			
	Stretch Factor in Y	1.0			
	Stretch Factor in Z	1.0			
	Coordinate System	Global Coordinate System			
	Write ICEM CFD Files	No	1		
	Smooth Mesh Spacing	No	1		

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 🗆 🗙 • 4 × Outline 🤁 🕘 📦 🏶 🖺 🕒 🔿 🔸 🤁 🥹 🍭 🍭 🤤 Select Wizard š Name 🝷 Search Outline 🛛 🖌 DED Proces /\nsys / ACT Project* 🛅 Model (A4, B4) È Geometry Imports € Geometry ⊕ √ Materials Input Source G-Code Clus 1 E Coordinate Systems Choose G-Code File E:\Simulations\ 2 Edit Connections ÷ Cluster Volume 1000 mm³ E Mesh A Body Fitted Cartesian ~___ Print Direction Ζ -🗸 🕼 Body Sizing Build to Base Conta... • Named Selections Yes Contact Generation ... Auto (planar t Ŧ 🗸 🛅 base_plate 🜆 Transient Thermal (A5) √T=0 Initial Temperature Analysis Settings Solution (A6) Static Structural (B5) ✓ ∰ Analysis Settings ė..... Solution (B6) 🌾 👘 Solution Information Help Select the input source for the welding path Details of "Mesh" • 4 □ × definition. Display Input Source: Display Style Use Geometry Setting Manual Clustering: Use predefined Defaults Named Selections as weld path order. Physics Preference Mechanical Make sure the naming convention is Element Order Linear correct. The elements of the first weld Element Size Default track must be named 'weld_1'. The + Sizing start face of the first weld track must Quality be named 'start_face_1'. For the second weld track the naming + Inflation + Batch Connections convention continues with 'weld_2' for + Advanced the elements and 'start_face_2' for the Statistics start face, and so on. For easier 40.00 Exit Wizard Ba 4 1 Message No Selection Metric (mm, t, N, s, mV, mA) Degrees rad/s Cels Ready
- 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 concentration		
Name	▼ Search Outline ∨ .	-
Project* ■ Model (A ■ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓	A4, B4) metry Imports metry erials dinate Systems mections ed Selections Process for DED Build Settings G-Code Clustering	
⊕ – <mark>⁄ №</mark> Trai ⊕ – <mark>⁄ </mark> ‴ Stat	Cluster Settings nsient Thermal (A5) tic Structural (B5)	
	Cluster Settings nsient Thermal (A5) tic Structural (B5) cess for DED"	···· → ₽ □ ×
Details of "AM Prod Part Geometry	Cluster Settings nsient Thermal (A5) tic Structural (B5) cess for DED"	→ ₽ 🗆 ×
Details of "AM Proc Scoping Method	Cluster Settings nsient Thermal (A5) tic Structural (B5) cess for DED" Named Selection	···· ▼ ₽ ⊡ ×
Details of "AM Prod Scoping Method Named Selection	Cluster Settings Insient Thermal (A5) Lic Structural (B5) Cess for DED Named Selection print_part	···· ∓ ₽ ⊡ ×
Details of "AM Proc Part Geometry Scoping Method Named Selection Base Plate Geom	Cluster Settings insient Thermal (A5) tic Structural (B5) cess for DED Named Selection print_part etry	▼ ₽ □ ×
Details of "AM Proc Part Geometry Scoping Method Named Selection Base Plate Geom Scoping Method	Cluster Settings Insient Thermal (A5) Lic Structural (B5) Cess for DED Named Selection print_part etry Named Selection	
 Details of "AM Prod Part Geometry Scoping Method Named Selection Base Plate Geom Scoping Method Named Selection 	Cluster Settings Insient Thermal (A5) Control (A5) Contro	▼ ¶ □ ×
 Details of "AM Prod Part Geometry Scoping Method Named Selection Base Plate Geom Scoping Method Named Selection Removal Step 	Cluster Settings insient Thermal (A5) tic Structural (B5) cess for DED" Named Selection print_part etry Named Selection base_plate	···· ₽ □ ×
 Training Training Training	Cluster Settings nsient Thermal (A5) tic Structural (B5) Cess for DED Named Selection print_part etry Named Selection base_plate No	
Details of "AM Prod Part Geometry Scoping Method Named Selection Base Plate Geom Scoping Method Named Selection Removal Step Removal Step Material	Cluster Settings Insient Thermal (A5) Constructural (B5) Constructural (B5) Construction Constru	···· → ₽ □ ×
 Details of "AM Prod Part Geometry Scoping Method Named Selection Base Plate Geom Scoping Method Named Selection Removal Step Material Source 	Cluster Settings insient Thermal (A5) tic Structural (B5) Cess for DED Named Selection print_part etry Named Selection base_plate No Engineering Data	···· ▼
 Training Training Training	Cluster Settings Insient Thermal (A5) Itic Structural (B5) Cess for DED Named Selection print_part etry Named Selection base_plate No Engineering Data	···· → ₽ □ ×

