Ansys LPBF Simulation Guide

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

Release 2025 R1
January 2025

Published in the U.S.A.

Table of Contents

Chapter 1: Introduction to Additive Manufacturing

Additive manufacturing (3D printing) can be a cost-effective way of producing parts, especially when making use of the design freedoms the manufacturing process enables, such as topological complexity and the ability to print assemblies in one step.

Metal additive manufacturing is used to produce parts for aerospace, automotive, medical and other industries. These are high-value parts that require careful design and manufacturing, and simulation has long been used to validate the as-built part performance.

The additive process for metals introduces inherent complexities and challenges, however, such that *the process itself* requires simulation to successfully produce the parts.

Additive Manufacturing Processes

Additive manufacturing (AM) is classified into a number of processes, most of which are applicable to polymers. Two are the primary processes for fully-dense (no porosity) production of metal parts: **laser powder bed fusion** (LPBF) and **directed energy deposition** (DED). Our focus is on modeling these two processes.

In a laser powder bed fusion process – also known as direct metal laser melting (DMLM), direct metal laser sintering (DMLS), or selective laser melting (SLM) – a thin layer of metal powder is deposited and a highly focused laser beam of energy is moved over its surface in order to melt the metal powder composing the current cross section and fuse it to the preceding layer. A solid part emerges as successive layers are deposited and processed. The initial layer is deposited on a build plate or substrate.

In a directed energy process (DED) – also known as laser engineered net shaping (LENS), electron beam additive manufacturing (EBAM®), or laser deposition technology (LDT) – a laser or electron beam creates a melt pool on previously solidified material where blown powder or fed wire is introduced to add material.

Both of these processes produce high temperatures and severe thermal gradients, leading to significant distortion and buildup of residual stresses as the layers are deposited. The distortion can be high enough to interfere with the application of the next layer, and the residual stresses high enough to break the part off the build plate or off its supports, or crack the part itself. Additionally, the residual stresses will produce more distortion when the part is removed from the build plate and its supports removed leading to an undesirable final shape.

How Simulation Can Assist with AM Challenges

Being able to simulate these distortions and stresses during the design of the part will help prevent failed builds and lead to better designs for additive manufacturing.

Supports are generally needed to anchor and support overhangs and other horizontal (and nearly horizontal) surfaces such as the tops of holes. They are also used to control distortions and provide heat transfer routes during the build. Supports add cost – material, build time, and removal effort – so their use should be minimized. Simulation can be used to determine the best build orientation for a part, best locations for supports, and support sizing requirements. Simulation is particularly powerful when used with topology optimization to minimize overhang regions requiring supports.

Chapter 2: Additive Simulation in the Mechanical Application

Ansys Additive Suite™ is a powerful collection of tools from Ansys, Inc. dedicated to additive manufacturing simulation. This document describes the LPBF additive manufacturing simulation capabilities in the Ansys Mechanical™ application using the Ansys Workbench™ framework.

Target Users

Target users of AM capabilities in the Mechanical application are the engineers involved in the design and analysis of mechanical components, not necessarily manufacturing engineers and technicians tasked with printing the parts on the machine floor, nor the R&D researchers responsible for determining the ideal printing machine process parameters. Current users of the Ansys Discovery™ and Ansys Mechanical applications will benefit greatly from running AM Process Simulations if they plan to use additive manufacturing to print their metal parts.

Simulation Goals

The goal of Additive simulations in Mechanical is to predict the macro-level distortions and stresses in parts to prevent build failures and provide trend data for improving designs for additive manufacturing including part orientation and support placement and sizing.

The simulation is not meant to provide detailed thermal or structural results needed for prediction of micro-level process phenomena (that is, microstructure). The simulation will also not provide detailed guidance on the setting of the machine's process parameters. Our complementary offerings of Ansys Additive Print and Additive Science (within Additive Suite) are the products to use to achieve those goals.

Methodology and Abstractions

Simulation of the manufacturing process requires that the analysis follows the build process itself: layer-by-layer solidification of the part. Since the thermal (temperatures) and structural (distortion and stress) physics are largely uncoupled (that is, a weak coupling), we can simulate the thermal phenomena first, layer-by-layer, and use those temperature results in a following structural simulation.

In an AM Process Simulation, the model evolves over time; that is, elements are added. We actually mesh the entire part first with a layered mesh (either Cartesian or Tetrahedrons) and then use the standard **element birth and death** technique to "turn on" element layers to simulate the build progressing. Additionally, the relevant boundary conditions also evolve such as thermal convection surfaces. The build step is complete when all the element layers have been added (made "alive").

The analysis times and time stepping are also driven by the process parameters and are not known a priori. These details are all handled internally during the solution.

Simulating the entire build process for a real part following the beam scan pattern would take enormous compute time making it impractical. To meet our goals in a reasonable compute time – meaning much less than the actual build time – we use the following abstractions:

Super layers: Actual metal powder deposit layers are aggregated into finite element "super layers" for simulation purposes. Since the temperature histories of each adjacent layer is similar, this lumping approach is appropriate. Note: The real machine build time is approximately the transient thermal build step simulation time multiplied by R^(1/3), where R is the number of deposit layers in one element super layer.

Layer-by-layer addition: Material is added and heated all at once for each element layer. For current generation machines and their scan patterns, this is a reasonable assumption. The in-plane thermal effects do not contribute to the distortion as much as the build direction thermal effects. This means we do not use scan pattern information as input.

Heat Application: Heat is applied to new layers in one of two ways, an applied *temperature* or *power-based heat generation*. For applied temperature, the assumption is that the process parameters for the build have been set appropriately so that (1) the developed temperature is always at or above melt (no lack of fusion) and (2) the developed temperature does not substantially exceed melt (no keyholing). For heat generation, the power input and absorptivity is specified and a heat generation load is applied to elements in the new layer. The time over which the heat generation is applied can be set to either a very quick "Flash" option where the energy is added in a short amount of time, or a longer "Scan Time" option where the energy is added over the amount of time it would take to scan the new element layer. In both timing options, the input energy is the same. More information on the heat generation option can be found in the command descriptions for the underlying Mechanical APDL™ commands **AMBEAM** and **AMBUILD**.

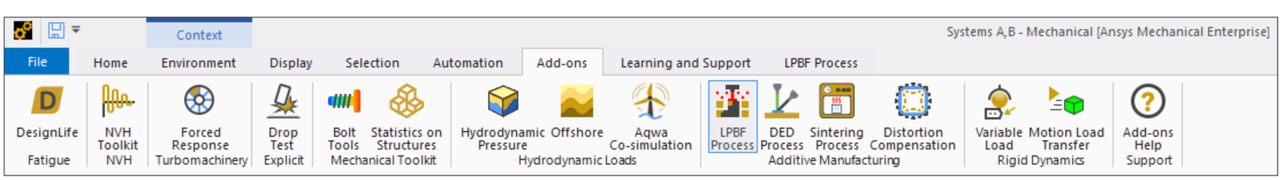
Time step size: Large integration time step sizes are used throughout the simulation. This is sufficient to capture the induced thermal and plastic strains driving the distortion. The localized smooth heating and cooling curves will not be captured in detail.

Supports: Supports are represented as an orthotropic homogenized solid. While you can provide detailed support geometry, modeling this way is sufficient to capture part distortion and obtain estimates of support failure.

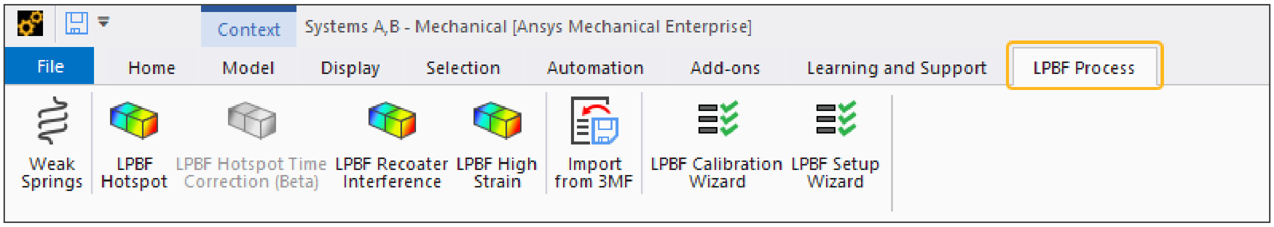
Surrounding powder: For the powder bed process, the surrounding unmelted powder need not be explicitly modeled. Instead, the heat loss into the powder can be accounted for in a simplified approach using a convective boundary condition at the interface between powder and solid material.

The LPBF Process Add-on and the LPBF Setup Wizard

An AM process simulation can be run using native Mechanical interface objects and working through the project tree. For an easier approach, however, we recommend you load the LPBF Process Add-on, accessible from the Add-ons Tab in the Mechanical ribbon. Simply click the LPBF Process button (in the Additive Manufacturing group) to load the add-on. The button is shaded blue when the add-on is loaded.



Click the LPBF Process tab to see the add-on's custom ribbon:



An easy-to-use LPBF Setup Wizard steps you through all the steps required to set up a simulation. If you are new to Ansys Workbench/Mechanical, or if you are an occasional user, we highly recommend you use the wizard. The workflow is described in Using the LPBF Setup Wizard (p. 17). We also recommend you read Application Interface in the *Mechanical User's Guide*.

Even existing Ansys Workbench/Mechanical users may want to try the LPBF Setup Wizard because it's is an excellent tool for keeping you on track and preventing you from missing a task. There are a few advanced capabilities that are not available in the LPBF Setup Wizard, but because it does not include a solve, you can easily modify your simulation set-up after you complete the wizard. If you decide not to use the wizard, see Workflow Through the Project Tree (p. 67) for the procedure.

Elements, Commands, and Interface Objects Used in AM Process Simulations

The following tables show the primary elements, Mechanical APDL commands, and Mechanical interface objects used in AM Process Simulations. These items are well documented in our reference guides and links are provided.

Element	Description
SOLID278	3D 8-node thermal solid element (Cartesian mesh)
SOLID185	3D 8-node structural solid element (Cartesian mesh)
SOLID291	3D 10-node tetrahedral thermal solid (Tetrahedrons mesh)
SOLID187	3D 10-node tetrahedral structural solid (Tetrahedrons mesh)
CONTA174	3D 8-node surface-to-surface contact element
TARGE170	3D target segment
SURF152	3D thermal surface effect

Command	Description
AMBEAM	For multiple-beam printers, specifies the number of beams.
AMBUILD	Specifies printer parameters for the build and other options.
AMENV	Specifies the build-environment thermal boundary conditions.
AMMAT	Specifies the build-material melting temperature.
AMPOWDER	Specifies the powder thermal conditions.
AMRESULT	Specifies AM-specific result data written to a `.csv` file.
AMSTEP	Specifies the process-sequence steps.
AMSUPPORTS	Specifies the information about supports.
AMTYPE	Specifies the printing process, PBF or DED.

Interface Object	Description
AM Bond (p. 209)	Establishes a connection between a meshed part and a meshed support in an AM Process Simulation.
AM Process (p. 211)	Identifies geometries and sets global options for all AM-related objects in an AM Process Simulation.
Build Settings (p. 214)	Defines strain assumptions, process parameters, and build conditions related to the additive machine, material, and process.
Generated Support (p. 224)	Creates a support structure consisting of finite elements.
LPBF High Strain (p. 227)	Identifies areas of high strain that are prone to cracking in an AM Process simulation.
LPBF Hotspot (p. 230)	Identifies areas of overheating that may result in problematic thermal conditions in an AM LPBF Thermal Structural simulation.
LPBF Hotspot Time Correction (Beta) (p. 233)	Identifies dwell times to correct overheating in an AM LPBF Thermal Structural simulation.
LPBF Recoater Interference (p. 233)	Identifies areas that may result in recoater interference issues in an AM Process simulation.
Predefined Support (p. 236)	Identifies a support structure that was imported with CAD geometry.
STL Support (p. 239)	Imports and meshes a support structure that is an STL (Stereolithography) file, of either volumeless or solid type.
Support Group (p. 243)	Groups Predefined Support objects and Generated Support objects.
Weak Springs (p. 245)	Loosely holds parts in place during AM post process steps to prevent rigid body motion.

Known Issues and Limitations

Note the following limitations:

- If you use a Mechanical Model from Workbench's Component Systems, you must manually create the AM Custom System. This is due to the fact that AM Custom Systems are pre-populated with AM sample materials in Engineering Data, and linking to the Mechanical Model necessitates the absence of materials in Engineering Data.

- The LPBF Setup Wizard is not yet fully supported when using the standalone Mechanical application. For example, if you open the Mechanical application independent of Workbench and use the LPBF Setup Wizard to configure a heat treatment with creep properties, the creep properties will be incorrectly configured. Using creep properties from the wizard necessitates the use of Engineering Data *in Workbench*.

- For a *multi-part* Inherent Strain simulation with either Scan Pattern or Thermal Strain strain definition (p. 104) and a generated scan pattern (rather than a build file), scan stripes will likely not align in the x-y direction between parts.

- On machines with AMD processors, an Inherent Strain simulation with Thermal Strain strain definition fails by default. Workaround: Set a global environment variable of TF_ENABLE_ONEDNN_OPTS=0 and restart the Workbench/Mechanical application.

- An Additive Manufacturing Process Simulation is not meant to be used in conjunction with any of these features: gasket elements, fracture, and remote boundary conditions.

- When using Workbench scripting to open SpaceClaim and then use Additive Prep, the Additive tab may be missing and/or the tools in the Additive ribbon may be grayed out.

- In certain configuration scenarios, and for some .scdoc geometries that were created prior to Release 2021 R2, if you have the Additive Prep license enabled in SpaceClaim you will not be able to open the Mechanical application from inside Ansys Workbench. Mechanical will begin to launch and then it gets stuck at this stage. If this happens, the workaround is to do one of the following:

  – Within SpaceClaim 2021 R2 or a subsequent release, save the older .scdoc geometry as a new file using Save, Save As, or Export from Additive Prep.

  – Clear the Additive Prep license check box in SpaceClaim, perform your work in Mechanical, and then go back into SpaceClaim to enable Additive Prep for your next session. Additive Prep license options are accessible in SpaceClaim by clicking **File** > SpaceClaim Options > **License** and then checking/unchecking **Additive Prep**.

- Result items from the LPBF Process Add-on (LPBF Recoater Interference and LPBF High Strain) with an AM Octree simulation (p. 163) may cause the Mechanical application to crash if an analysis is cleared and resolved and then the result items are evaluated again.

- Multiframe restarts are not supported for the build step. Restarts are supported for the other AM process steps (p. 97).

- **Linux system**:

  – The LPBF Setup Wizard introduced as the default wizard at Release 2023 R2 is not supported on a Linux platform. Linux users use the Legacy LPBF Setup Wizard instead, which is loaded as the default wizard on Linux systems.

  – The voxelized mesh option (Cartesian Mesh with Voxelization Option), one of the three recommended mesh methods (p. 78) for layered simulations, is not available on a Linux platform because the underlying voxelizer code is incompatible with Linux. Choose between a Cartesian mesh and a layered tetrahedrons mesh.

High Performance Computing

An AM Process Simulation can be very compute intensive. We recommend you use high performance computing using an Ansys HPC license to take advantage of more than four cores.

Chapter 3: Preparing the Part for Simulation

There are several Design for Additive Manufacturing (DfAM) considerations to be aware of that affect the cost and quality of your part. Variables include:

- Part design – Relevant design parameters include thicknesses, overhangs, edges, holes, and desired surface finish.
- Orientation on the base plate – Orientation affects time to print, number of support structures, and number of replicate parts that can be nested on a base plate.
- Support structures – Support structures serve to stabilize the part, lower deformations, and conduct heat away from part, but using more supports increases print and finishing time and cost.

  In particular for support structures, you should think ahead about how you will remove them. In some cases you may even need to design tooling rails onto the part in order to be able to hold the part while supports are removed.

Determining the Location of Supports

You can determine ahead of time which overhang locations on your part will require supports. Use the Overhangs Tool in the Ansys Discovery™ application, located on the Facets tab, to determine where supports will be generated.

Cleaning Up Facets

If you have an `.stl` file, 3D Printing requires that the faceted body be watertight and free of self-intersecting facets, or other defects. Be sure to clean up your file to eliminate gaps and slivers before importing it into Ansys Workbench.

In Ansys Discovery under **Facets**, the **Check Facets** and the **Auto Fix** tools are quick ways to find and fix problems in faceted bodies.

Many additional tools for working with faceted parts are available in the Discovery application. Refer to the Facets section in the Discovery Documentation.

To Share Topology or Not?

The requirement that geometric bodies be meshed in uniform layers in the Z direction is unique to additive manufacturing simulations. These uniform mesh layers must be conformal, even for multi-body parts or geometries consisting of separate part and support bodies. That is, not only must faces, edges, and vertices of distinct 3D printed bodies be aware of one another in order to communicate information in the simulation, but they must also be sliced into layers with the same step size in the Z direction. Together, the part and the supports constitute *"the build."*

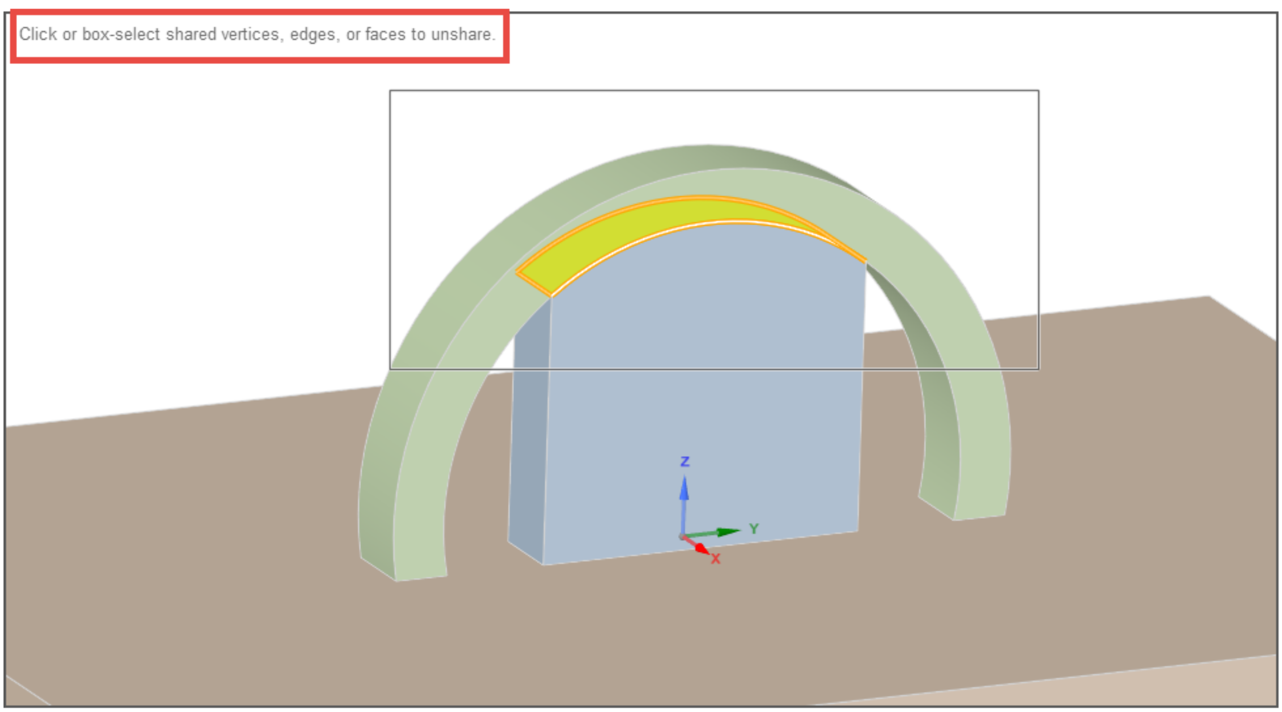
In the Mechanical application, there are three approaches for achieving these uniform mesh layers: using a Layered Tetrahedrons mesh, using a Cartesian (brick) mesh, and using a voxel (cubic) mesh. Which of these mesh approaches you employ will determine whether or not you share topology between bodies in the build—upstream in the CAD program.

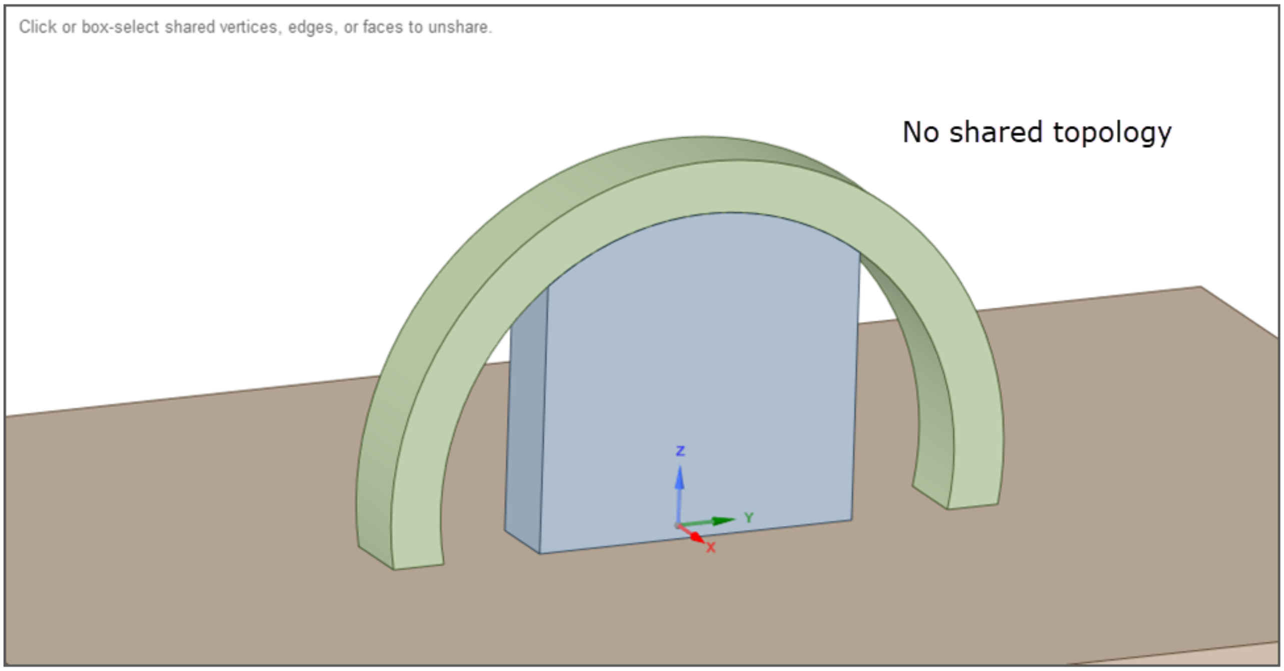
Use the following *criteria for deciding which type of mesh method to use:*

- **Use a Layered Tetrahedrons mesh if:**
  - Your geometry has thin walled features, organic curves, and/or holes.
- **Use a Cartesian mesh if:**
  - Your geometry is blocky or chunky without fine features or holes.
  - You know you will be using AM octree adaptive meshing (p. 163) to coarsen the mesh for reduced simulation time. (The AM Octree method is not supported with the layered Tetrahedron mesh method.)
  - You know you will be generating supports in Ansys Mechanical. (Auto-generated supports are not supported with the layered Tetrahedron mesh method.)

Once you've decided which type of mesh method to use, use one of the following approaches, as appropriate.

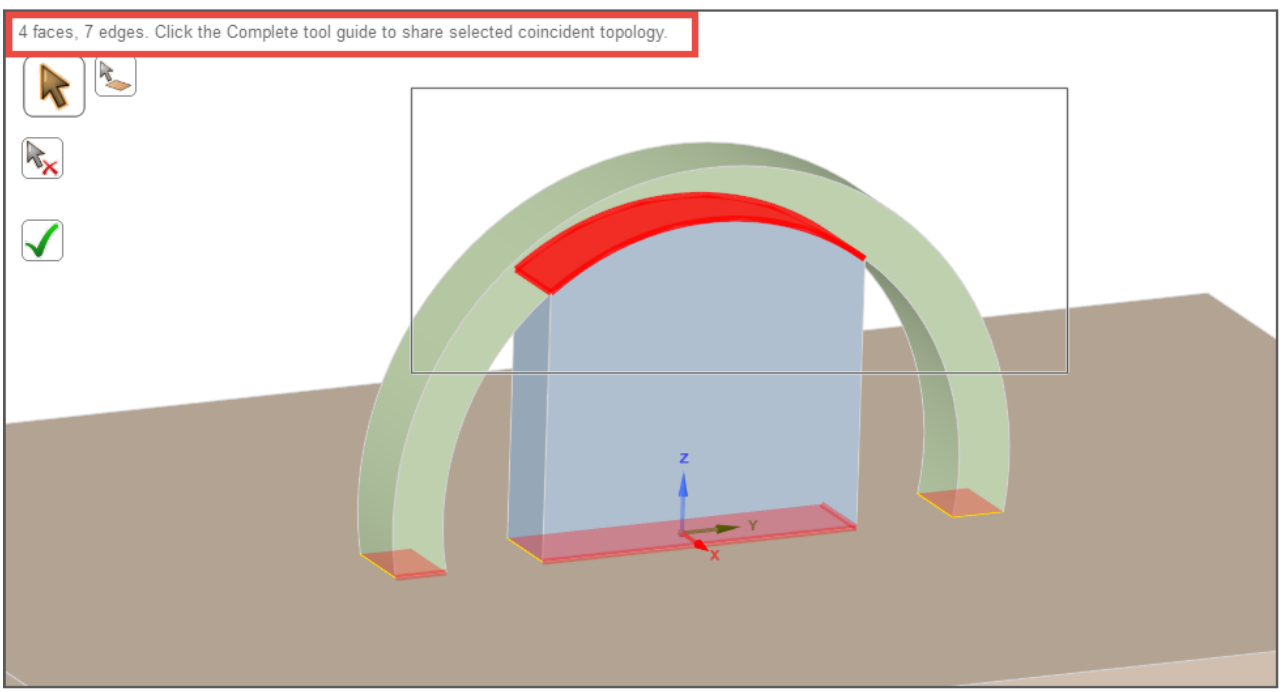
- **If using a Layered Tetrahedrons mesh:**
  - Do *not* share topology in the CAD program. For example, in Discovery, use the Unshare tool in the Shared Topology group to unshare coincident topology.

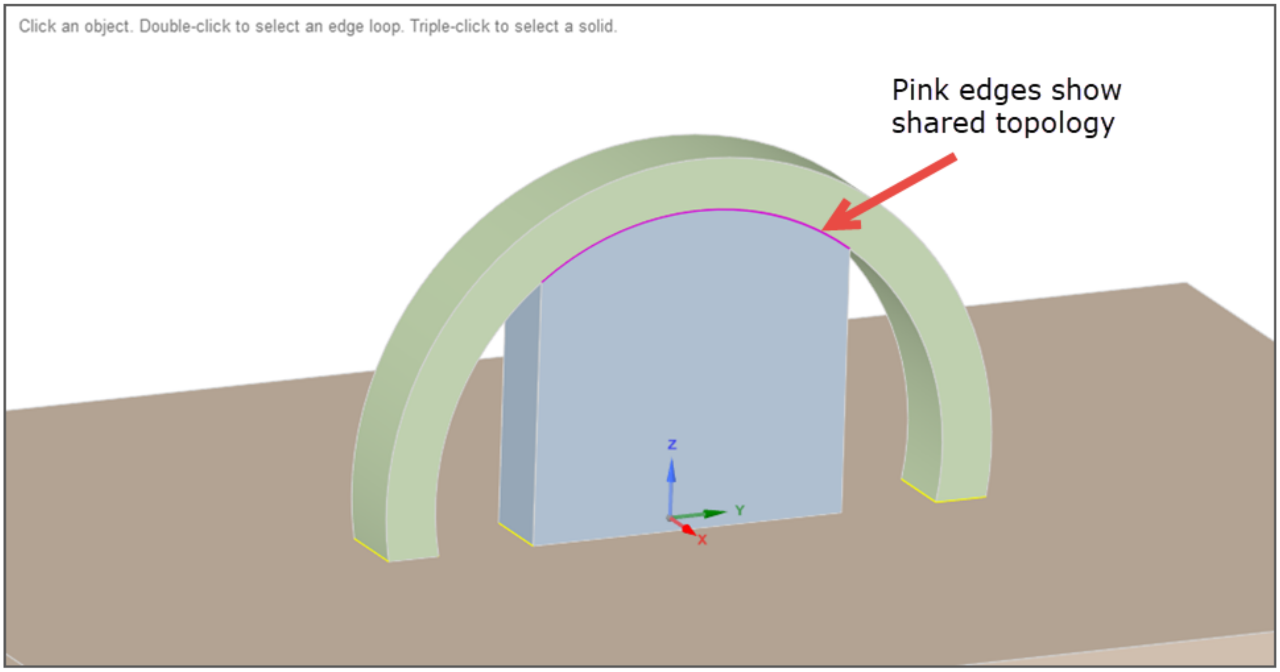




Or use **Explode Part** in DesignModeler.

- **If using a Cartesian mesh:**
  - *Share coincident topology* in the CAD program. For example, in Discovery, select all bodies that are associated with the part and the supports. Use the Share tool in the Shared Topology group to share coincident topology.





- **If using a voxel mesh:**
  - It is recommended to use *unshared* topology. Shared topology between part bodies, predefined supports, and/or powder bodies is allowed with the voxelization option but not recommended because some elements may get assigned to the wrong body at the interface.

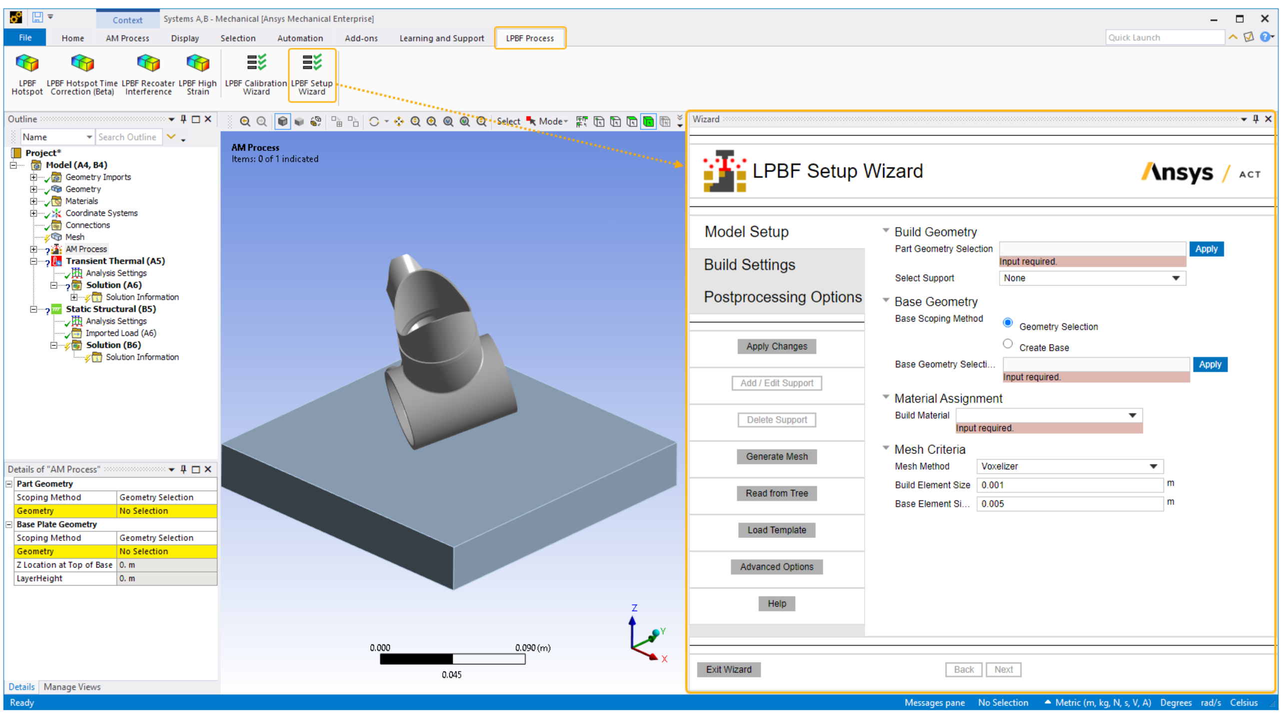
Chapter 4: Using the LPBF Setup Wizard

Begin your laser powder bed fusion (LPBF) simulation by using the LPBF Setup Wizard. Follow the wizard steps, which are organized into logical tasks for additive simulations. As you proceed through a step, click Apply Changes, which automatically adds objects to the project tree in the Mechanical application's user interface, and enters values for object properties. During solution, the settings in the project tree are executed, so it is important to understand that the wizard serves to simplify and focus your workflow for the purpose of setting up the project tree. Unless you are doing some advanced customization, the simulation is ready to solve when you have completed the last step in the wizard. Click Finish or Exit (p. 19) to leave the wizard.

Opening the Wizard

First make sure the LPBF Process Add-on is loaded (p. 8).

To start the wizard, select the **LPBF Process** tab and then select the **LPBF Setup Wizard** button.



The geometry in this example, which is used throughout the documentation, is courtesy of Aconity3D GmbH and Brightlands Chemelot Campus.

Overview of the Wizard Steps

The LPBF Setup Wizard guides you in a step-by-step manner through the following steps:

1. Model Setup (p. 21):

- Identify geometries of the part(s), supports, and base plate. Use Advanced Options for the simulation of powder or non-build items (such as clamps, bolts, etc.), or if you want to take advantage of symmetry.
- Select your material. Use Advanced Options if the base plate is a different material than the build items.
- Set mesh criteria, such as type and sizing.

2. Build Settings:

   This step is slightly different depending on whether you have an Inherent Strain simulation (p. 49) or a Thermal-Structural simulation (p. 53).

   - Simulation assumptions are set by default based on your chosen AM LPBF system. For Inherent Strain simulations, choose among the options for strain definition.
   - Enter calibration scaling factor(s) as determined from calibration experiments (p. 199). Use default calibration factors if your goal is simply to examine *trends*, that is, the effects of variable changes on stress or distortion *relative to each other*.
   - Enter machine settings where applicable, which are process parameters that directly influence how the process deposits material. These can include Deposition Thickness, Hatch Spacing, Scan Speed, and a number of other factors.
   - Enter build and cooldown conditions, which are settings pertaining to the environment *around the part* during the deposition process and during cooldown. These include Preheat Temperature and gas and powder boundary conditions.
   - For Thermal-Structural simulations, enter boundary conditions applied on the specified *surface of the base*, usually the underside surface, during the build and during cooldown. The wizard automatically selects the underside surface of the base by default.

3. Postprocessing Options (p. 59):

   - Select the post-build processing steps you want to simulate, such as base unbolting, heat treatment, and base and/or support removal.
   - Select LPBF result items to be calculated during solution, such as Hotspot, Recoater Interference, and High Strain. Specify criteria for warning and critical thresholds for these items, or add multiple versions with different criteria for comparison.

Recommended Workflow and Best Practices

- **Know the relationship between the wizard and the project tree objects**

  As you progress through the wizard and take actions, such as Apply Changes, Generate Mesh, etc., the options you specified in the wizard are implemented as objects and options in the project tree. Mesh objects, boundary condition 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, support bodies, contact connections, and constraint nodes. For convenience, the help for each wizard option lists the affected tree objects.

- **Use the wizard first to set up your simulation before using more advanced customization**

  The wizard is capable of reading standard objects relevant to the AM Process from the project tree. These include assigned geometry types, mesh settings, build settings, boundary conditions and heat treatment settings, if applicable. Upon opening, the wizard will pre-populate it's properties based on the given project if the objects have default Ansys names (not custom names). If the project itself was set up using the wizard, the wizard will read all required properties so you can simply jump in and navigate to any step and change any properties as desired.

  Important:

  The wizard is not designed to accommodate advanced custom settings, such as complex mesh or contact settings, that you may set directly in the project tree. As a best practice, we recommend that you use the wizard first to set up the majority of your simulation, Finish or Exit from the wizard, and then tweak the advanced settings in the tree as needed. Advanced custom settings refers to detailed options that are not present in the wizard.

- **Use Read from Tree to update the wizard for standard setup options**

  If you change some *standard* project details (see Important note above) directly in the project tree while a wizard session is open, you can use the Read From Tree button to update the wizard. Think of the Apply Changes and Read from Tree actions as opposite functions. Apply Changes updates the project tree with inputs from the wizard. Read from Tree updates the wizard with inputs from the project tree. The LPBF result items (Hotspot, Recoater Interference, High Strain) in the Postprocessing Options step are an exception to this rule. You can add multiple versions of the same result item in order to set different criteria so the wizard does not track these items.

- **Know the difference between the Exit and the Finish buttons**

  Exit leaves the wizard without applying actions. Finish applies changes to all steps sequentially and then exits the wizard. If you've been using the Apply Changes button for each step along the way, we recommend using the Exit button rather than the Finish button because the Finish button will reapply all changes from each step, including meshing, which may take some time. If you prefer to have the wizard execute all steps simultaneously at the end of the wizard, use the Finish button instead of using Apply Changes along the way.

  Once you start using the wizard, we recommend you complete all the wizard steps.

- **Close the wizard before solving**

  The wizard does not include a solve step. It is best to complete the wizard using either Exit or Finish and then close the wizard window before solving. If you initiate a solve while the wizard is open and then later click Finish to complete the wizard, the existing result data is cleared and the wizard executes all steps simultaneously again.

Assumptions and Known Limitations for Using the Wizard

- The LPBF Setup Wizard, introduced as the default wizard at Release 2023 R2, is not supported on a Linux platform. Linux users will use the Legacy LPBF Setup Wizard instead, which opens by default for Linux users.

- Simulation of advanced items such as bolt pretension and other types of conditions are not available in the wizard and need to be defined in the AM Process Sequence worksheet (p. 100). See Define AM Process Steps (p. 97).

- If multiple mesh sizing objects are created for the build or base prior to opening the wizard, the wizard will use the *first* mesh sizing object in the tree to define the corresponding Build Element Size or Base Element Size.

- Supports can be deleted either all at once or one at a time when using the Delete Support button in the wizard. Deletion of more than one but fewer that all supports can be done outside the wizard. Note that even suppressed supports appear in the Select Support dropdown box and can be edited or deleted although you cannot suppress or unsuppress supports from within the wizard.

- When you click Finish, the wizard may overwrite advanced custom settings, such as complex mesh or contact settings, that you may have set in the project tree while the wizard was open. Conversely, the Read from Tree action does not read advanced custom settings. As a best practice, we recommend that you use the wizard first to set up the majority of your simulation, Finish or Exit from the wizard, and then tweak the advanced settings in the tree as needed. Advanced custom settings refers to detailed options that are not represented in the wizard.

- The LPBF Setup Wizard is not yet fully supported when using the standalone Mechanical application. For example, if you open the Mechanical application independent of Workbench and use the LPBF Setup Wizard to configure a heat treatment with creep properties, the creep properties will be incorrectly configured. Using creep properties from the wizard necessitates the use of Engineering Data *in Workbench*.

- When you use Creep Properties for heat treatment, the wizard automatically unsuppresses the material model in Workbench's Engineering Data, requiring the wizard to refresh the whole Workbench project. During this process, any other upstream updates to the project, including changes to geometry, will be updated.

  Also, when switching from Relaxation Temperature to Creep Properties or vice versa under Properties, you may get this message if Engineering Data was the active tab in Workbench: *"Unexpected error while refreshing pane Toolbox: Object not set to an instance of an object."* It is safe to close the message and ignore the error. This happens only the first time you switch the heat treatment method. On subsequent changes in the same project, the error message should not pop up.