D	Details of "G-Code Clustering" 🗢 🕂 🗖 🗙				
⊡	Import G-Code file				
	G-Code File	DED_Racetrack_G-Code.txt			
Ξ	Cluster Volume				
	Cluster Volume	1000 mm ³			
Ξ	Visualization	·			
	Number of Layers	0			
	Show Every n-th Layer	1			
	Line Segments For Arc	5			
	Show Positioning Lines	On			
Ξ	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			
Ξ	Definition				
	Read Feed Rate	No			
	Read Preheat Temperature	No			
	Read Dwell Time	No			
Ξ	G-Code Transformation				
	Translate X	0 mm			
	Translate Y	0 mm			
	Translate Z	0 mm			
	Rotate X	0 °			
	Rotate Y	0 °			
	Rotate Z	0 °			
Ξ	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.

Name Search Outline V Project* Model (A4, B4) Geometry Imports Model Competity
Project* ⊡ Model (A4, B4) ⊡ √ B Geometry Imports
Materials Coordinate Systems Connections Contacts Connection Group Connection Group DED_Contact Mesh Named Selections print_part base_plate Base plate upper element faces Print part lower element faces Print part lower element faces Transient Thermal (A5) Static Structural (B5)
Details of "DED Contact"
- Scope
Scoping Method Named Selection
Contact Print part lower element faces
Target Base plate upper element faces
Contact Bodies DED_Racetrack\PrintPart
Target Bodies DED_Racetrack\BasePlate
Protected No
Definition
Type Bonded
Scope Mode Manual
Behavior Asymmetric
Trim Contact Program Controlled
Trim Contact Program Controlled Suppressed No
Trim Contact Program Controlled Suppressed No Object ID (Beta) 71
Trim Contact Program Controlled Suppressed No Object ID (Beta) 71 Advanced Program Controlled
Trim Contact Program Controlled Suppressed No Object ID (Beta) 71 Advanced Formulation Program Controlled
Trim Contact Program Controlled Suppressed No Object ID (Beta) 71 Advanced Formulation Formulation Program Controlled Small Sliding Program Controlled
Trim Contact Program Controlled Suppressed No Object ID (Beta) 71 Advanced Formulation Formulation Program Controlled Small Sliding Program Controlled Detection Method Nodal-Normal From Contact
Trim Contact Program Controlled Suppressed No Object ID (Beta) 71 Advanced Formulation Formulation Program Controlled Small Sliding Program Controlled Detection Method Nodal-Normal From Contact Penetration Tolerance Program Controlled
Trim Contact Program Controlled Suppressed No Object ID (Beta) 71 Advanced Formulation Formulation Program Controlled Small Sliding Program Controlled Detection Method Nodal-Normal From Contact Penetration Tolerance Program Controlled Elastic Slip Tolerance Program Controlled Normal Stiffness Program Controlled
Trim Contact Program Controlled Suppressed No Object ID (Beta) 71 Advanced Formulation Formulation Program Controlled Small Sliding Program Controlled Detection Method Nodal-Normal From Contact Penetration Tolerance Program Controlled Elastic Slip Tolerance Program Controlled Normal Stiffness Program Controlled
Trim Contact Program Controlled Suppressed No Object ID (Beta) 71 Advanced Formulation Formulation Program Controlled Small Sliding Program Controlled Detection Method Nodal-Normal From Contact Penetration Tolerance Program Controlled Elastic Slip Tolerance Program Controlled Normal Stiffness Program Controlled Update Stiffness Program Controlled Thermal Conductance Program Controlled

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.