- If you initiate a solve while the wizard is open and then later click Finish to complete the wizard, the existing result data is cleared and the wizard executes all steps simultaneously again.

- Clicking X in the upper, right corner to close the wizard window without first clicking either Exit or Finish leaves AM-related highlights (such as the red color for the part and blue color for the base) on the items in the graphics window. Opening and closing the wizard may not reset this.

- Limitations related to Generated Supports:

  - (Mechanical limitation): Generated Supports are available for multibody scenarios only when the bodies have *shared topology* (p. 13) between them. This means you can use Generated Supports for a multibody part with shared coincident surfaces but not for a multibody part with unshared coincident surfaces, nor for multiple parts separated by space on the build plate.

– (Mechanical limitation): Generated Supports are not available when the part is meshed with the Layered Tetrahedral mesh method.

– (LPBF Setup Wizard limitation): When you click the "Show Mesh" button and then proceed to select element faces for adding multiple instances of Generated Supports, a problem occurs. This interferes with the highlighting functionality. Hence it is recommended not to use the "Show Mesh" button while using the wizard to add additional Generated Supports via the Element Face Selection option.

– (LPBF Setup Wizard limitation): Clicking Finish without having generated a mesh first does not generate the Generated Supports object. Generated Supports depend on an existing mesh of the part so a two-step process is required. The Finish button executes the steps simultaneously. As a best practice, click Apply Changes to establish mesh criteria and then Generate Mesh to establish the part mesh before using Generated Supports.

Closing the Wizard

1. Upon completing the last step, click **Exit** or **Finish** to complete the wizard.

2. Click the **X** in the upper, right corner to close the wizard window.

Note:

Clicking X in the upper, right corner to close the wizard window without first clicking either Exit or Finish leaves AM-related highlights (such as the red color for the part and blue color for the base) on the items in the graphics window. Opening and closing the wizard may not reset this.

If no changes are necessary you are ready to proceed to the solution step (p. 117). Remember that the wizard is only a tool to help set up the simulation. You should review your setup in the project tree before proceeding to solve.

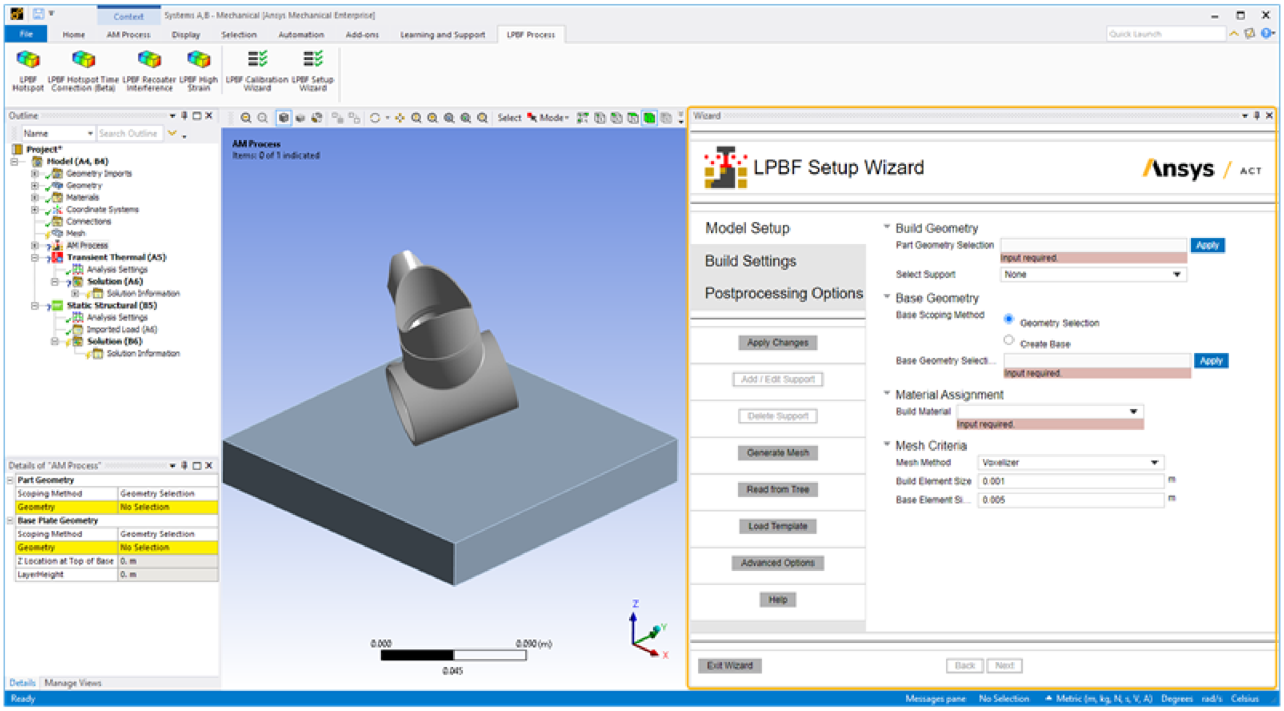
4.1. Wizard Step 1: Model Setup

In the Model setup step:

- Identify geometries of the part(s), supports, and base plate. If you imported a 3MF file, the geometries for part and supports are automatically identified. Use Advanced Options for the simulation of powder or non-build items (such as clamps, bolts, etc.), or if you want to take advantage of symmetry.

- Select your material. Use Advanced Options if the base plate or non-build items are a different material than the build material.

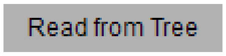
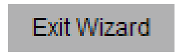
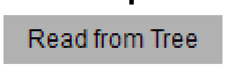
- Set mesh criteria, such as type and sizing.



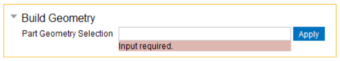
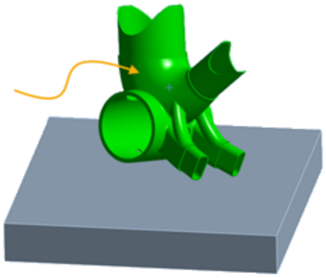
Action buttons are described in the following table.

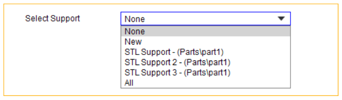
Action Button	Function	Action Button	Function
Apply Changes Apply Changes	Click to apply the actions for this wizard step, which updates the project tree. If you make additional changes, click Apply Changes again. Alternatively, you may choose to fill in the wizard fields for each step without clicking Apply Changes and then click Finish on the last step to execute all actions simultaneously.	**Advanced Options** Advanced Options	Click to toggle on/off options for materials, supports, and advanced geometry items of non build, powder, and symmetry.
Add / Edit Support Add / Edit Support	Click to import the new support(s), or edit the selected support(s), to the project tree. This action is separate from the Apply Changes button to allow the possibility of using multiple folders of support files.	**Help** Help	Click to bring up help for this wizard step.

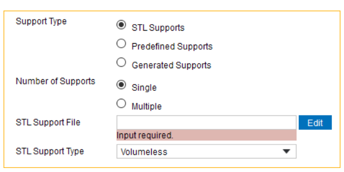
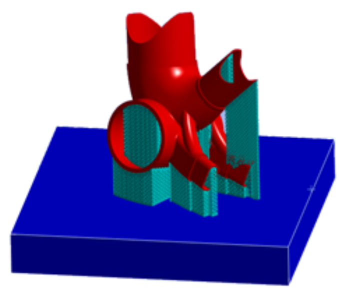
Action Button	Function	Action Button	Function
Delete Support Delete Support	Click to delete the selected support(s) from the project tree.	**Next** Next	Click to move to the next wizard step. All required inputs on this step must be filled in to be able to move to the next step. If the button does not appear blue, not all required inputs are filled in. **No actions are performed when you click Next.** At times, you may want to fill out all the wizard inputs completely before applying any actions. Clicking Next without first clicking Apply Changes allows you to do that.
Generate Mesh Generate Mesh	Click to generate the mesh using the mesh criteria and to generate the appropriate contact connections. **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. This action is separate from the Apply Changes button because meshing may take several minutes for large models. Sometimes users prefer to wait before generating the mesh. If you make	**Back** Back	Not applicable for this step.

Action Button	Function	Action Button	Function
	changes to mesh criteria, click Apply Changes first and then Generate Mesh.		
Read from Tree  	Click to read the status of objects in the project tree and update the wizard input fields accordingly. Think of this as the opposite action of Apply Changes, which updates the objects in the project tree with inputs from the wizard. The status of the tree is read only from certain relevant tree objects. Advanced custom settings, such as complex mesh or contact settings, are not read from the tree and may be overwritten by the wizard upon a subsequent Finish click. It is best to make advanced custom tweaks after exiting from the wizard. See Recommended Workflow and Best Practices (p. 18).	**Exit Wizard**  	Click to exit the wizard. Any actions you have performed using Apply Changes will be maintained in the project tree. No additional actions will be performed upon exit.
Load Template  	Click to load a previously saved template of wizard inputs. You will still need to click Apply Changes or Finish, as usual. The setup items saved on the xml file include materials, mesh settings, build settings, and postprocessing selections. Geometry selections and advanced options involving geometry selections/picks are not included in saved files. Use Save Template in the Postprocessing Options step (p. 59) to save a template.		

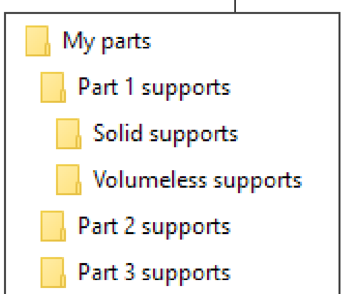
Input fields are described in the following table.

Input Field	Description	Example
Build Geometry: **Part Geometry Selection**  	Select the body representing the part you are manufacturing. Then click Apply. Use Ctrl-click to select multiple bodies. The body should be a closed volume (watertight), and should be oriented with the global Z axis as the build direction. It may be modeled either resting on the base plate (Z=0), or elevated off the base by supports. See additional guidelines (p. 70), including information about topology (p. 13) and symmetry (p. 189). *Affects these tree objects: AM Process, creates BUILD named selection*	
Select Support	Choose **None** if the part is resting	

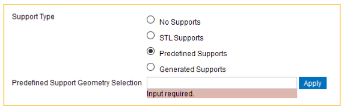
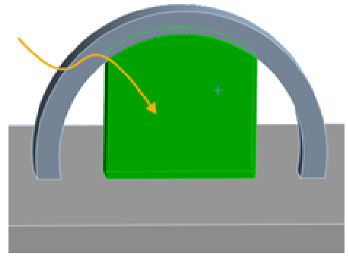
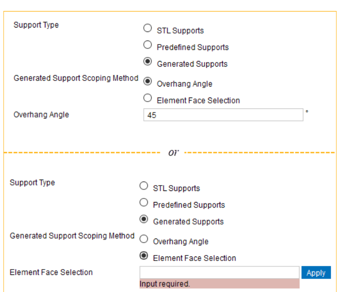
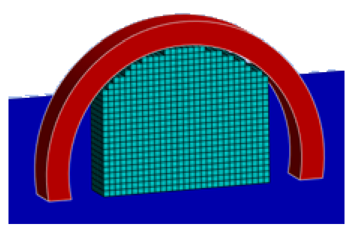
Input Field	Description	Example
Support options shown with some supports already added 	on the base and no other supports are in the model. Choose **New** to add a support, which opens the Support Type options. Choose one individual support, or **All**, followed by clicking the Delete Support button, to delete one or all supports in the model. (Selecting more than one but fewer than all supports is not possible at this time.) When deleting a Predefined Support, the geometry body will remain in the tree but the Predefined Support definition will be deleted. Note that even suppressed supports appear in the selection dropdown	

Input Field	Description	Example
	and can be edited or deleted, although you cannot suppress or unsuppress supports from within the wizard. *Affects these tree objects:* *AM Process, Support Group, STL Support, Predefined Support, Generated Support.*	
Support Type - **STL Supports** 	Choose **STL Supports** if you have support structures saved in .stl files that were not imported along with the part geometry. **Important:** STL support files must be in millimeters. Number of Supports: • Choose **Single** if you have only one	

Input Field	Description	Example
	support file and then click Edit to navigate to the .stl file. • Choose **Multiple** if you have more than one support file and then click Edit to navigate to the *folder* that contains the support files. Support files in a folder must be of one type (either volumeless or solid .stl files) and belong to	

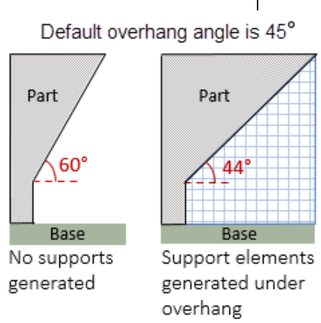
Input Field	Description	Example
	only one part. So for a simulation with multiple parts, multiple supports, and multiple support types, the folder structure might look like this:  Do not include the part in the support folders if it is an .stl file because the application will attempt to	

Input Field	Description	Example
	import all .stl files as supports. STL Support Type: • **Volumeless** supports are usually thin-walled supports created by support generation tools such as Additive Prep. • **Solid** supports are comprised of a solid body. Click Add / Edit Support to import the file(s) and see the supports. See additional information about supports (p. 86). *Affects these tree objects: Support Group, STL Support*	

Input Field	Description	Example
Support Type - **Predefined Supports** 	Choose **Predefined Supports** if you have solid support bodies that were imported with your part geometry. Select the body and click Apply. Use Ctrl-click to select multiple bodies. Click Advanced Options if you want to make material adjustments to the supports. *Affects these tree objects: Predefined Support*	
Support Type - **Generated Supports** 	Choose **Generated Supports** to have the Mechanical application generate supports directly as elements with no associated geometry. Elements are generated straight down from the part mesh by either automatic detection based on the Overhang Angle criterion or from mesh element faces that you select on the part.	

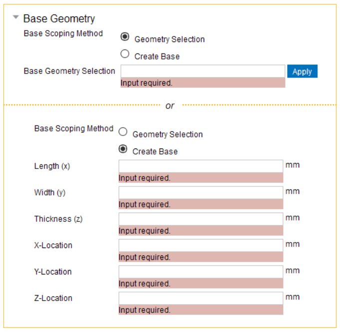
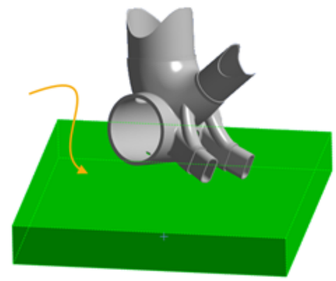
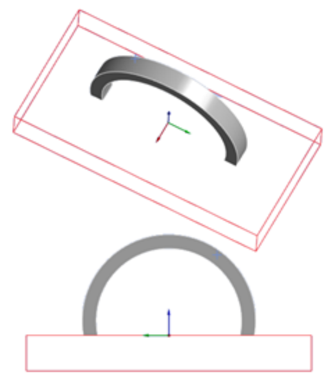
Input Field	Description	Example
	Because this type of support is generated from an *existing mesh*, you first must have a mesh established for the part and base in order to apply a Generated Support. Select the part and base with Support Action set to None, add appropriate mesh criteria, click Apply Changes, then click Generate Mesh. After the basic AM process is set up using the above method, select the Support Action to Add Support and the Support Type to be Generated Support, and then proceed to identify how and where the supports will be generated by choosing either Overhang Angle or	

Input Field	Description	Example
	Element Face Selection. Then click Apply Changes to see the generated elements. **Overhang Angle**: Enter the angle measured from the horizontal base plate (0 degrees) to the surface of the part. Any surface measuring less than the Overhang Angle will be supported. Elements are generated vertically straight down from the overhanging portion of the build to the base, or to a lower portion of the model if it is in the way. Default is 45°.	

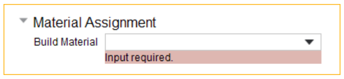
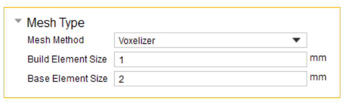
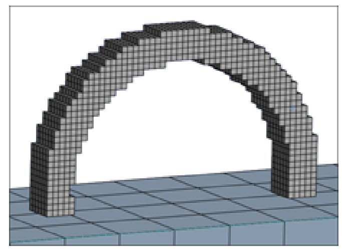


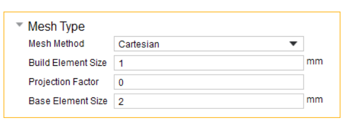
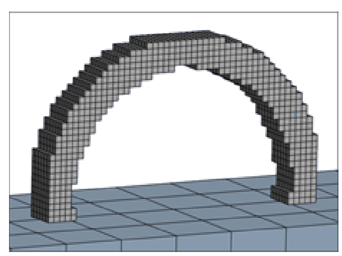
Input Field	Description	Example
	Element Face Selection: In the graphics window, select the element faces of the part mesh from which support elements will be generated straight down. Click and drag to select multiple element faces. Then click Apply in the wizard. Notes: • Generated Supports are available for multibody scenarios only when the bodies have *shared topology* (p. 13) between them. This means you can use Generated Supports for a multibody	

Input Field	Description	Example
	part with shared coincident surfaces but not for a multibody part with unshared coincident surfaces, nor for multiple parts separated by space on the build plate. • Generated Supports are not available when the part is meshed with the Layered Tetrahedral mesh method. *Affects these tree objects:* *Generated Support*	

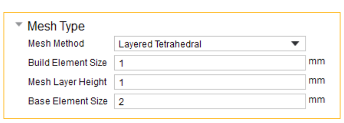
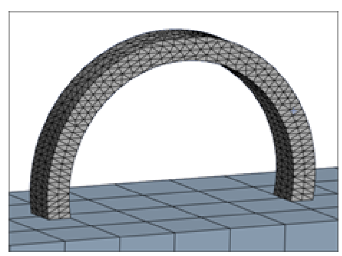
Input Field	Description	Example
Base Geometry 	Select the base body and click Apply. This is the platform on which the build (part plus supports) is to be printed. It is included in the simulation because it acts as a heat sink. Choose **Create Base** if none was included with your part geometry import. Enter overall dimensions of the body and coordinates from the center of the top of the base body. A red outline will guide you as you create the base. Use pan, zoom, and rotate mouse options to view the base preview from all angles. If you choose Create Base and you do not have STL Supports, base sizing is automatic. The wizard generates one that is	Selecting a base  Creating a base 

Input Field	Description	Example
	appropriately sized and positioned for your model. Internally, the application reads the bounding box coordinates of all your build geometries and sizes the base plate to twice the dimensions in the X-Y plane and with an appropriate thickness starting at the minimum Z coordinate. Note that base creation does not take into account any STL Support dimensions so you may have to adjust the minimum Z-location depending on the STL Support. *Affects these tree objects: AM Process, creates BASE named selection, and Construction Geometry if you created a base*	

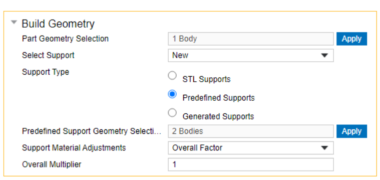
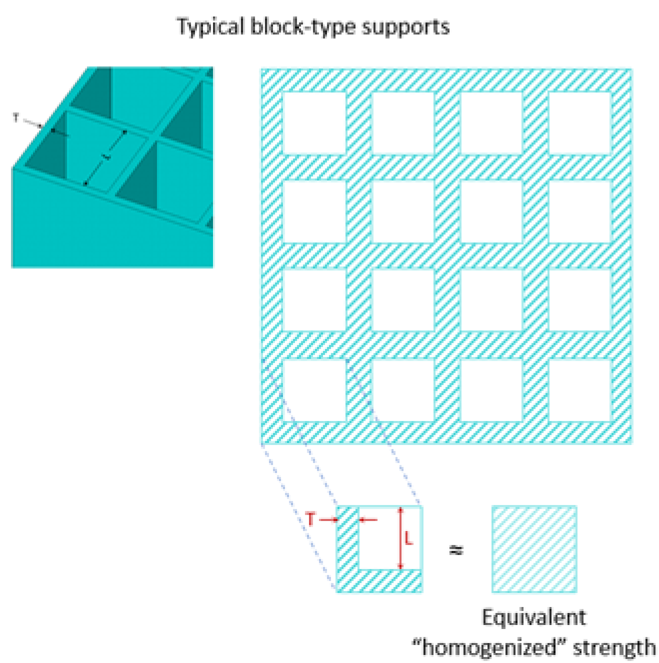
Input Field	Description	Example
Material Assignment 	Select the desired material from the drop-down list of Ansys-supplied sample additive materials. Both the build and the base are assigned the same material by default. Click Advanced Options if you want to designate a different material for the base. By default, nonlinear effects are turned on in the simulation. Click Advanced Options to turn off nonlinear effects. See additional information about materials (p. 69). *Affects these tree objects: Material assignment for each geometry component under Geometry*	
Mesh Criteria - Mesh Method = **Voxel** 	Choose **Voxelizer** to create a mesh of cubic elements for the geometry. Small features, curved surfaces, and horizontal or vertical surfaces that are not multiples of the mesh size are automatically accounted for by a knock-down factor technique. Note: The voxelizer mesh method is not available on a Linux platform. **Build Element Size** is the edge length of the cubic elements of the part and support mesh, which defines the super layer height (p. 7). **Base Element Size** is the edge length of the	

Input Field	Description	Example
	elements of the base mesh. See additional information about mesh controls (p. 78). *Affects these tree objects: Mesh Method renamed to Part Voxelizer, Mesh Sizing renamed to Base Sizing. After mesh generation, creates Build contact element faces and Base contact element faces named selections and Build to Base contact.*	
Mesh Criteria - Mesh Method = **Cartesian**  	Choose **Cartesian** to create a hex mesh that approximates the geometry. Small features, curved surfaces, and horizontal or vertical surfaces that are not multiples of the mesh size are not captured accurately unless a small mesh size is used. **Build Element Size** is the edge	

Input Field	Description	Example
	length of the hex elements of the part and support mesh, which defines the super layer height (p. 7). **Projection Factor** defines how well the mesh will fit to the geometry. A value of 0 (default) results in cubic elements with a rough fit to the geometry. Increasing the Projection Factor will change the shape of the elements to better fit the geometry and may yield better results in some cases but may also result in a failed mesh. Our recommendation is to leave it at 0, or close to 0, initially and then iterate from there to see the effects of changing Projection Factor.	

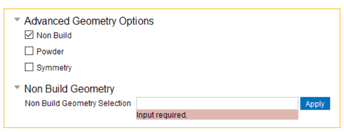
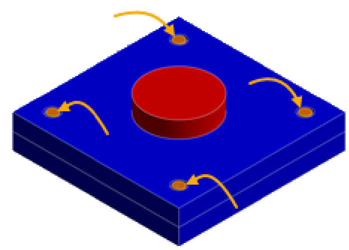
Input Field	Description	Example
	Base Element Size is the edge length of the elements of the base mesh. *Affects these tree objects: Mesh Method renamed to Part Cartesian, Mesh Sizing renamed to Base Sizing. After mesh generation, creates Build contact element faces and Base contact element faces named selections and Build to Base contact.*	
Mesh Criteria - Mesh Method = **Layered Tetrahedral**  	Choose **Layered Tetrahedral** to create an unstructured tetrahedral mesh in layers based on a specified layer height fit to the geometry. It captures the geometry well, and is useful if there are organic curves, small features, such as holes, or thin-walled parts. Note: Generated Supports are not available when using Layered	

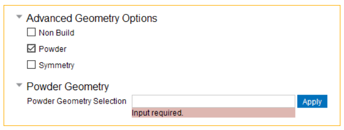
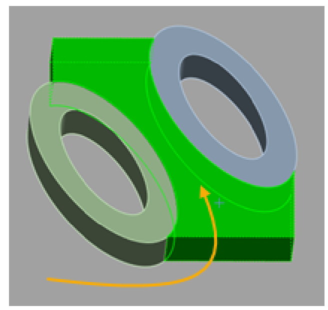
Input Field	Description	Example
	Tetrahedral mesh method. **Build Element Size** is the triangle edge length of the elements of the part and support mesh. It can be greater than the Mesh Layer Height to allow the mesher to produce elements that are skewed with a longer direction parallel to the element layers. **Mesh Layer Height** is the height of each element layer, sometimes referred to as the super layer (p. 7). Actual metal powder deposit layers are aggregated into finite element "super layers" for simulation purposes. **Base Element Size** is the edge	

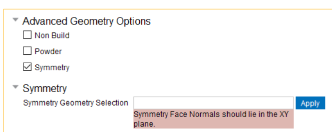
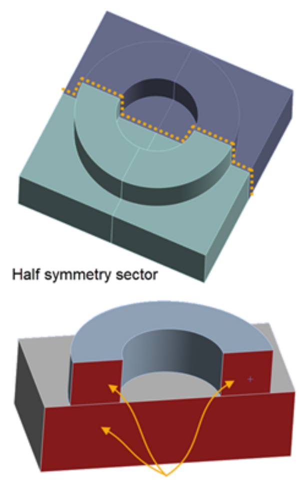
Input Field	Description	Example
	length of the elements of the base mesh. *Affects these tree objects: Mesh Method renamed to Part Layered Tetrahedral, Mesh Sizing renamed to Base Sizing. After mesh generation, creates Build contact element faces and Base contact element faces named selections and Build to Base contact.*	
Advanced Options		
Build Geometry, **Predefined Supports** and **Generated Supports**  	In additive machines, supports are printed with the same material as the part but as thin walled structures with less mass than the part. In the simulation we model the supports as an equivalent "homogenized" solid rather than as thin-walled structures. Their properties will be scaled down to account for this homogenization	

Input Field	Description	Example
	technique. Affected properties are elastic modulus, shear modulus, yield strength, density, and thermal conductivity. Select from one of the following methods for Support Material Adjustments for the Predefined/Generated Supports: • **Overall Factor** is the ratio of the actual support area to the area of the part area. For example an Overall Multiplier of 0.33 will adjust the properties of the supports to be a third of that	

Input Field	Description	Example
	of the part material. • **Block Support Dimensions** allows you to modify Block Wall Thickness and Block Wall Spacing for the supports. • **Property Specific Factors** allows you to modify property-by-property multipliers. These multipliers are the ratio of geometric areas of the supports to the areas of the part geometry, where the areas are the projected areas	

Input Field	Description	Example
	in the X, Y, and Z directions. These properties can be changed after a support is added by changing the values and clicking Add / Edit Support.	
Advanced Geometry Options, **Non Build** 	Choose **Non Build** to simulate geometry items that are present on the base plate but that are not being 3D-printed. These items may include clamps, bolts, measuring devices, instrumentation, etc. Select the body representing the non build component. Use Ctrl-click to select multiple bodies. Selecting the Non Build option will also make the field for the Non Build Element Size visible under Mesh Criteria. *Affects these tree objects:*	

Input Field	Description	Example
	Will create a named selection for non build elements, and a mesh sizing object for the selected non build geometry	
Advanced Geometry Options, **Powder** 	Choose **Powder** to select a geometry body that represents unmelted powder. Modeling the powder as elements may be useful if you are simulating multiple parts close together on the build plate or if the part has features close together, where accounting for the heat transfer occurring between the parts or features is important. Select the body representing the powder and click Apply. Use Ctrl-click to select multiple bodies. The mesh size for this will be the same as the build mesh size in order to have	

Input Field	Description	Example
	the same layer height. See additional information about modeling powder (p. 194). *Affects these tree objects: Will add a powder elements named selection for the selected powder geometry.*	
Advanced Geometry Options, **Symmetry**  	Choose **Symmetry** if your model has symmetrical geometry—specifically, if it is symmetric with respect to the part, supports, and build plate, *and* it has symmetric loading and boundary conditions. Simulating a sector of a large model will result in a faster simulation time while using fewer resources. Select the faces representing the symmetry plane and click Apply. Use Ctrl-click to select multiple faces.	

Input Field	Description	Example
	See additional information about modeling symmetry (p. 189). *Affects these tree objects: Will add a SYMM_NODES named selection and a Symmetry Region object*	

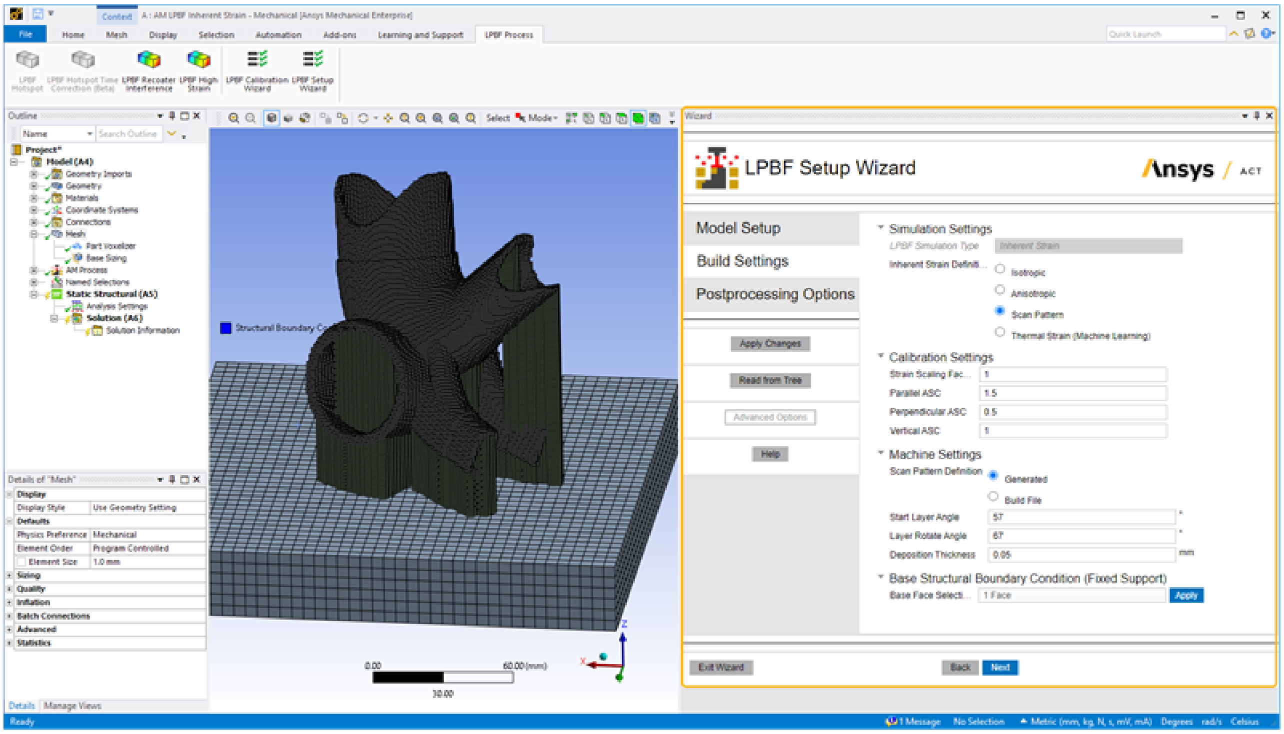
The next step of the wizard is Build Settings. The procedure varies depending on whether you have a Thermal-Structural system or an Inherent Strain system:

- Wizard Step 2: Build Settings (for Inherent Strain Simulations) (p. 49)
- Wizard Step 2: Build Settings (for Thermal-Structural Simulations) (p. 53)

4.2. Wizard Step 2: Build Settings (for Inherent Strain Simulations)

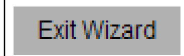
In the Build Settings step:

- Simulation assumptions are set by default based on your chosen AM LPBF system. Choose the Inherent Strain Definition.
- Enter calibration factor(s) as determined from calibration experiments (p. 199). Use default calibration factors if your goal is simply to examine *trends*, that is, the effects of variable changes on stress or distortion *relative to each other*.
- Enter machine settings, which are process parameters that directly influence how the process deposits material. These can include Deposition Thickness, Hatch Spacing, Scan Speed, and a number of other factors.
- Enter build conditions, which are settings pertaining to the environment *around the build* during the deposition process and during cooldown. This includes Preheat Temperature.

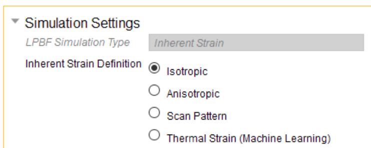


Action buttons are described in the following table.

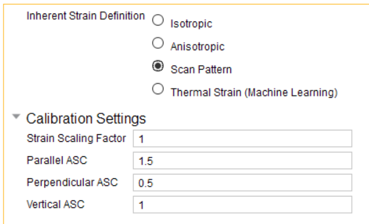
Action Button	Function	Action Button	Function
Apply Changes Apply Changes	Click to apply the actions for this wizard step, which updates the project tree. If you make changes, click Apply Changes again.	**Help** Help	Click to bring up help for this wizard step.
Read from Tree Read from Tree	Click to read the status of objects in the project tree and update the wizard input fields accordingly. Think of this as the opposite action of Apply Changes, which updates the objects in the project tree with inputs from the wizard.	**Next** Next	Click to move to the next wizard step. All required inputs on this step must be filled in to be able to move to the next step. If the button does not appear blue, not all required inputs are filled in. **No actions are performed when you click Next.** At times, you may want to fill out all the wizard inputs completely before applying any actions. Clicking Next without first clicking Apply Changes allows you to do that.
Advanced Options Advanced Options	No advanced options for this step.	**Back** Back	Click to go back to the previous wizard step.

Action Button	Function	Action Button	Function
		Exit Wizard  	Click to exit the wizard. Any actions you have performed using Apply Changes will be maintained in the project tree. No additional actions will be performed upon exit.

Input fields are described in the following table.

Input Fields	Description
Simulation Settings  	Choose one of the following strain definitions, which reflect how inherent strain is calculated as an input to the structural solver: • **Isotropic** assumes that a constant, uniform strain occurs at every location within a part as it is being built. This is the simplest assumption, resulting in the shortest simulation time. • **Anisotropic** subdivides the strain into anisotropic components in the X, Y, and Z directions based on the Global coordinate system. • **Scan Pattern** subdivides strain into anisotropic components based on the local orientation of scan vectors within the part. Scan vectors are generated internally via a slicing function assuming a rotating stripe scan pattern or input directly from a build file. • **Thermal Strain (Machine Learning)**, using a Machine Learning prediction, is a method that provides the highest level of fidelity and takes thermal cycling into account at each location within the part. As with Scan Pattern, scan vectors are generated internally via a slicing function assuming a rotating stripe scan pattern or input directly from a build file. **Important:** The Machine Learning Prediction feature requires a Structures AI+ license. **Machine Learning Model** is a list of materials that were used to train the ML prediction, in particular, the materials validated for the Additive application. Choose the material that most closely matches your material assignment in Engineering Data. ML models may be based on different material properties than those in Engineering Data. The ML models are used to generate loading strains. Materials in Engineering Data are used for the structural analysis. *Affects these tree objects: Build Settings*
Calibration Settings	Enter strain scaling factors. These are optional inputs that scale the inherent strains by a given value. They are usually determined from calibration

Calibration settings for Scan Pattern

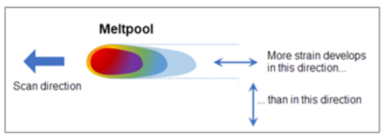


experiments (p. 199) and account for differences in materials, machines, and other factors.

- Isotropic allows one **Strain Scaling Factor** reflecting uniform scaling in all directions. It defaults to 1.

- Anisotropic allows a strain scaling factor for each global coordinate direction. These default to 1.

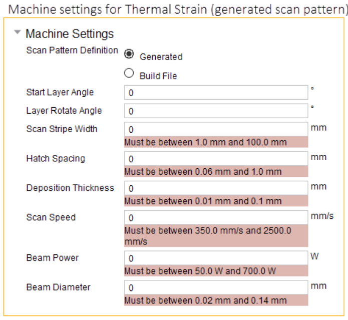
- Scan Pattern and Thermal Strain allow strain scaling factors, called Anisotropic Scaling Factors, parallel and perpendicular to the scanning direction in the X-Y plane, and one that is vertical in the build direction. The default ASCs take into account the fact that more strain develops in the scan direction than perpendicular to it, with Parallel ASC = 1.5, Perpendicular ASC = 0.5, and Vertical = 1. You can also enter a uniform Strain Scaling Factor, which defaults to 1.



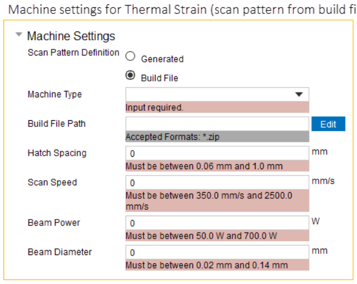
Affects these tree objects: Build Settings

Machine Settings

Machine settings for Thermal Strain (generated scan pattern)



Machine settings for Thermal Strain (scan pattern from build file)



Enter the Machine Settings, which are settings and process parameters that directly influence how the process deposits material. Every simulation except isotropic uses some, or all, of these parameters:

- **Scan Pattern Definition** is how the scan pattern is defined, either generated using a rotating stripe pattern (default) or input via a build file.

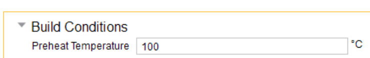
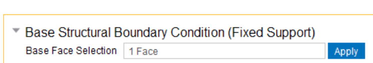
  Build File Path is the location of a .zip file containing the scan pattern file(s), and an stl of the part geometry. See Build File Requirements (p. 109).

  Machine Type identifies the machine or OEM associated with the specified build file. Options are Additive Industries, EOS, HB3D, Renishaw, Sisma, SLM, and Trumpf.

- **Start Layer Angle** is the orientation of fill rasters on the first layer of the build. It is measured from the X axis, such that 0 degrees results in scan lines parallel to the X axis. The starting layer angle is commonly set to 57 degrees. Must be between 0 and 180°.

- **Layer Rotation Angle** is the angle at which the major scan vector orientation changes from layer to layer. It is commonly 67 degrees. Must be between 0 and 180°.