C	utline managements	→ ‡ □ ×	C	Outline accession accession		
	Name 💌 Se	arch Outline 💙 🖕		Name 💌 Se	arch Outline 🛛 🖌 🖕	
Project* Model (A4, B4) Mod		YND E	Project* Model (A4, B4) Model (A4, B4) Materials Ma	mports Racetrack\BasePlate Racetrack\PrintPart Systems s ctions for DED Thermal (A5) uctural (B5)		
D	etails of "DED_Racetrack	\BasePlate" 🕶 🔻 🗖 🗙	D	etails of "DED_Racetrack	\PrintPart" 🗸 🖛 🕇 🗖 🗙	
÷	Graphics Properties		÷	Graphics Properties		
Ξ	Definition		Ξ	Definition		
	Suppressed	No		Suppressed	No	
	ID (Beta)	28		ID (Beta)	31	
	Stiffness Behavior	Flexible		Stiffness Behavior	Flexible	
	Coordinate System	Default Coordinate System		Coordinate System	Default Coordinate System	
	Reference Temperature	By Environment		Reference Temperature	By Environment	
	Treatment	None		Treatment	None	
Ξ	Material		Ξ	Material		
	Assignment	Inconel 718		Assignment	Inconel 718	
	Nonlinear Effects	Yes		Nonlinear Effects	Yes	
	Thermal Strain Effects	Yes		Thermal Strain Effects	Yes	
+	Bounding Box	·	÷	Bounding Box		
+	Properties		+	Properties		
+	Statistics		+	Statistics		
Ξ	CAD Attributes		Ξ	CAD Attributes		
	PartTolerance:	0.00000001		PartTolerance:	0.0000001	
	Color:143.175.143			Color:143.143.175		

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.

	Wizard		→ ‡ ×
	DED Proces	s /\nsy	/S / ACT
	 Machine Settings Material Deposition F 72 	mm^3	s^-1
	 Build Conditions Preheat Temperature 	80	°C
+	 Geometry For Preheat T Scoping Method Geometry S 	emperature relection	
	Geometry 1 Face Preheat During Printing	Off	
	Process Temperature 🛛 🕘	1370	°C
	Room Temperature	23	°C
	Gas Convection Coeff Build	1E-05	W/mm ^{z.} °C
	Gas Convection Coeff Base Plate	1E-05	W/mm².°C
	Radiation	Off 🗸	
	Thermal Calibration	Off	
	 Cooldown Conditions 		
	Room Temperature	23	°C
	Gas Convection Coeff Build	1E-05	W/mm ^{2.°} C
	Gas Convection Coeff Base Plate	1E-05	W/mm².°C
	Radiation	Off	
	Time 5	3600	s
	Add Temperature Result Yes	\checkmark	~

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.

🔍 🝳 📦 📦 📽 🖺 🗅 🔿 🔸 🝳 🍭 🍭 Select 🍹	Wizard 4
Build Settings Build Settings	DED Process ACT Fixed Support Geometry Geometry Selection Geometry Selection Base Removal 2 No
	Add Deformation Result Yes
0.00 90.00 (mm)	Help Define structural boundary conditions for build and base removal. Choose Fixed Support: Define the area/points to fix the entire build assembly (part and base) in the machine. Usually this can be one or multiple sides of the base. Surfaces, edges and points can be selected.
45.00	Base Removal: Defines if the build should be removed from the base.
Selection Information : • 4	Yes: Build will be removed from the base after cooldown.
No Selection	No: Build and base will be kept as one unit.
C Tabular Data Graph	Exit Wizard Ba

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.

Outline to the second second	→ ‡ □ ×
Name	▼ Search Outline V
Project* Model (A Model (A Mate Mate Corr Mate	4, B4) netry Imports netry rials dinate Systems nections ded Selections rocess for DED Build Settings G-Code Clustering Cluster Settings sient Thermal (A5) Initial Temperature Analysis Settings Solution (A6) Solution Information Competence ic Structural (B5) Analysis Settings Fixed Support Solution (B6) Solution Information Competence Solution (B6) Total Deformation
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
ID (Beta)	77
Туре	Fixed Support
Suppressed	No

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 mm³, 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
- 5.2. Create Analysis System
- 5.3. Attach Geometry and Set Units
- 5.4. Load Sintering Process Add-on
- 5.5. Wizard Step 1 Identify Geometries
- 5.6. Wizard Step 2 Define Contact
- 5.7. Wizard Step 3 Define Constraints
- 5.8. Wizard Step 4 Generate Mesh
- 5.9. Wizard Step 5 Define Gravity
- 5.10. Wizard Step 6 Define Sinter Material
- 5.11. Wizard Step 7 Define Sinter Schedule
- 5.12. Wizard Step 8 Define Results and Solver Settings
- 5.13. Generate Sinter Schedule and Solve
- 5.14. Review Results

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.