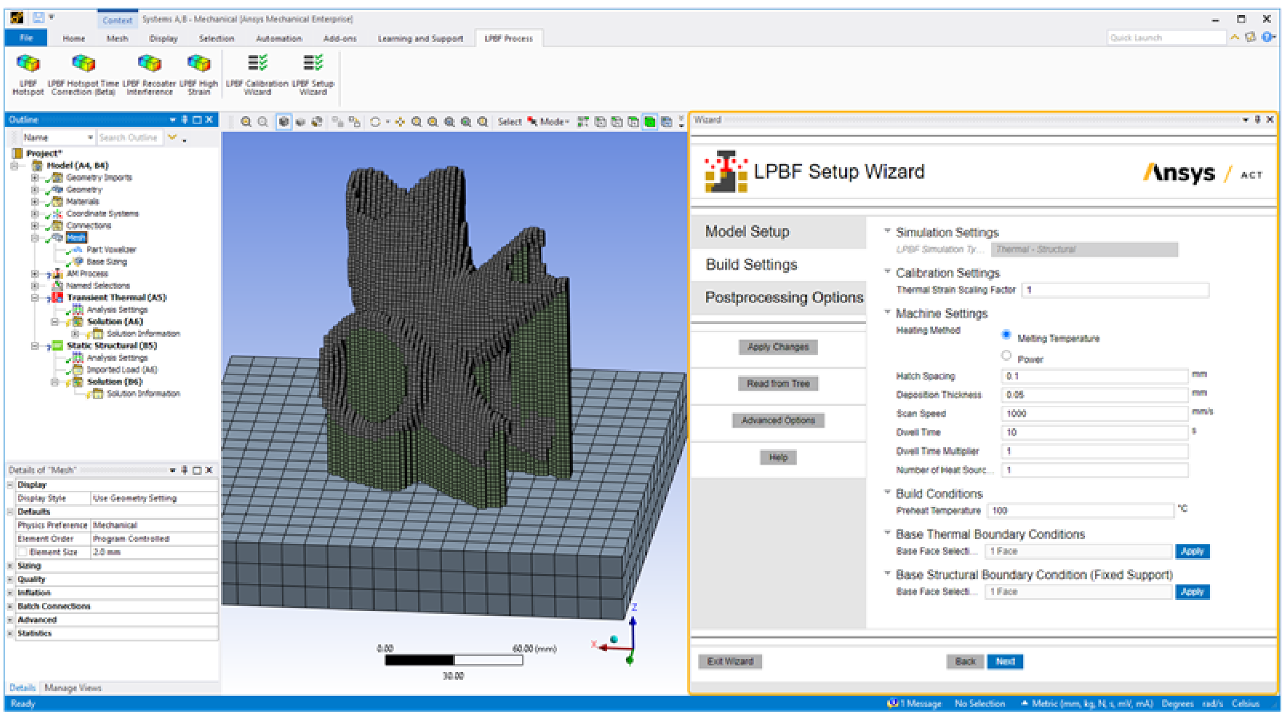
- **Scan Stripe Width** is the width of the sections, called stripes, into which the geometry is sliced. The stripes are scanned sequentially to break up what would otherwise be very long continuous scan

	vectors. Slicing Stripe Width is typically set to 10 mm wide. Must be between 1 and 100 mm. • **Hatch Spacing** is the average distance between adjacent scan vectors when rastering back and forth with the laser. Hatch spacing should allow for a slight overlap of scan vector tracks such that some of the material re-melts to ensure full coverage of solid material. For Thermal Strain (Machine Learning) strain definition, must be between 60 and 1000 microns. • **Deposition Thickness** is the thickness of deposited material in every pass of the recoater blade. Specifically, use the amount the base plate drops between layers. • **Scan Speed** is the average speed at which the laser spot moves across the powder bed along a scan vector to melt material, excluding jump speeds and ramp up and down speeds. For Thermal Strain strain definition, must be between 350 and 2500 mm/sec and the recommended range is between 500 and 2500 mm/sec. • **Beam Power** is the power setting for the laser in the machine. Must be between 50 and 700 Watts. The recommended range is between 50 and 500 Watts. • **Beam Diameter** is the width of the laser on the powder or substrate surface defined using the D4σ beam diameter definition. Usually this value is provided by the machine manufacturer. Sometimes called laser spot diameter. Must be between 20 and 140 µm. The recommended range is between 80 and 120 µm. *Affects these tree objects: Build Settings*
Build Conditions 	Enter Build Conditions, which are settings pertaining to the environment around the build during the deposition process. • **Preheat Temperature** is the starting temperature of the build plate. Used when Inherent Strain Definition = Thermal Strain. Must be between 20 and 500 °C and the recommended range is between 20 and 200 °C. *Affects these tree objects: Build Settings*
Base Structural Boundary Conditions 	Select the face of the base where the fixed support should be applied, usually the underside of the base. Then click Apply. Use Ctrl-click to select multiple faces. By default, the underside surface of the base is already selected. *Affects these tree objects: Fixed boundary condition under Static Structural system*

4.3. Wizard Step 2: Build Settings (for Thermal-Structural Simulations)

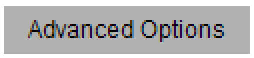
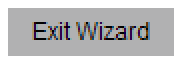
In the Build Settings step:

- Simulation assumptions are set by default based on your chosen AM LPBF system.
- Enter calibration factor(s) as determined from calibration experiments (p. 199). Use default calibration factors if your goal is simply to examine *trends*, that is, the effects of variable changes on stress or distortion *relative to each other*.
- Enter machine settings, which are process parameters that directly influence how the process deposits material. These include Deposition Thickness, Hatch Spacing, Scan Speed, and a number of other factors.
- Enter build and cooldown conditions, which are settings pertaining to the environment *around the part* during the deposition process and during cooldown. These include Preheat Temperature and gas and powder boundary conditions.
- Enter boundary conditions applied on the specified *surface of the base*, usually the underside surface, during the build and during cooldown. The wizard automatically selects the underside surface of the base by default.

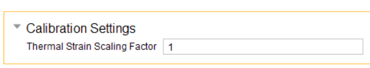
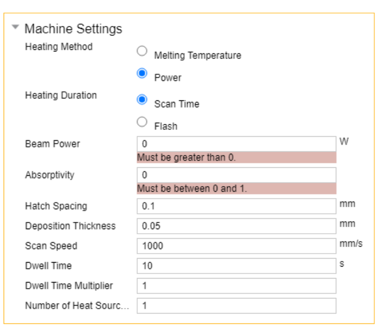


Action buttons are described in the following table.

Action Button	Function	Action Button	Function
Apply Changes Apply Changes	Click to apply the actions for this wizard step, which updates the project tree. If you make changes, click Apply Changes again.	**Help** Help	Click to bring up help for this wizard step.
Read from Tree Read from Tree	Click to read the status of objects in the project tree and update the wizard input fields accordingly. Think of this as the opposite	**Next** Next	Click to move to the next wizard step. All required inputs on this step must be filled in to be able to move to the next step. If the

Action Button	Function	Action Button	Function
	action of Apply Changes, which updates the objects in the project tree with inputs from the wizard.		button does not appear blue, not all required inputs are filled in. **No actions are performed when you click Next.** At times, you may want to fill out all the wizard inputs completely before applying any actions. Clicking Next without first clicking Apply Changes allows you to do that.
Advanced Options 	Click to toggle on/off options related to gas and powder convection.	**Back** 	Click to go back to the previous wizard step.
		Exit Wizard 	Click to exit the wizard. Any actions you have performed using Apply Changes will be maintained in the project tree. No additional actions will be performed upon exit.

Input fields are described in the following table.

Input Fields	Description
Calibration Settings 	Enter Thermal Strain Scaling Factor. This is an optional input that scales the thermal strains in the structural portion of a Thermal-Structural simulation by a given value. It is usually determined from calibration experiments (p. 199) to account for differences in materials, machines, and other factors. *Affects these tree objects: Build Settings*
Machine Settings 	Enter the Machine Settings, which are settings and process parameters that directly influence how the process deposits material. • **Heating Method** is the simulation assumption related to how the material is heated for each new layer as it is added (made alive). Choose either Melting Temperature (default) or Power. For Power, additional inputs are required: – **Heating Duration** is the amount of time the heat is applied, either Scan Time (default) or Flash. → **Scan Time** means that heat is applied for the amount of time it takes to scan the volume of material in each element layer. This setting may give better temperature results at the end of the layer but may

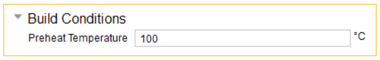
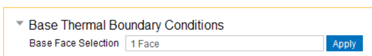
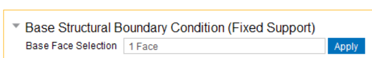
not yield a temperature spike above melting. Note that this Scan Time option will have a different end time of the simulation because the layer thickness adjustment for cooling will not be used.

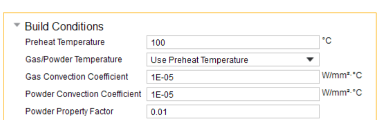
→ **Flash** means that heat is applied in a very short time increment resulting in a spike in temperature then cooling before the next element layer is added.

- **Beam Power** is the power of the laser.
- **Absorptivity** is a fraction that describes how much energy is absorbed by the material and contributes to the heating process. Value must be between 0 and 1.

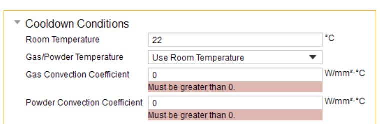
- **Hatch Spacing** is the average distance between adjacent scan vectors when rastering back and forth with the laser. Hatch spacing should allow for a slight overlap of scan vector tracks such that some of the material re-melts to ensure full coverage of solid material.
- **Deposition Thickness** is the thickness of deposited material in every pass of the recoater blade. Specifically, use the amount the base plate drops between layers.
- **Scan Speed** is the average speed at which the laser spot moves across the powder bed along a scan vector to melt material, excluding jump speeds and ramp up and down speeds.
- **Dwell Time** is the span of time from the end of the laser scan of one layer to the start of the laser scan of the next layer. It includes the time required for recoater-blade repositioning and powder-layer spreading as well as deposition of other parts that may be part of the build but are not being simulated.
- **Dwell Time Multiplier** multiplies the scan time by the specified number to represent identical parts built at the same time, but not present in the simulation. If they are the same part arranged in the same orientation on the build plate, the multiplier is the number of parts.

  In the case of symmetry, the Dwell Time Multiplier is required to reconcile the reduced time to simulate the build and should be equal to the number of repetitions (or sectors). For example, the build time for a half symmetry model is ½ of the build time of the full model, so the Dwell Time Multiple should be 2. Similarly, for a quarter symmetry model, enter 4 for Dwell Time Multiplier, and so on.
- **Number of Heat Sources** is the number of lasers if using multiple-beam printers. This divides the amount of time it takes to scan a layer by the number of heat sources specified.

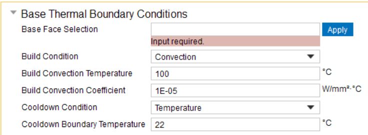
	Affects these tree objects: Build Settings
Build Conditions 	Enter Build Conditions, which are settings pertaining to the environment around the build during the deposition process. In LPBF, heat loss to the powder is simulated as a convection boundary condition unless it is explicitly modeled as a body (see Advanced Options in the Model Setup step (p. 43)). By default, these conditions will be added as thermal boundary conditions with the default values for temperature and convection coefficients set by the wizard. Convection coefficients default to 10 W/(m)^2(°C). Use Advanced Options (p. 57) to customize build conditions. • **Preheat Temperature** is the starting temperature of the build plate. *Affects these tree objects: Build Settings*
Base Thermal Boundary Conditions 	Select the face of the base where preheat temperature should be applied, usually the underside of the base. Then click Apply. Use Ctrl-click to select multiple faces. By default, the underside surface of the base is already selected. *Affects these tree objects: Thermal boundary conditions and loads under Transient Thermal system*
Base Structural Boundary Conditions 	Select the face of the base where the fixed support should be applied, usually the underside of the base. Then click Apply. Use Ctrl-click to select multiple faces. By default, the underside surface of the base is already selected. *Affects these tree objects: Fixed boundary condition under Static Structural system*
Advanced Options	

Build Conditions

	Enter details about the Build Conditions, which are settings pertaining to the environment around the build during the deposition process. In LPBF, heat loss to the powder is simulated as a convection boundary condition unless it is explicitly modeled as a body (p. 194).
	
	Convection coefficients default to 10 W/(m)^2(°C).
	
	• **Preheat Temperature** is the starting temperature of the build plate.
	
	• **Gas/Powder Temperature** setting allows you to use either Preheat Temperature or user-specified.
	  – Gas Temperature is the temperature in the chamber during the build.
	  – Powder Temperature is the temperature of the powder surrounding the part.

	• **Gas Convection Coefficient** is the convection coefficient from the build to the enclosure gas. The convection is applied only to the top of a newly laid layer. • **Powder Convection Coefficient** is the effective convection coefficient from the sides of the build to the powder bed. To estimate, divide the conduction property of the powder by a characteristic conduction length into the powder (for example, a quarter of the distance from the build boundary to the build-chamber wall). • **Powder Property Factor** is a knockdown factor used to estimate the powder properties. The Mechanical application applies the factor to the solid material properties to estimate the properties of the material in its powder state. The powder-state properties are used in the newly added layer during the heating of the new layer (before its subsequent solidification and cooldown) prior to the next layer being applied. The default value is 0.01. This powder knockdown factor is also used if powder is explicitly modeled (p. 194) in the build. *Affects these tree objects: Thermal boundary conditions and loads under Transient Thermal system*

Cooldown Conditions
 	

	Enter details about Cooldown Conditions, which are settings pertaining to the environment in the build chamber around the build in the cooldown step after the build is completed.
	
	• **Room Temperature** is the ambient room temperature.
	
	• **Gas/Powder Temperature** setting allows you to use either Room Temperature or user-specified.
	
	  – Gas Temperature is the temperature in the chamber during the cooldown.
	
	  – Powder Temperature is the temperature of the powder during the cooldown.
	
	• **Gas Convection Coefficient**: Same definition as in build conditions.
	
	• **Powder Convection Coefficient**: Same definition as in build conditions.
	
	Affects these tree objects: Thermal boundary conditions and loads under Transient Thermal system

Base Thermal Boundary Conditions	Enter details about the thermal boundary conditions applied on the specified surface of the base, usually the underside surface, during the

<table>
<tr>
<td>


</td>
<td>
build and during cooldown. Boundary condition options include temperature, convection, or an adiabatic condition (no heat transfer).

Base Face Selection: Select the face of the base where preheat temperature should be applied, usually the underside of the base. Then click Apply. Use Ctrl-click to select multiple faces. By default, the underside surface of the base is already selected.
Build Condition is the boundary condition applied to the selected surface during the build, either Temperature, Convection, or Adiabatic (no heat transfer).

Build Boundary Temperature (default): Temperature applied on the specified surface of the base during the build.
Build Convection Temperature: Temperature for convection applied on the specified surface of the base during the build.
Build Convection Coefficient: Convection coefficient representing heat loss through the bottom of the base to the machine during the build.

Cooldown Condition is the boundary condition applied to the selected surface during the cooldown, either Temperature, Convection, or Adiabatic (no heat transfer):

Cooldown Boundary Temperature: Temperature applied on the specified surface of the base during the cooldown.
Cooldown Convection Temperature (default): Temperature for convection applied on the specified surface of the base during the cooldown.
Cooldown Convection Coefficient (default): Convection coefficient representing heat loss through the bottom of the base to the machine during the cooldown.

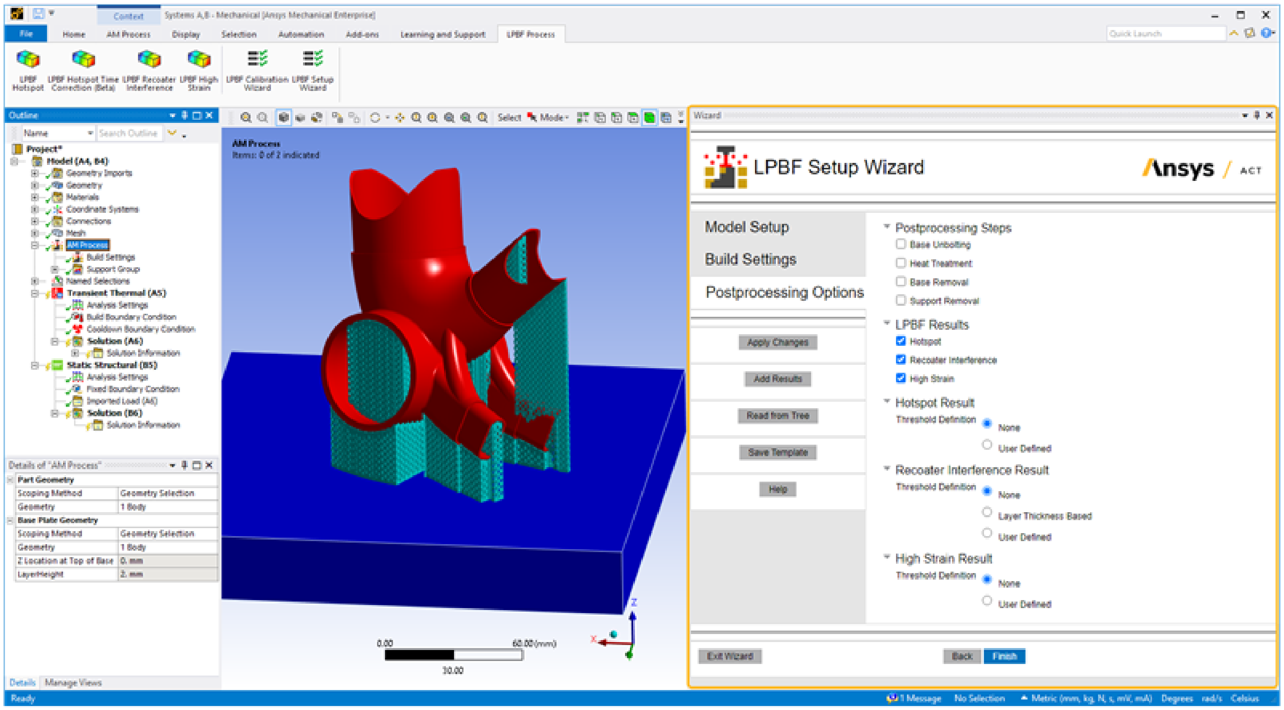
Affects these tree objects: Thermal boundary conditions and loads under Transient Thermal system
</td>
</tr>
</table>

4.4. Wizard Step 3: Postprocessing Options

In the Postprocessing Options step:

- Select the post-build processing steps you want to simulate, such as base unbolting, heat treatment, and base and/or support removal. The default sequence for post processing will be base unbolting, heat treatment, support removal, and then base removal at the end, depending on which options are chosen. In the case of multiple supports, removal will be in the order that the supports were added. To reorder how the steps will be performed in the simulation, exit or finish out of the wizard and then modify the sequence directly in Mechanical using the AM Process Sequence worksheet (p. 97).

- Click Add Results to add standard result items to the project tree, specifically Temperature for a thermal system and Total Deformation, Equivalent Stress, and Equivalent Total Strain for the structural system. Also, select LPBF result items to be calculated during solution, such as Hotspot, Recoater Interference, and High Strain. Specify criteria for warning and critical thresholds for these LPBF result items, or add multiple versions with different criteria for comparison.



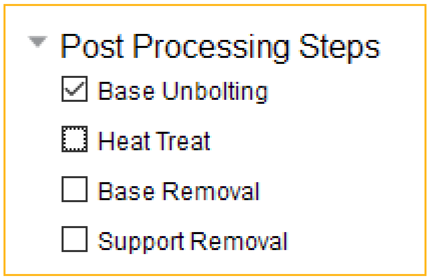
Action buttons are described in the following table.

Action Button	Function	Action Button	Function
Apply Changes Apply Changes	Click to apply the actions for this wizard step, which updates the project tree. If you make changes, click Apply Changes again.	**Help** Help	Click to bring up help for this wizard step.
Add Results Add Results	Click to add standard result items to the tree, specifically Temperature (for a thermal system) and Total Deformation, Equivalent Stress, and Equivalent Total Strain (for the structural system), *and* the specified LPBF result items, specifically Hotspot, Recoater Interference, and High Strain. Click Add Results again to add *multiple* LPBF result items, perhaps with different criteria defined for each.	**Back** Back	Click to go back to the previous wizard step.

Action Button	Function	Action Button	Function
	Note that if the standard result items mentioned above are already in the project tree, new ones will not be added by the wizard. However, if LPBF-specific result items are already in the project tree, new ones will be added by the wizard.		
Read from Tree Read from Tree	Click to read the status of objects in the project tree and update the wizard input fields accordingly. Think of this as the opposite action of Apply Changes, which updates the objects in the project tree with inputs from the wizard. For this step of the wizard, only objects in the tree related to the sequencer (p. 100) are read.	**Finish** Finish	Click to apply changes to all steps sequentially and then exit the wizard. (This includes mesh generation even if you may have already generated a mesh.)
Save Template Save Template	Click to save wizard inputs as a template. Navigate to the desired folder, enter a file name, and click Save. The file is saved as an xml file. The items saved on the xml file include materials, mesh settings, build settings, and postprocessing options. Geometry selections and advanced options involving geometry selections/picks are not included in saved templates. Use Load Template in the Model Setup step (p. 21) to use a template in subsequent simulations.	**Exit Wizard** Exit Wizard	Click to exit the wizard. Any actions you have performed using Apply Changes will be maintained in the project tree. No additional actions will be performed upon exit.

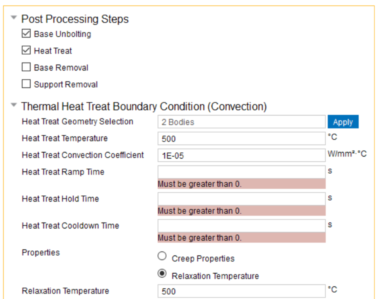
Input fields are described in the following table.

Input Fields	Description
Post Processing Steps: **Base Unbolting**	Choose Base Unbolting to simulate the base plate being unbolted from the machine after the build and cooldown processes are

<table>
<tr>
<td>


</td>
<td>
complete. Specifically, the constraints for UX, UY, and UZ will be removed from the base plate nodes and weak springs are turned on in Analysis Settings. Unbolting the base prior to heat treatment can be especially important to prevent any artificial stress in the base due to fixed boundary conditions and thermal expansion during heating.

Affects these tree objects: AM Process Sequence worksheet (p. 100), Analysis Settings
</td>
</tr>
<tr>
<td>
Post Processing Steps: **Heat Treat**



</td>
<td>
Choose Heat Treat to simulate a heat treatment process, such as annealing, to relieve residual stresses. In an annealing process, metal is heated in a furnace to a particular high temperature, held there for a long time (hours or days), and then allowed to cool slowly.

- **Heat Treat Geometry Selection** Select all the bodies in the model, including the base plate. By default, all bodies are selected for you. (Note: Any STL Supports present in the model are selected for heat treatment although they are not included in the body count.)
- **Heat Treat Temperature** is the temperature for the isothermal hold.
- **Heat Treat Convection Coefficient** is the convection coefficient of the gas in the furnace.
- **Heat Treat Ramp Time** is the time, in seconds, to ramp up to the Heat Treat Temperature.
- **Heat Treat Hold Time** is the isothermal temperature duration, in seconds.
- **Heat Treat Cooldown Time** is the time, in seconds, allowed for cooldown.

Choose one of the following properties as the mechanism to account for the stress relaxation:

- **Creep Properties** uses a creep model as the mechanism for stress relief. When selecting this option, and upon clicking Apply Changes or Finish:
  - The creep model for your chosen material is automatically unsuppressed (p. 179) in Workbench's Engineering Data.
</td>
</tr>
</table>

<table>
<tr>
<td></td>
<td>
– The units used during solution will automatically switch to the unit system required for the creep model.

Important:

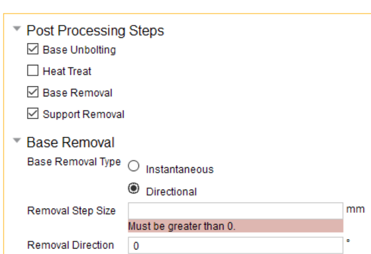
Be aware that changes in Engineering Data, such as suppressing/unsuppressing the creep material model, causes the wizard to refresh the whole Workbench project. Any other upstream changes made to the project, including changes to geometry, will be updated during this process.

• Relaxation Temperature is the temperature at which strains begin to soften. It is lower than the melting temperature. Using the Relaxation Temperature option is a simplified approach resulting in an abrupt stress relaxation.

Note:

We strongly recommend you also choose the Base Unbolting option if you choose Heat Treatment. Unbolting the base prior to heat treatment can be especially important to prevent any artificial stress in the base due to fixed boundary conditions and thermal expansion during heating.

Affects these tree objects: AM Process Sequence worksheet (p. 100), Analysis Settings, and Imported Load under Static Structural system. Adds a Transient Thermal system with all its child objects.
</td>
</tr>
<tr>
<td>
Post Processing Steps: Base Removal



</td>
<td>
Choose Base Removal to simulate the removal of the build from the base.

Base Removal Type determines how the base is removed from the analysis at the end of cooldown:

• Instantaneous simulates instantaneous cutoff at the base (bottom layer of elements only).

• Directional simulates a progressive cutoff at the base (bottom layer of elements only), in which you specify the distance of each cut increment and the angular direction for removal from the base.

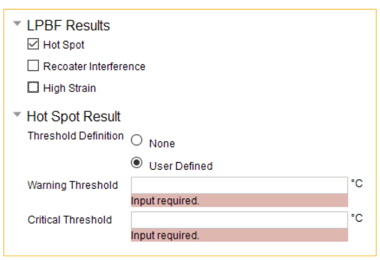
– Removal Step Size is the distance removed in each cut step. Cannot be 0 or a negative number.
</td>
</tr>
</table>

<table>
<tr>
<td></td>
<td>
– Removal Direction is the directional cutoff angle on the X-Y plane as measured from the +X axis. For example, a value of 90 results in a cutoff direction in the +Y direction.

Affects these tree objects: AM Process Sequence worksheet (p. 100) and Analysis Settings under Static Structural system
</td>
</tr>
<tr>
<td>
Post Processing Steps: Support Removal

</td>
<td>
Choose Support Removal to simulate the instantaneous removal of the supports from the part bodies. In the case of multiple supports, removal will be in the order that the supports were added.

Affects these tree objects: AM Process Sequence worksheet (p. 100) and Analysis Settings under Static Structural system
</td>
</tr>
<tr>
<td>
LPBF Results: Hotspot



</td>
<td>
Choose Hotspot to add an LPBF Hotspot result item to your project so that the appropriate data is written out during solution. Available only for simulations with thermal solves (LPBF Thermal-Structural simulation). The Hotspot result tool is used to identify areas of overheating that may result in problematic thermal conditions.

Threshold Definition: Either None or User Defined:

• None (default): The result tool will show the temperature of each layer right before the next layer is added. The most concerning hotspots are going to be the areas with the highest temperature for that layer. Results are localized (based on nodal values), not averaged across the layer. Use a section plane to reveal hotspots inside the part.

• User Defined: You define your own warning and critical thresholds:

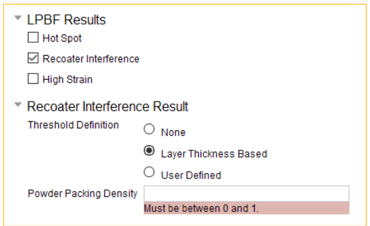
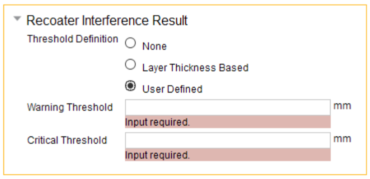
– Warning Threshold: Temperature above which results are considered a warning.

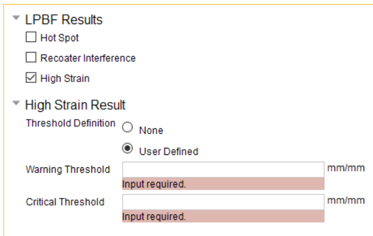
– Critical Threshold: Temperature above which results are considered critical.

Important:

The data is based on the temperature that the build cools down to at the end of each layer right before the next layer is added. It does not represent the "final state" of temperatures at the end of the cooldown step.

For more information about interpreting results after the simulation is solved, see Review Results (p. 123).
</td>
</tr>
</table>

	Affects these tree objects: LPBF Hotspot result item
LPBF Results: **Recoater Interference**  	Choose **Recoater Interference** to add an LPBF Recoater Interference result item to your project so that the appropriate data is written out during solution. The Recoater Interference result specifies the level of Z-deformation at which the build may interfere with the recoater's spreading. **Threshold Definition**: Either None, Layer Thickness Based, or User Defined: • **None** (default): The result tool will show pure Z-deformation in the build, where the value at each point corresponds to the Z-deformation when that material was a new layer. • **Layer Thickness Based**: The result tool uses the Deposition Thickness and Powder Packing Density to determine the thresholds for warning and critical deformation. Specifically, if the Z-deformation for any layer is equal to or greater than the Deposition Thickness it is considered a warning. If the Z-deformation is equal to or exceeds the value of Deposition Thickness/Powder Packing Density, it is considered critical. – **Powder Packing Density**: May be dependent on the powder particle size, material, spreading mechanism, or other factors. • **User Defined**: You define your own warning and critical thresholds: – **Warning Threshold**: Z deformation above which results are considered a warning. – **Critical Threshold**: Z deformation above which results are considered critical. **Important:** *The data is based on the Z deformation of each layer right before the next layer is added*. It does not represent the "final state" of deformation at the end of the cooldown step. For more information about interpreting results after the simulation is solved, see Review Results (p. 123). *Affects these tree objects: LPBF Recoater Interference result item*
LPBF Results: **High Strain**	Choose **High Strain** to add an LPBF High Strain result item to your project so that the appropriate data is written out during solution.

	The High Strain result shows the maximum equivalent strain experienced during the build process. It can help to identify regions at risk of cracking. **Threshold Definition**: Either None or User Defined: • **None** (default): The result tool will show pure maximum equivalent strain in the build, where the value at each point corresponds to the strain when that material was a new layer. • **User Defined**: You define your own warning and critical thresholds: – **Warning Threshold**: Maximum equivalent strain above which results are considered a warning. – **Critical Threshold**: Maximum equivalent strain above which results are considered critical. For more information about interpreting results after the simulation is solved, see Review Results (p. 123). *Affects these tree objects: LPBF High Strain result item*

Chapter 5: Workflow Through the Project Tree

An AM process simulation involves most of the general steps found in any Ansys analysis with some additional steps and considerations. The workflow is described in the following sections:

5.1. Create the Analysis System

Ansys analysis systems are the mechanism to define physics type, analysis type, and solver type. There are two predefined, custom systems specifically for Laser Powder Bed Fusion (LPBF) additive manufacturing:

- AM LPBF Inherent Strain: A structural-only system in which strains are calculated from the use of a Strain Scaling Factor rather than from material properties and thermal loads.

- AM LPBF Thermal-Structural: A linked transient thermal analysis followed by a static structural analysis where strains are calculated from material properties and thermal loads. For simplification, we assume the physics are uncoupled in that data flows one-way from the thermal analysis to the structural.

While it is possible to set up an appropriate system without using an AM custom system in Workbench, we recommend you use a custom system because AM sample materials and several settings are set automatically for additive simulations. If you use a Mechanical Model from Workbench's Component

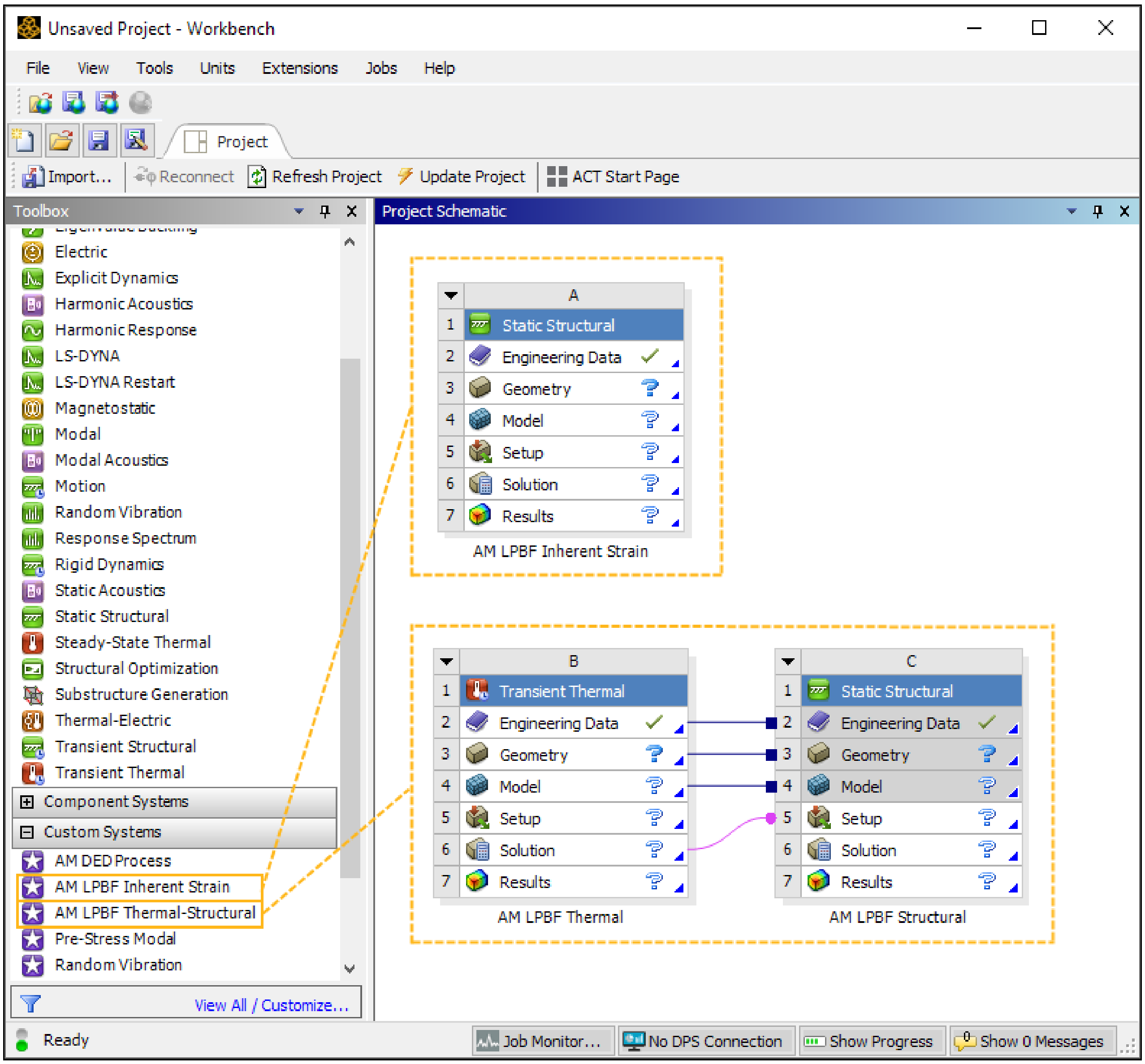
Systems, you must manually create the AM custom system. This is due to the fact that AM custom systems are pre-populated with AM sample materials in Engineering Data, and linking to the Mechanical Model necessitates the absence of materials in Engineering Data. If you open the Mechanical application independent of Workbench, you must manually insert the appropriate systems.

The remaining steps in this workflow assume you use an AM custom system.

Procedural Steps

1. Open Ansys Workbench.

2. Scroll down within the list of analysis systems to the list of custom systems and double-click your preferred AM custom system, either AM LPBF Inherent Strain or AM LPBF Thermal-Structural.



5.2. Define Engineering Data

In this step, you define the source of your materials using the Engineering Data repository for material properties. The Engineering Data workspace is designed to allow you to create, save, and retrieve material models, as well as to create libraries of data that can be saved and used in subsequent projects and by other users.

We recommend using temperature-dependent properties covering the range from room temperature to melt temperature. For the thermal analysis portion of an LPBF Thermal-Structural simulation, the properties required are thermal conductivity, density, and specific heat, and melting temperature must be defined for the material. For the structural analysis we require Young's Modulus, Poisson's Ratio, coefficient of thermal expansion, and a plasticity model, such as bilinear isotropic hardening (BISO). For an AM LPBF Inherent Strain simulation, both bilinear isotropic hardening (BISO) and multilinear isotropic hardening (MISO) plasticity models are supported.

The following popular materials for AM are provided as Ansys predefined samples in the Additive Materials library portion of the Engineering Data repository:

- 17-4PH Stainless Steel
- 316 Stainless Steel
- AlF357
- AlSi10Mg
- Co-Cr
- Inconel 625
- Inconel 718
- Ti-6Al-4V

The Ansys predefined materials are defined with bilinear isotropic hardening (BISO) plasticity models.

All of the Ansys AM sample materials have creep properties defined. Use creep properties as the mechanism for stress relief when simulating heat treatment to achieve more realistic residual stress relief effects.

Note:

The material properties used to generate thermal (loading) strains are not customizable when using **Machine Learning Thermal Strain** as your chosen strain definition for LPBF Inherent Strain simulations, as specific materials were used to train the ML model. In particular, the materials used to train the model are the same as those validated for thermal simulations in the Additive application.

Procedural Steps

Using Ansys Predefined Materials

The sample materials in the Additive Materials library (within the Engineering Data repository) are automatically available when you insert an AM custom system, either AM LPBF Inherent Strain or AM LPBF Thermal-Structural. If you plan to use one of the sample materials without heat treatment, no action is required here and you can move to the next step.

To use creep properties as the mechanism for stress relief in a heat treatment analysis using Ansys predefined materials, unsuppress the creep model in Engineering Data (p. 179) as described in the heat treatment advanced topic.

Using Granta MI's JAHM Curve Data

For information on using a material from the JAHM Curve Data database, see Using JAHM Temperature-Dependent Material Data in AM Simulation (p. 140). Note that either a Granta Selector or a Granta MI license is required to get access to JAHM datasets.

Using Custom Materials

To add your own custom material, follow the procedures as described in Material Data in the *Engineering Data User's Guide*. A nominal strength should be provided at the melt temperature, as a near zero modulus and/or yield strength could lead to convergence problems. (Ansys predefined materials take care of this internally.)

For information about defining your own creep model, see Creep Material Models in the *Material Reference*.

5.3. Attach Geometry and Launch Mechanical Application

Go directly to procedural steps. (p. 71)

Typically, you'll have these geometric bodies in the simulation model:

- **Part** – This is the part you are manufacturing. It should be a closed volume (that is, watertight), and should be oriented with the global Z axis as the build direction. It may be modeled either resting on the base plate (Z=0), or elevated off the base plate by supports.

  The part can be made of multiple bodies but must have boundaries that are aware of each other in order to assure a proper conformal mesh throughout the part. A conformal mesh is one where the elements fit together seamlessly along their shared boundaries or faces. In other words, there are no gaps or overlaps between adjacent elements. This is achieved through either shared topology, a voxelization mesh option, or AM Bond connections, depending on your meshing approach, as described in the meshing step (p. 78).

  If your model exhibits symmetry, you can simulate a sector of a large model to achieve a faster simulation time. See Modeling a Symmetrical Part (p. 189) for more details.

  Usually only one part is simulated even if there will be many duplicate parts nested on the base plate for efficiency. (A multiplier on the build time should be used if this is the case, as described in Machine Settings (p. 97).)

- **Support Structures** – Supports are needed to anchor overhanging part features so they do not break away from the platform during the 3D print process because of residual stress buildup. Overhanging features are usually those with angles less than 45° to the horizontal X-Y plane. Supports may be modeled in several ways. *Together, the part and supports constitute the build.*

- **Base Plate** – This is the platform on which the build (part plus supports) is to be printed. It is included in the simulation because it acts as a heat sink.

Other geometric bodies that may be simulated include:

- **Powder** – Modeling the powder may be useful if you will be simulating multiple parts close together on the build plate or if the part has features close together, where accounting for the heat transfer occurring between the parts or features is important. Create a separate, closed-volume (watertight) body to represent the powder in-between multiple parts or in-between features of the same part. Details of how to model powder (p. 194) in the simulation are found in Advanced Topics.

- **Non-build Components** – At times you may want to simulate geometry items that are present on the build plate but that are not being 3D-printed. These items may include clamps, bolts, measuring devices, instrumentation, etc. They may influence the heat dissipation and/or distortion of the part being built so they need to be included in the simulation. Details of how to model non-build components (p. 196) are found in Advanced Topics.

For your part geometry, commonly imported file types include `.dsco` files from the Ansys Discovery application, `.3MF` files, such as those from Materialise Magics, and stereolithography `.stl` files (faceted). For other options, see Attach Geometry/Mesh in the *Mechanical User's Guide*.