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).

Mechanical [Ansys Mechanical Enterpr	ise]					_		×
D				Quick La	unch		^ 🛛	8-
Commands	ts Worksheet Keyframe	∾ Tags ⊘Wizard ⊡ Show Errors	Manage View:	s rmation r	🖨 Print Pre 🔐 Report I 🕮 Key Assi	eview Preview gnments	Layout	
👂 🤁 🔍 🍭 🔍 Select 🔩 ।	Unit Systems Metric (m, kg, N, s, V, A)		🔤 🗂 Clipk	ooard 👻	[Empty]	🛞 Exten	r d.≁	• «
4	Metric (cm, g, dyne, s, V, Metric (mm, kg, N, s, mV, Metric (mm, t, N, s, mV, m	A) mA)						
	Metric (mm, dat, N, s, mV, Metric (µm, kg, µN, s, V, r U.S. Customary (ft, Ibm, I U.S. Customary (in, Ibm, I	, mA) mA) Metr Set bf, °F, s, V, A)	ic (mm, t, N, s, m the units to this b	I V, mA) ase syster	m.			

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 **(I)** 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.

ି Q Q 🖗 କ 🏶 🕒 C ་ 💠 Q Q Q Q Q Select 🧏 Mode∽ 🕅 🦤	Wizard • 4 ×
	 Wizard Step: Identify Geometries Part Scoping Method Geometry Selection Geometry 1 Body Baseplate Yes No
	Help Select the part and baseplate (if any). If the bodies have been selected via the Geometry Selection scoping method, corresponding Named Selections for "Part" and "Baseplate" will be created in the tree.
	Baseplate: <u>Yes:</u> Select if a baseplate is present.
0.000 10.000 (mm)	Scoping Method: Part and Baseplate. Exit Wizard Back Next

- 3. Select the base body and click **Apply** in the Geometry field under Baseplate.
- 4. Click **Next** to move to the next step.

🛛 🔍 🔍 📦 📽 📲 🔿 = 💠 🤨 🔍 🍭 🍭 🕲 Select 💺 Mode = 👯 🍹	Wizard Vizard
Model	Sintering Process / ACT
	 Wizard Step: Identify Geometries Part
	Scoping Method Geometry Selection
	Geometry 1 Body Apply
	Baseplate Yes 🔻
	▼ Baseplate
	Scoping Method Geometry Selection
	Geometry 1 Body 3 Apply
	11-1-
	нер
	Select the part and baseplate (if any). If the bodies have been selected via the Geometry Selection scoping method, corresponding Named Selections for "Part" and "Baseplate" will be created in the tree.
	Baseplate:
	<u>Yes:</u> Select if a baseplate is present.
	<u>No:</u> Select if baseplate is absent.
, v ∳	Scoping Method: Part and Baseplate.
0.000 10.000 (mm)	Exit Wizard Ba 4 Next

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.

🛛 🭳 🔍 📦 📽 🖺 ⊖ - 🛟 Q Q Q 🍹	Contact Body View 🔻 🖡 🗖 🗙	Wizard			→ ‡ ×
Contact Region Contact Region (Contact Bodies) Contact Region (Contact Bodies)		Sinte	ring Process	/\nsys /	АСТ
Contact Region (Target Bodies)		 Wizard Step: D 	efine Contact		
		Select Contact Region	Contact Region (Id = 29)	•	
		 Frictional Conta 	act Settings		
		2 Friction Coefficient	0.2		
		3 Rigid Baseplate	Yes	•	
	Y	4 Small Sliding	Off	•	
	0.00	5 Update Stiffness	Each Iteration (Aggressive)	•	
	10100	6 Adjust To Touch	Yes	•	
	Target Body View 🛛 🔻 👖 🗙				•
		Help			
		Specify the part-base contact settings.	eplate contact region baseplate	and its corresponding	Î
		NOTE: This step is in Region. For multiple tree.	ntended for a simple setup of a Contact Regions, manually set	baseplate to part Contac up contact settings in the	t
		Select Contact Reg	gion:		
Ý		<u>None (skip this s</u> settings in the tr	<u>step):</u> Skip this step in the wizard a ee.	and manually setup contact	¥
0.00 (mm) 10.00 Z	0.00 ¥ 10.00 ×	Exit Wizard	Bac 7 Next		

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

Outline 👻 🖡 🗖 🗙	D	etails of "Contact Region"	
Name Search Outline V	Ξ	Scope	
Project*		Scoping Method	Geometry Selection
Model (A4)		Contact	1 Face
Firm Geometry Imports		Target	1 Face
		Contact Bodies	bridge_with_base\Bridge
		Target Bodies	bridge_with_base\Baseplate
🗄 🛶 🙀 Coordinate Systems		Protected	No
E	Ξ	Definition	
Contacts		Туре	Frictional
		Friction Coefficient	0.2
Named Selections		Scope Mode	Automatic
Part		Behavior	Program Controlled
Baseplate		Trim Contact	Program Controlled
		Trim Tolerance	0.13009 mm
√⊞ Analysis Settings		Suppressed	No
Solution (A6)		Object ID (Beta)	29
Solution Information	Ξ	Display	
		Element Normals	No
	Ξ	Advanced	
		Formulation	Program Controlled
		Small Sliding	Off
		Detection Method	Program Controlled
		Elastic Slip Tolerance	Program Controlled
		Normal Stiffness	Program Controlled
		Update Stiffness	Each Iteration, Aggressive
		Stabilization Damping Factor	0.
		Pinball Region	Program Controlled
		Time Step Controls	None
	-	Geometric Modification	
		Interface Treatment	Adjust to Touch
		Contact Geometry Correction	None
		Target Geometry Correction	None

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.

🔄 🔍 🔍 📦 🗣 🖺 🔾 * 💠 🍳 🍳 🍭 🍭 Select 🍡 Mode* 👫 🍹	Wizard 🗸 🗸 🗸 🗸 🗸
Model	Sintering Process
	 Wizard Step: Define Constraints
	Constraint Type 🗨
	None (skip this step) Fixed Support
	•
	Help
	Specify physical constraints to prevent rigid body motion during the sintering
	simulation.
	NOTE: There exists many ways to apply physical constraints to prevent rigid body motion. The combinations provided here are simply suggestions to guide the user
	toward an initial setup and are not intended to be comprehensive.
	Constraint Type:
Z X	None (skip this step): Skip this step and specify constraints manually in the tree.
0.000 10.000 (mm)	Fixed Support: Select a fixed surface on the part. When applied unto a rigid body, a
	Exit Wizard Back Next
5.000	

- 2. Rotate the model in the graphics window, use the Face picker **b** 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.

Outline concentration	
🕺 Name 💌	Search Outline 🖌 🖕
Project*	
🗄 🗝 🐻 Model (A4)	
🗄 🗤 🖉 Geometr	y Imports
	у
H. Materials	S .
	ions
	ntacts
	Contact Region
⊡	nnection Group
÷	Fixed - Ground To bridge_with_base\Baseplate
🗸 🌍 Mesh	
Hamed S	Selections
⊢	Structural (AS)
	arysis seconds
	Solution Information
71	· · · · · · · · · · · · · · · · · · ·
Details of "Fixed - Grou	und To bridge with base\Baseplate 🗙 🎚 🗔 🗙
Connection Type	Body-Ground
Туре	Fixed
Solver Element Type	Program Controlled
Element APDL Name	
Suppressed	No
- Reference	
Coordinate System	Reference Coordinate System
Behavior	Rigid
- Mobile	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Scope	1 Face
Body	bridge_with_base\Baseplate
Initial Position	Unchanged
Behavior	Rigid
Pinball Region	ΔΠ

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).

🔆 Q Q 📦 📽 📴 🔿 - 🎨 Q Q Q Q Select 🥄 Mode 🎼 👻	Wizard	×
Model	Sintering Process	Г
	Wizard Step: Generate Mesh	
	Mesh via Wizard Yes	
	Element Size 1 mm	
	Contact Sizing Yes	
		^
	Help	
	Select the to-be-sintered geometric part for meshing, with ability to specify body- sizing for the selected parts. Upon completion of this step, all geometries will be meshed.	ł
	NOTE: For more advanced meshing controls, directly interact with the 'mesh' object within the project tree.	
	Element Size: Specify the element body-sizing for the selected part.	
	Contact Sizing: Only visible if contact with baseplate has been specified	~
0.000 10.000 (mm)	Exit Wizard Ba	

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.