You may create the base plate and supports ahead of time in the CAD program or in a support generation tool, or wait to create those bodies in the Mechanical application (described in subsequent steps). If supports are created in CAD, the part and support bodies should be kept as separate bodies (that is, not merged) so that they can be distinguished as such in the AM simulation in the Mechanical application. Whether you set the bodies to "share topology" in the CAD program depends on whether you will be using a Cartesian mesh, a Cartesian mesh with voxelization option, or a layered tetrahedrons mesh in the simulation in Mechanical. For a discussion about this topic, see To Share Topology or Not? (p. 13). When everything is imported into the Workbench application, you will later identify the support bodies as predefined supports.

Alternatively, you may choose to *not* include supports when you attach the part but import the supports separately later as `.stl` files.

Procedural Steps

Refer to one of the following topics depending on your import method:

5.3.1. Attaching Geometry from an `.stl` File

To attach geometry and launch the Mechanical application:

1. Right-click the **Geometry** cell of the left-most analysis block (either AM LPBF Inherent Strain or AM LPBF Thermal) and select **Import**. Browse for your file and import it. (Double-clicking on the Geometry cell opens the Ansys Discovery application.)

2. After importing geometry, double-click the **Model** cell (or right-click, and select **Edit**) of the left-most block to launch the Mechanical application. It may take a few minutes for the application to open and your geometry to appear in the Geometry window. (In addition to the Workbench icon, , in the taskbar, note the icon once the Mechanical application opens; there are now two applications open.)

3. Once your geometry is loaded in the Mechanical application, review how it is presented in the project tree.

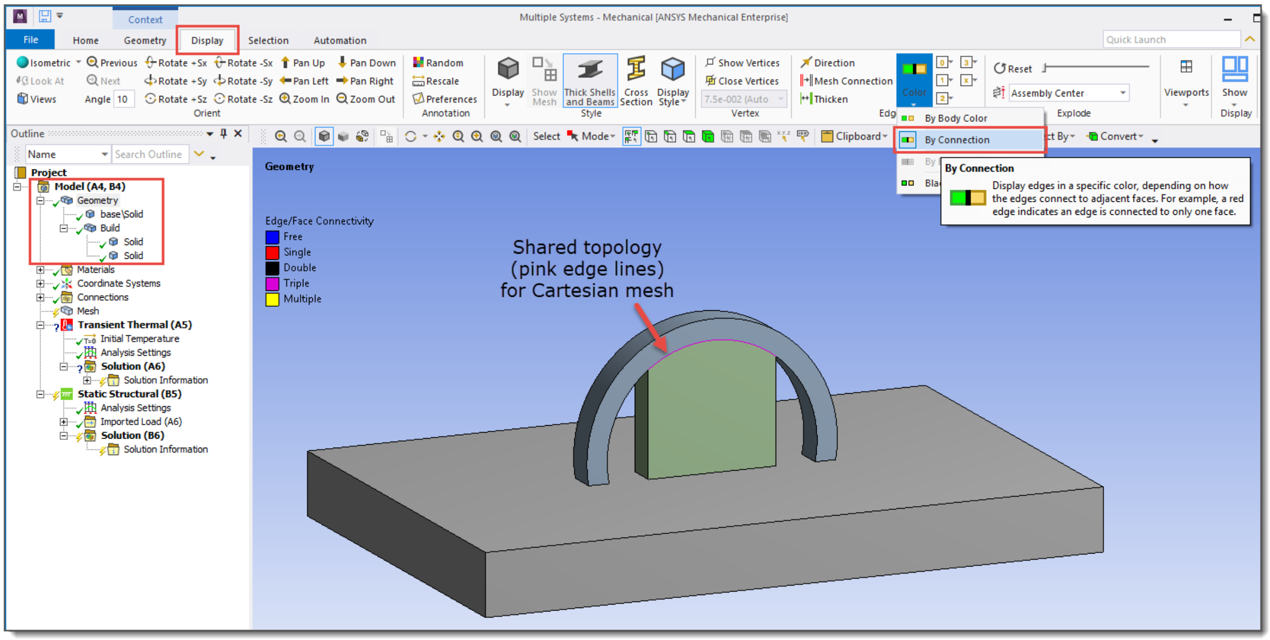
   We will use an example model to be simulated that includes an arch part and a solid support as separate bodies comprising the build, and the base plate as another separate body.

   Recall that there are different requirements for connectedness of geometric bodies depending on the mesh method you will use:

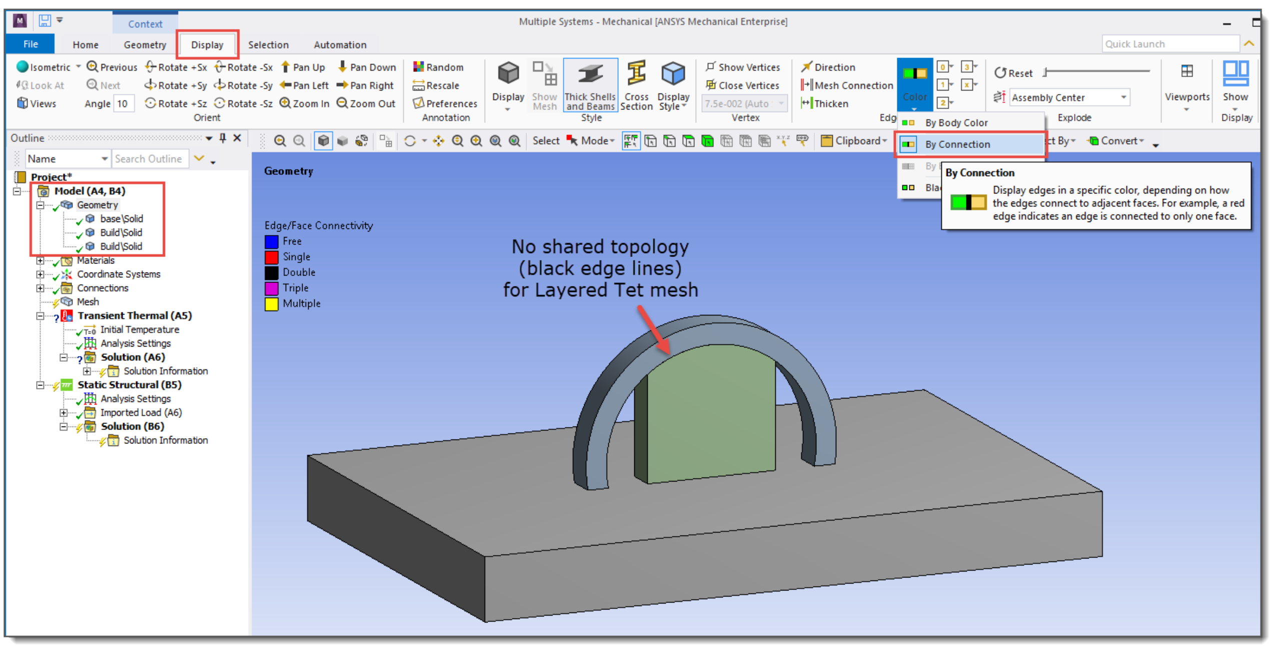
   - A Cartesian mesh method requires shared topology between the part and predefined supports.
   - A Cartesian mesh method *with the voxelization option* may have shared or unshared topology between the part and predefined supports.
   - A layered tetrahedrons mesh method requires that there be no shared topology (that is, unshared topology) between the part and predefined supports.

   To check whether your geometry has shared topology, click the Display tab and in the Edge group, choose **Color** and then choose **By Connection** from the drop-down menu. Pink edges means that an edge is shared by three faces which is an indication of shared topology.

   In the arch with support example, the geometry in the first figure has shared topology. First notice the structure of the bodies in the project tree. The arch part and support bodies are child objects under Build. Also, in the Geometry window, note the pink edges at the body interfaces when Edge Coloring > By Connection is turned on. We will mesh this model with a Cartesian mesh method (without using the voxelization option).



The geometry in the next figure has unshared topology. Notice how the bodies are shown as independent bodies in the project tree, not as child objects under the Build object. Also, there are no shared edges between bodies (lines are black instead of pink), indicating there is no shared topology. We will mesh this model with a layered tetrahedrons mesh method.



4. Once the Mechanical application opens, it is a good time to adjust the number of processors (cores) you are using 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 in the Solve group on the **Home** tab and change the **Cores** to something appropriate for your simulation.

5.3.2. Importing Geometry from a `.3MF` File

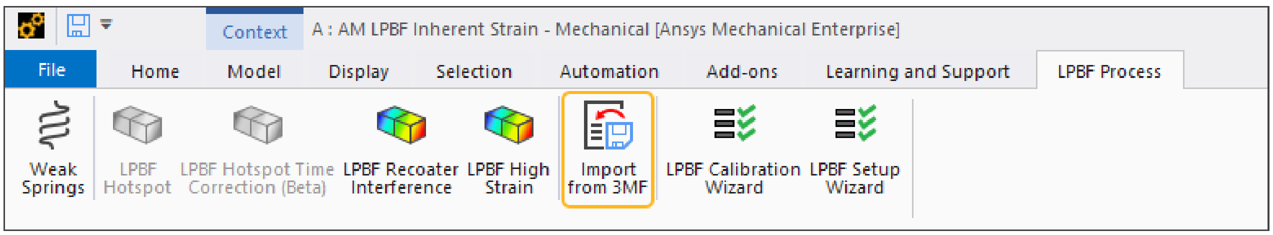
To import an AM model from a *3D Manufacturing Format*, or `.3MF` file, do not attach geometry in the Workbench application. Instead, you will import the geometry directly into the Mechanical application when it opens.

1. Double-click the **Model** cell (or right-click, and select **Edit**) of the left-most analysis block to launch the Mechanical application. It may take a few minutes for the application to open. (In addition to the Workbench icon, , in the taskbar, note the icon once the Mechanical application opens; there are now two applications open.)

2. Set the unit system in the Mechanical application to be the same as used for the Geometry Length in your `.3MF` file. On the right-hand side of the status bar (blue bar on the bottom of the UI), click the expandable triangle and change to the correct unit system as needed.

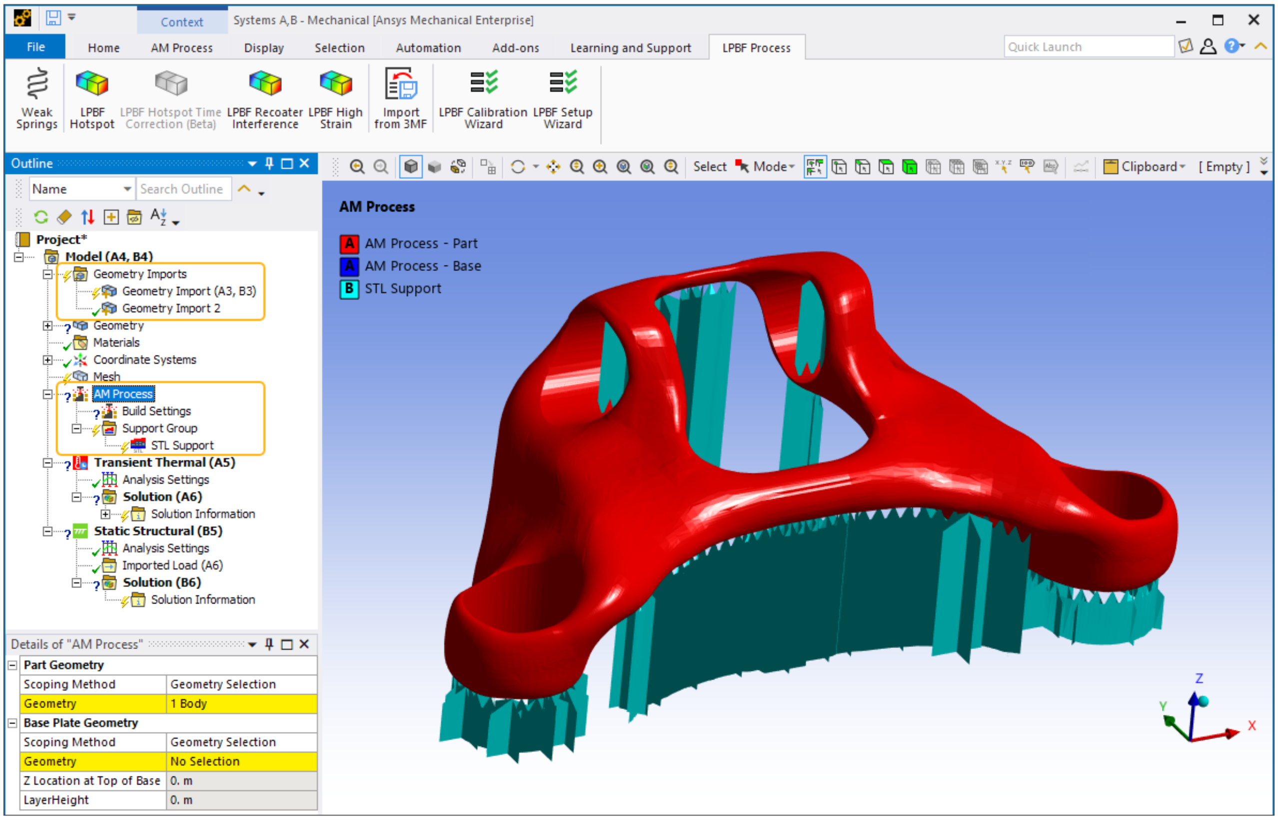
3. Load the LPBF Process Add-on (p. 8).

4. From the LPBF Process ribbon, click **Import from 3MF**. Navigate to your `.3MF` file and click OK.



The import function works by reading the structure of the `.3MF` file. A `.3MF` file describes geometry by listing the coordinates of vertices and constituent triangles, much like an `.stl` file. The **Import from 3MF** feature supports the *core* specification as defined by the 3MF Consortium, which includes information about the geometries and their relationships but not material or slicing information.

During an import, Geometry Import objects are created and added in the Mechanical application's project tree, and an `.stl` file is generated for each part and support body and placed beneath the appropriate Geometry Import object. Additionally, an AM Process object is added, and the newly inserted geometries are automatically scoped to it. That is, the part and support geometries (and base plate if present) are automatically identified and the supports for each part are scoped within a single Support Group under AM Process. The image below shows a model from an imported `.3MF` file.



After import, there is a message such as the following: *Geometries from the .3MF file have been imported as .stl files. Meshing options may be limited. We recommend using the LPBF Setup Wizard to set up the simulation.* Using the wizard is the best way to proceed as it has automatic short-cuts built-in. See Using the LPBF Setup Wizard (p. 17).

5.3.3. Working with Supports from Additive Prep

If you created your model for AM simulation that includes generated supports in Additive Prep:

- From within the Ansys SpaceClaim application, use the recommended transfer to Workbench feature, which attaches the geometry automatically. You can then skip step 1 in the procedural steps (p. 71) above.
- If you do not use the recommended transfer to Workbench feature and instead open the Mechanical application directly with a saved .scdoc file:
  - The Mechanical application recognizes the file as an AM file and launches with the AM Process and STL Supports objects automatically populated. If you had not joined the individual supports together into one support for each part, they will be imported as many individual stl supports and it may be unmanageable to continue on for simulation.
  - With the AM Process object already in place, the LPBF Setup Wizard cannot be used to set up this simulation. Continue setting up the simulation manually in the project tree.

5.4. Identify Geometries (AM Process Object)

On the imported geometry, identify which bodies are associated with the build (part and supports, if any) and the base plate. If your geometry from CAD does not include a base plate, use the Construction Geometry feature to create one.

Procedural Steps

We use the AM Process object in the project tree to establish the options and assumptions appropriate for an additive manufacturing simulation. Note that the AM Process object is available only if you have an Ansys Additive Suite software license with Ansys Mechanical Enterprise or one of the multiphysics bundles. If AM Process is grayed out, check your software license.

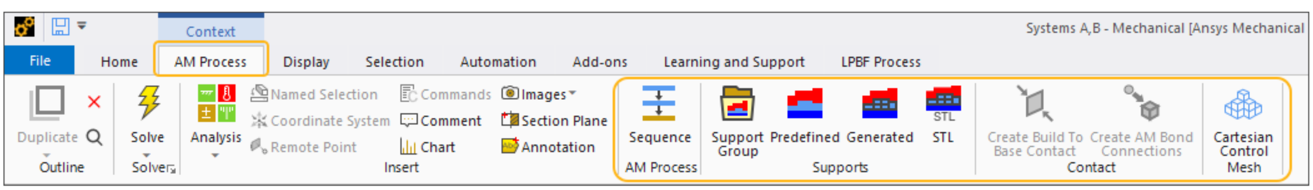
If you have inserted an AM custom system—either AM LPBF Inherent Strain or AM LPBF Thermal-Structural— as the analysis system, or you are working with a model transferred from Additive Prep using the transfer to Workbench feature, then the AM Process object is already added and you can skip the first step in the procedural steps below.

1. Select the **Model** object in the project tree, then select the **Model** contextual tab on the ribbon and then the **AM Process** option 

in the Define group. Or, right-click the **Model** object or in the Geometry window and select **Insert** > **AM Process**.

   Once you insert the AM Process object, certain automatic actions take place. The application automatically:

   - Displays the AM Process context tab, providing useful shortcut options specific to an AM Process Simulation. These options will be demonstrated in subsequent steps. Note that the AM Process context tab is visible only when the AM Process object in the tree is selected.



   - Sets Step Controls in an AM Process Sequence worksheet (discussed later) and in Analysis Settings for each analysis.

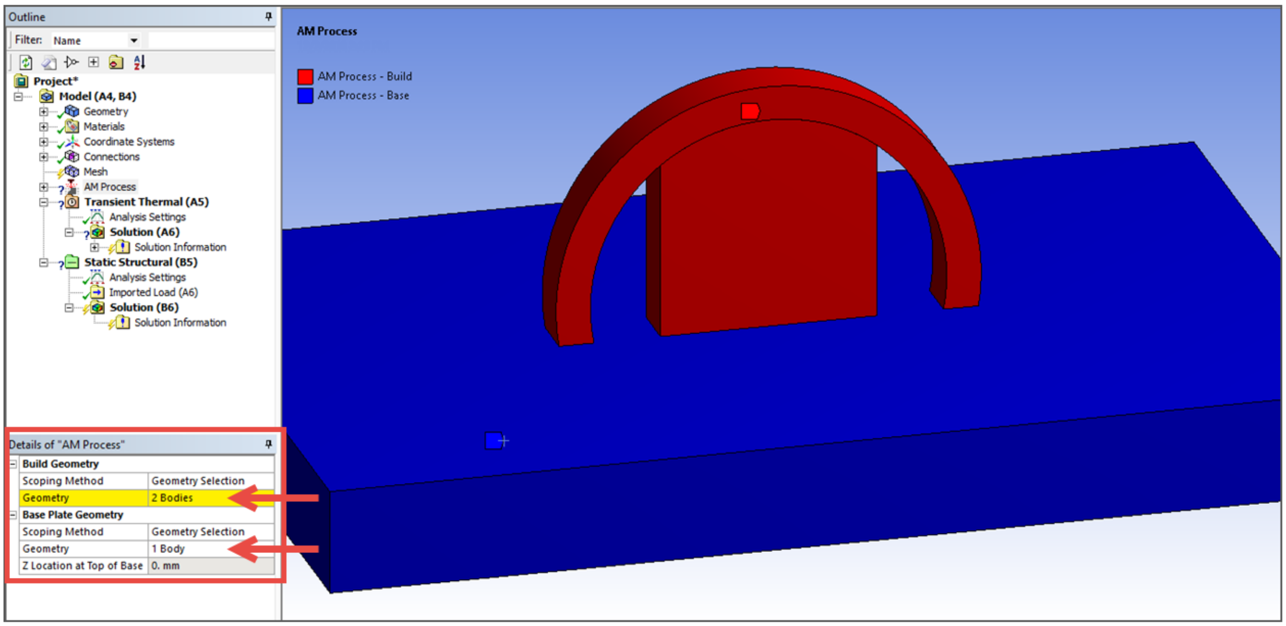
   - Suppresses the calculation of thermal fluxes, nodal forces, Euler angles, volume and energy, and other miscellaneous items to reduce the size of the results file. (Set in Analysis Settings for the transient thermal analysis.)