📃 🝳 🍳 📦 😜 🐁 🔿 + 🔆 🤨 🕲 🍭 🍭 🧕 Select 💺 Mode+ 🛒 🍹	Wizard 🗸 🗸 🗸 🗸
	Sintering Process
	Wizard Step: Define Gravity
	Gravity Direction -Z
	Help
	Select a gravitational direction along the one of the primary coordinate axis. Upon completion of this step, a 'Standard Earth Gravity' load object will be created in the
	model tree.
Z X	
0.000 10.000 (mm)	Exit Wizard Bac 2 Next

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.

💿 🥺 🔕 📽 🕒 🔿 🔸 🤨 🍳 🍭 🍭 🤇 Select 🍡 Mode 🛛 🕵 🍹	Wizard		→ ‡ ×
A: Static Structural Standard Earth Gravity Time: 1. s	Sintering F	Process 🔨	nsys / act
Standard Earth Gravity: 9906.6 mm/s ² Components: 0,0,-9806.6 mm/s ²	Wizard Step: Define Sinter Material Material Selection		
	Material 316L (P	IM) 🔻	
	2 Pre-defined Models Type 1	•	
	Initial Material Data		
	Initial Relative Density	0.5	
	4 Mean Powder Diameter	0.025	mm
	5 Sinter Activation Temperature	1000	C v
	Help		^
2000 10.000 (mm)	Specify the material model to between pre-defined materials	be implemented for sintering. The s or a user-defined model.	user can select
5.000	Initial Material Data		
Graph • 4 🗆 X Tabular Data • 4 🗆 X	<u>Initial Relative Density:</u> Sta applied to the entire part.	arting relative density of the part. The v	alue will be uniformly
1. Steps Time [s] ✓ X [mm/s ²] ✓ Y [mm/s ²] 0. -9-= 1 1 0. = 0. = 0.	ⁱ <u>Mean Powder Diameter:</u> The average powder particle diameter.		
-9806.6	Sinter Activation Temperature: Temperature above which the sintering process and		
1.	Exit Wizard	Bac 6 Next	
< >			

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.

Outline 👻 🖡 🗖 🗙	Details of "Sinter Material" 👻 🕈 🗖	×
Name	Geometry	
Project*	Scoping Method Geometry Selection	
Em B Model (A4)	Geometry 1 Body	
Geometry Imports	Sintering Model	
	Material 316L (PIM)	
🕀 Materials	Pre-defined Models Type 1	
🗄 🛶 🙀 Coordinate Systems	Initial State Data	
	Green Density 0.5	
	Mean Powder Diameter 0.025 mm	
The second selections	Sintering Stress	
一 · · · · · · · · · · · · · · · · · · ·	☐ Activation Temperature 1000 °C	
Standard Earth Gravity	Model Olevsky	
Sinter Material	Input by Tabular	
🖻 🌾 Solution (A6)	Sintering Stress Parameters Tabular Data	
Solution Information	Uniaxial Viscosity	
	Model Arrhenius	
	Input by Tabular	
	Arrhenius Parameters Tabular Data	
	Grain Growth Kinetics	
	Model Parabolic	
	Initial Grain Size 0.006 mm	
	Input by Tabular	
	Arrhenius Parameters Tabular Data	
	Viscous Moduli	
	Model Riedel	
	Shear Moduli density Coefficient 1	
	Shear Moduli density Exponent 2	
	Bulk Moduli density Coefficient 1	
	Bulk Moduli density Exponent 2	
	Viscous Poissons coefficient 0.5	

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.

💿 😟 💿 📦 🌚 🕒 🔿 🤹 🤁 🥹 🥹 🕲 🕲 🕲 🕲 Select 💺 Moder 🛒 🍹	Wizard		→ ‡ ×
A: Static Structural Sinter Material Time: 1. s	Sinte	ering Process	Ansys / Act
Sinter Material	 Wizard Step: 	Define Sinter Schedule	
	Room Temperature	22	C
	Input Furnace Cycle	Temp-Duration	
2	Add Delete selected Temperature [C] 2 1380 [C]	Time/Duration 7200 [sec]	
	Help		Â
	Specify the thermal simulation. The load	schedule as used in the sinteri ds are ramped (default).	ng furnace and applicable to the
	Room Temperatu	re: The room temperature from	which the furnace will be ramped.
	Input Furnace Cy	cle: The input type for the temp	erature table.
Z Y X	Temp-Duration	Specifies in terms of the temperat	ture and duration needed to reach \checkmark
0.000 10.000 (mm)	Exit Wizard	Back Next	

- 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.

🔍 🔍 📦 😜 🕒 🔾 + 💠 🍳 🍭 🍭 🧶 Select 💺 Mode - 🕵 🍹	Wizard		→ ‡ ×						
A: Static Structural Sinter Material Time: 1. s	Sinte	Ansys / Act							
Sinter Material	 Wizard Step: I 								
	Room Temperature	С							
	Input Furnace Cycle	Temp-Duration							
3	3 Add Delete selected rows								
	Temperature [C]	Time/Duration							
	1380 [C]								
	31380 [C]	3 3600 [sec]							
	Help								
	Specify the thermal schedule as used in the sintering furnace and applicable to the simulation. The loads are ramped (default).								
	Room Temperature: The room temperature from which the furnace will be ramped.								
	Input Furnace Cycle: The input type for the temperature table.								
Z X	Temp-Duration:	Specifies in terms of the tempera	ture and duration needed to reach 🗸						
0.000 10.000 (mm)	Exit Wizard	Ba(4 Next							

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.

Outline	🕂 🗖 🗙						
Name 💌 Search (Dutline 🗸 🗸						
🗄 🗝 🐻 Model (A4)	^						
🗄							
🗄 🗸 🧐 Geometry							
🗄 🗤 🔽 Materials							
⊡							
Elim Mesh							
1 Named Selections							
Static Structure	al (A5)						
🗸 🛺 Analysis Set	tings						
Sinter Material							
Sinter Sched							
Solution (A6)							
	V						
Details of "Sinter Schedule"	→ ‡ 🗆 ×						
Definition							
Heating Mode	Isothermal						
Environment Temperature	22 °C						
Start Time	1 s						
Geometry							
Scoping Method	Geometry Selection						
Geometry	1 Body						
Thermal Schedule							
Number of Set Points	2						
Set Point :	1						
Input by :	Duration						
Temperature	1380 °C						
Duration	7200 s						
Set Point :	2						
Input by :	Duration						
Temperature	1380 °C						
Duration	3600 s						
Information							
Final Time	Not Generated						
Adjust Analysis Time/Step	Yes						

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.

💿 🝳 📦 🗣 🖺 🔿 - 🔆 🍳 🕲 🍭 🥹 Select 🥆 Mode 🛒 🌷	Wizard	6 ×					
A: Static Structural Sinter Schedule Time: 1. s	Sintering Process //nsy	/S / ACT					
Sinter Schedule	 Wizard Step: Define Results and Solver Settings Results Generate Result Objects Yes Recommended Settings Large Deflection Yes Quasi-Static Solution Yes 	^					
	4 Auto Time Stepping Yes 4 - Min Time Step 1 4 - Max Time Step 1000	~					
0.000 (mm)	Help Generates result objects and configures the solver for recommended settings. Generate Result Features: Switch to generate the output results for the sintering material model. Yes: Result objects will be generated. No: Result objects will not be generated Large Deflection: Applies large deflection assumption into the solver. Quasi-Static Solution: Applies quasi-static assumption into the solver. Exit Wizard Ba 5 Finish						
Graph							

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 **Deform-ation Plot Tracker** to see the deformation during solution.

Outline 👻 🕂 🗖 🗙	Worksheet	- 1				
Value Search Outline Project* Projec	WORKNEET DISP CONVERGENCE VALUE = 0.2454E-02 CRITERION= 0.1460E-03 LINE SEARCH PARAMETER = 1.000 SCALED MAX DOF INC = -0.2454E-02 FORCE CONVERGENCE VALUE = 0.1253E-04 CRITERION= 0.6929E-03 «< CONVERGED EQULI ITER 3 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC = -0.7463E-09 Kinetic Energy = 0.6960E-14 Potential Energy = 0.1778E-03 DISP CONVERGENCE VALUE = 0.7463E-09 CRITERION= 0.1490E-03 «<< CONVERGED LINE SEARCH PARAMETER = 1.000 SCALED MAX DOF INC = -0.7463E-09 >>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATION 3 *** TIME = 11.0000 TIME INC = 10.0000 **** MAX VISCOELASTIC STRAIN STEP = 0.1464E-07 CRITERION = 0.1000E-01 Kinetic Energy = 0.6960E-14 Potential Energy = 0.1778E-03 **** AUTO STEP TIME: NEXT TIME INC = 10.0000 UNCHANGED FORCE CONVERGENCE VALUE = 0.7857E-02 CRITERION= 0.1402E-03 «<< CONVERGED EQUIL ITER 1 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.9990E-08 Kinetic Energy = 0.6960E-14 Potential Energy = 0.1780E-03 TOREC CONVERGENCE VALUE = 0.9990E-08 CRITERION= 0.1402E-03 «<< CONVERGED EQUIL ITER 1 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.9990E-08 Kinetic Energy = 0.6860E-14 Potential Energy = 0.1780E-03 TORVERGENCE VALUE = 0.1474E-10 CRITERION= 0.1401E-03 «<< CONVERGED FORCE CONVERGENCE VALUE = 0.1474E-10 CRITERION= 0.6790E-03 TREGENCE VALUE = 0.1474E-10 CRITERION= 0.6790E-03 H PARAMETER = 1.000 SCALED MAX DOF INC = 0.9990E-08 Kinetic Energy = 0.6960E-14 Potential Energy = 0.1780E-03 acker 1.0000 TIME INC = 10.0000 KINETRE ENERGENCE VALUE = 0.1474E-10 CRITERION= 0.1402E-03 *** AUTO TIME STEP = 0.1465E-07 CRITERION = 0.1000E-01 Kinetic Energy = 0.6960E-14 Potential Energy = 0.1780E-03 *** AUTO TIME STEP = 0.1465E-07 CRITERION = 0.1000E-01 KINETIC ENERGY = 0.6960E-14 Potential Energy = 0.1780E-03 *** AUTO TIME STEP: NEXT TIME INC = 15.000 INCREASED (FACTOR = 1.5000)					
Details of "Solution Information"	DISP CONVERGENCE VALUE = 0.1873E-07 CRITERION= 0.1402E-03 <<< CONVERGED					
Solution Information Solution Output Solver Output	EQUIL ITER 1 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -0.1873E-07 Kinetic Energy = 0.6960E-14 Potential Energy = 0.1286E-03					
Newton-Raphson Residuals 0	DISP CONVERGENCE VALUE = 0.1873E-07 CRITERION= 0.1431E-03 <<< CONVERGED LINE SEARCH PARAMETER = 1.000 SCALED MAX DOF INC = -0.1873E-07					
Identify Element Violations 0						
Update Interval 2.5 s	FURCE CONVERGENCE VALUE = 0.2936E-10 CRITERION= 0.6790E-03 <<< CONVERGED					
Display Points All						
FE Connection Visibility	*** TIME = 36.0000 TIME INC = 15.0000					
Activate Visibility Yes	*** MAX VISCOELASTIC STRAIN STEP = 0.1467E-07 CRITERION = 0.1000E-01					
Display All FE Connectors	Kinetic Energy = 0.6961E-14 Potential Energy = 0.1786E-03					
Draw Connections Attached To All Nodes	*** AUTO TIME STEP: NEXT TIME INC = 22.500 INCREASED (FACTOR = 1.5000)					
Line Color Connection Type	FORCE CONVERGENCE VALUE = 0.3315E-01 CRITERION= 0.6658E-03					
Visible on Results No		\sim				
Line Thickness Single						
		2				

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.)

	Context		A : Static Structural - Mechanical [Ansys Mechanical Enterprise]									_		x	
File Hol	Result	Display	Selection	Automation	n Add-ons	Sinteri	ng Process	5				Quick Launch		^ ☑	\ ? •
Duplicate Q Solv Outline Solv	re Analysi	Pamed S ﷺ Coordina ■ Remote F	election [ate System [Point] Inse	Commands Comment Chart ert	Images ▼ Images ■ </td <td>Display</td> <td>→ Vector Display*</td> <td>Capped Isosurface</td> <td>Views</td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td>	Display	→ Vector Display*	Capped Isosurface	Views						
Outline Name ▼ ⊡ ☐ Model (A4) ⊕ ∑ Geometr	Search Outl	∝	Q (Q 📦 ¥ 🐇) = = <mark>1</mark>	1.0 (Tri Scopec ✓ Lar	ue Scale) d Bodies ge Vertex (Tontours G	eometry Co Dis	ontours Edges	123) Probe 1230 Maxir 1230 Maxir 1230 Minir	e			*

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.