2. Next you need to identify which bodies are which in the geometry you imported. Select the **AM Process** object and then:

   - Select the body or bodies representing the part and the supports that make up the **Build Geometry** and hit **Apply** in the Details view. To select a body, be sure that your *mode of selection* is on body  (rather than on face, edge, or vertex). Use the Ctrl key while clicking the left mouse button to select multiple entities (or click, hold, and drag). Or use named selections as a basis for selection. The build is now shown in red.

When selecting supports for identification, select only *solid* support bodies and not volumeless supports (STL Supports) that may have been imported via transfer from Additive Prep. If you want to have the application create supports automatically, you'll do that later. For more information about supports, see Identify and/or Generate Supports (p. 86).

- Select the body that is the **Base Plate Geometry** and hit **Apply** in the Details view. The base is now shown in blue. (If your imported geometry does not have a base plate, construct one using step 3.)



At the bottom of the Details view, the offset of the part from the base is shown as Z Location at Top of Base. This is simply a confirmation of what the Mechanical application reads from the CAD file.

3. To create the base plate if not imported with the CAD geometry, select the **Model** object and then:

   - Right-click the **Model** object or in the Geometry window and select **Insert** > **Construction Geometry**.
   - Right-click the new **Construction Geometry** object (under Model) and choose **Insert** > **Solid**.
   - In the Details panel, fields appear for you to enter overall dimensions of the base and coordinates for the center of the top of the base. Enter values for **X1**, **X2**, **Y1**, **Y2**, **Z1** and **Z2**. An outline is provided to preview the dimensions. Right-click on **Solid** and select **Add to Geometry**. See Construction Geometry for more information.
   - To identify this newly created body as the Base Plate Geometry, highlight the **AM Process** object, select the new body and hit **Apply** in the Details view for Base Plate Geometry.

5.5. Assign Materials

In this step, you assign a material to each geometry body, considering the following guidelines:

- All bodies comprising the build (part and supports) must be the same material.
- The base plate can be a different material.
- If you are modeling powder, use the same material as the build.
- Non-build components can be different materials.

Procedural Steps

When you select a geometry entity in the project tree, the Details view lists all the settings associated with that body.

1. In the project tree under Model, expand the **Geometry** object to see its child objects below it. Select the geometric entity that is the *build geometry* and, in Details view, change the **Assignment** (under Material) to be the AM material that you want to use, either one of the Ansys predefined samples, or a custom material you defined in Engineering Data (p. 69). When you assign a material to the build, the child objects below it (the part and supports) are also assigned that same material.
2. Select the entity that is the *base plate body* and change its material **Assignment**, as desired.

5.6. Apply Mesh Controls and Generate Mesh

Go directly to procedural steps. (p. 79)

The layer-by-layer additive printing process is simulated with element layers added one-by-one using the element birth/death technique. As such, the mesh must have a uniform size in the build (global Z) direction. That is, each element layer must have the same height (constant Z coordinate).

For the build, we recommend using one finite element "super layer" of elements to represent 10-20 actual metal powder layers. If your machine has a 25-micron powder layer thickness (also called deposition thickness), your element size should be between 0.25 and .5 mm. The element sizing does *not* have to be an even multiple of the deposit layer thickness. Note that the real machine build time is approximately the transient thermal build step simulation time multiplied by R^(1/3), where R is the number of deposit layers in one element layer. (This estimate of the real build time is provided for you in the simulation results.)

A much coarser mesh is acceptable for the base plate because it is simply serving as a heat sink and a fixed support in the simulation.

Three primary meshing methods/options are available for additive manufacturing process simulation, each with their strengths and weaknesses.

- The **Cartesian Mesher** creates a hex mesh that approximates the geometry. Small features, curved surfaces, and horizontal or vertical surfaces that are not multiples of the mesh size are not captured accurately unless a small mesh size is used. The method is fast and, for most distortion and residual stress predictions, is quite adequate.

The Mechanical application can automatically generate supports when a part has a Cartesian mesh. (See Generated Supports (p. 90) in the next step.)

- The **Cartesian Mesher with the voxelization option** creates a voxel (cubic element) mesh for the geometry. Small features, curved surfaces, and horizontal or vertical surfaces that are not multiples of the mesh size are accounted for by a knock-down factor technique.

  Limitations with the voxelization option:

  – The voxelized mesh option (Cartesian Mesh with Voxelization Option) is not available on a Linux platform because the underlying voxelizer code is incompatible with Linux. Choose between a Cartesian mesh or a layered tetrahedrons mesh.

  – Shared topology between part bodies, predefined supports, and/or powder bodies is allowed with the voxelization option but not recommended because some elements may get assigned to the wrong body at the interface.

- The **Layered Tetrahedrons Mesher** creates a tetrahedrons mesh that conforms to a specified layer size. It captures the geometry well, and is useful if there are organic curves, small features, such as holes, or thin-walled parts.

  Limitations with the layered tetrahedrons mesher:

  – Bodies with shared topology cannot be meshed with the layered tetrahedrons mesher. We recommend you separate bodies into individual parts in CAD. (Use the Unshare tool in the Shared Topology group to unshare coincident topology in Discovery, or Explode Part in DesignModeler to do this.) See To Share Topology or Not? (p. 13).

  – The AM octree adaptive meshing (p. 163) method is not supported for parts meshed with layered tetrahedrons.

  – Auto-generated supports are not available for parts meshed with layered tetrahedrons. If you want to use layered tetrahedrons mesh, you must either include supports with your geometry as predefined supports or import a support as a separate `.stl` file.

  – The model should not have other suppressed bodies.

  – The mesh is not associated back to the geometry.

  – This method cannot be used in conjunction with mesh controls such as Inflation, Refinement, Match Control, Pinch, Face Meshing, Face Sizing, and Edge Sizing controls.

Procedural Steps

We will describe the meshing steps using the arch with support model for both the Cartesian and layered tetrahedrons mesh techniques.

Using a Cartesian Mesh

1. Set mesh controls for the part and support bodies. The size of the mesh is required to be the same throughout all bodies in the build. Remember that the build requires a finer mesh than the base plate.

    a. To set mesh controls for the build, we will use a meshing shortcut designed for AM Process Simulations. In the project tree, select the **AM Process** object and then select the **Cartesian Control** option from the ribbon (AM Process context tab). Or, right-click the **AM Process** object and select **Insert** > **Cartesian Mesh**. Notice that a Body Fitted Cartesian object has appeared under the AM Process object and become active and that all the bodies that are part of the build are selected.

    b. In Details of the Body Fitted Cartesian object, under Definition, set the **Element Size** to the desired value. (Click in the right column on the word Default to activate the text field.)

    c. Under Advanced, you may choose to set the **Projection Factor** slider to a value between 0 and 1. The Projection Factor defines how well the mesh will fit to the geometry. A value of 0 (default) results in cubic elements with a rough fit to the geometry. Increasing the Projection Factor will change the shape of the elements to better fit the geometry and may yield better results in some cases but may also result in a failed mesh. Our recommendation is to leave it at 0, or close to 0, initially and then iterate from there to see the effects of changing Projection Factor.

    Important:

    The Projection Factor must be set to 0 when using generated supports (p. 86).

    You can apply a Coordinate System for the mesh. Typically, the default global coordinate system is appropriate but you may want to set it to one located on the top surface of the base plate.

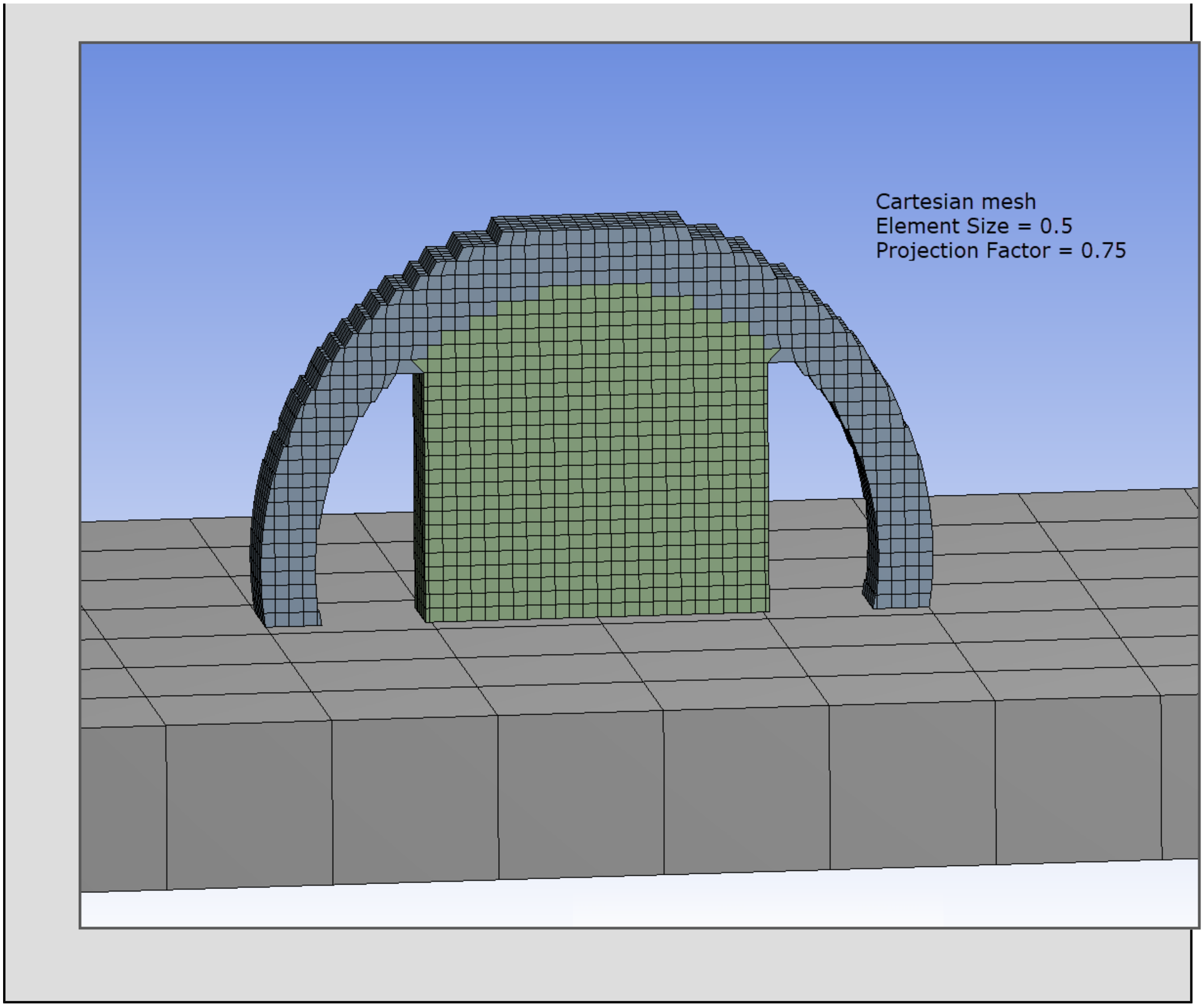
    For additional information about Cartesian meshes, see Cartesian Method Control in the *Meshing User's Guide*.

2. Set mesh controls for the base plate. Highlight the **Mesh** object and right-click **Insert** > **Sizing**. A Sizing object is created under the Mesh object and becomes active.

    a. Select the base plate body and hit **Apply** in the Details view.

    b. Under Definition, change the **Element Size** to the desired value, something that will result in a fairly coarse mesh.

3. Finally, highlight the **Mesh** object and right-click **Generate Mesh**. If you don't like the generated mesh you can easily go back and change element sizes, then right-click Mesh and select Update.


Cartesian mesh
Element Size = 0.5
Projection Factor = 0.75

Using a Cartesian Mesh with Voxelization Option

1. First, set mesh controls for the part and support bodies. The size of the mesh is required to be the same throughout all bodies in the build. Remember that the build requires a finer mesh than the base plate.

   a. To set mesh controls for the build, we will use a meshing shortcut designed for AM Process Simulations. In the project tree, select the **AM Process** object and then select the **Cartesian Control** option from the ribbon (AM Process context tab). Or, right-click the **AM Process** object and select **Insert** > **Cartesian Mesh**. Notice that a Body Fitted Cartesian object has appeared under the AM Process object and become active and that all the bodies that are part of the build are selected.

   b. In Details of the Body Fitted Cartesian object, under Definition, set the **Mesh Using Voxelization** option to **Yes**. (This option is set to No and is grayed out on a Linux operating system.)

   c. Under Advanced, set the **Element Size** to the desired value.

   d. Adjust the **Wall Thickness**, as needed, according to the single-bead thickness set by your machine.

   e. Adjust the **Subsample Rate**, as needed. For most cases, we recommend using the default value of 5.

      The part will be meshed with cubic elements. Each cubic element is divided into sampling regions to determine density of material within that element. These are used as material property knockdown factors. A Subsample Rate of 5 (default) = 5 x 5 x 5 = 125 subdivisions. Subsample Rate affects the accuracy of element density.

2. Next, set mesh controls for the base plate. Highlight the **Mesh** object and right-click **Insert** > **Sizing**. A Sizing object is created under the Mesh object and becomes active.

   a. Select the base plate body and hit **Apply** in the Details view.

   b. Under Definition, change the **Element Size** to the desired value, something that will result in a fairly coarse mesh.

3. Finally, highlight the **Mesh** object and right-click **Generate Mesh**. If you don't like the generated mesh you can easily go back and change element sizes, then right-click Mesh and select Update.

Important:

The mesh generated using the voxelization option is not fully associated to the part geometry. Loads scoped to the part body (face, edge, vertex) are not possible. However, loads scoped to the mesh of the part (that is, FE loads) are valid. These can be defined with Named Selections or manual selection.

As a best practice when using the voxelizer mesh method, select all build bodies at one time for a single mesh operation. This includes part bodies, Predefined Supports (p. 86), and powder bodies, if any. As a result of the single voxel mesh operation, conformal mesh will be created among all bodies and contact connections between them will not be needed. If each geometry has its own voxel mesh method object, then you will need to define AM Bond connections (p. 93) or contacts between them.

Consider using AM octree adaptive meshing to reduce simulation time if your part has bulky or blocky areas, especially near the bottom (that is, near the lower layers). The AM octree method of nonlinear mesh adaptivity coarsens the mesh in previously deposited layers, resulting in a reduction in total element count. The method is only available for parts that have been voxelized. See Using AM Octree Adaptive Meshing (p. 163).

Using a Layered Tetrahedrons Mesh

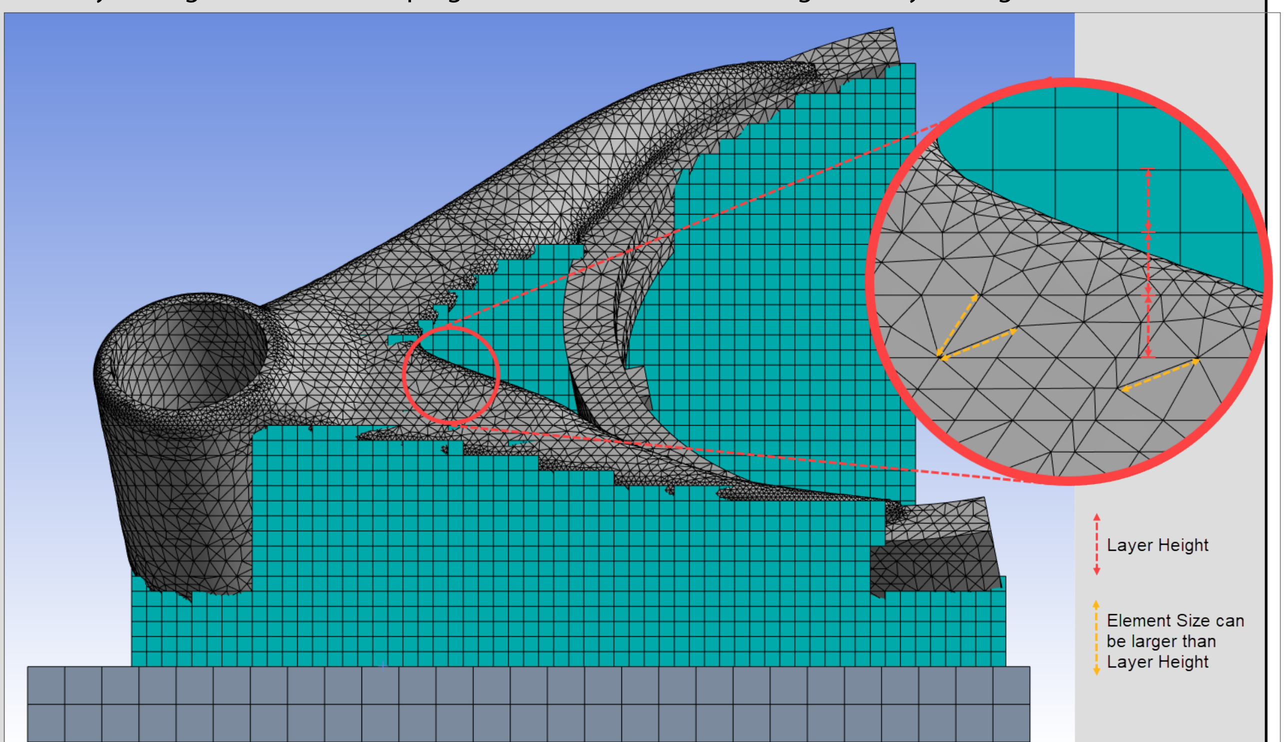
To use a layered tetrahedrons mesh, in Ansys Mechanical:

1. Click **Mesh** in the Tree Outline, right-click and select **Insert** > **Method**. Select the build geometry—use the Ctrl key to select multiple bodies, for example if you have part and predefined support bodies—and in Details view, click **Apply**. In Details under Definition, change the **Method** to Layered Tetrahedrons.

2. In the **Layered Tetrahedrons** Details view, set the **Layer Height** as desired. Note that the layer height should balance the need to capture features and the need for a reasonable simulation run time. The recommended setting for this "super layer" is 10-20 times the size of the machine deposition thickness.

   Most of the remaining advanced settings in the Details view have adequate defaults.

3. Click the **Mesh** object, and in its Details view, set the **Element Size**. Element Size can be less than, equal to, or greater than the Layer Height but should not be greater than 6 times the Curvature Min Size. Consider starting with an initial value slightly larger than Layer Height. This allows individual elements to have non-Z-direction edge lengths longer than Layer Height while still keeping the overall Z-direction height at Layer Height.



   Initially, use the default settings for the remaining options. The default settings are customized for additive process simulations only when the AM Process object is in the project tree so *it is always best to add the AM Process object before meshing*. Review the following global mesh settings and change them as necessary:

Defaults

Element Order	Quadratic (default if the AM Process object is in the project tree). Note that mid-noded elements will be generated, but the mid-nodes will be kept straight and will not conform to the geometry.
Element Size	Can be greater than Layer Height. The Element Size should not be greater than 6 times the Curvature Min Size specified. The mesh quality reduces as the Element Size / Curvature Min Size ratio increases.
Sizing	
Use Adaptive Sizing	No (default if the AM Process object is in the project tree)
Capture Curvature	Yes (default if the AM Process object is in the project tree)
Curvature Min Size	Dynamically calculated to be 0.3 times the Layer Height (specified in Details of Layered Tetrahedrons mesh method object). Used by the mesher to generate a surface mesh before the slice layers are generated, hence this min size drives the resolution of the mesh. The min size should be decided based on the features that are to be resolved and the Layer Height. A recommended value is 1/4 to 1/3 of the Layer Height.
Curvature Normal Angle	27 (default if the AM Process object is in the project tree). The recommended range is 18 to 36. Use a lower angle if the model has small features like holes and fillets that you want to preserve.
Defeature Size	Dynamically calculated to be 10% of the Curvature Min Size. Used to define the default value for sliver face height. Based on the model, you might be required to increase it. We do not recommend using a value greater than one-half of the Curvature Min Size.

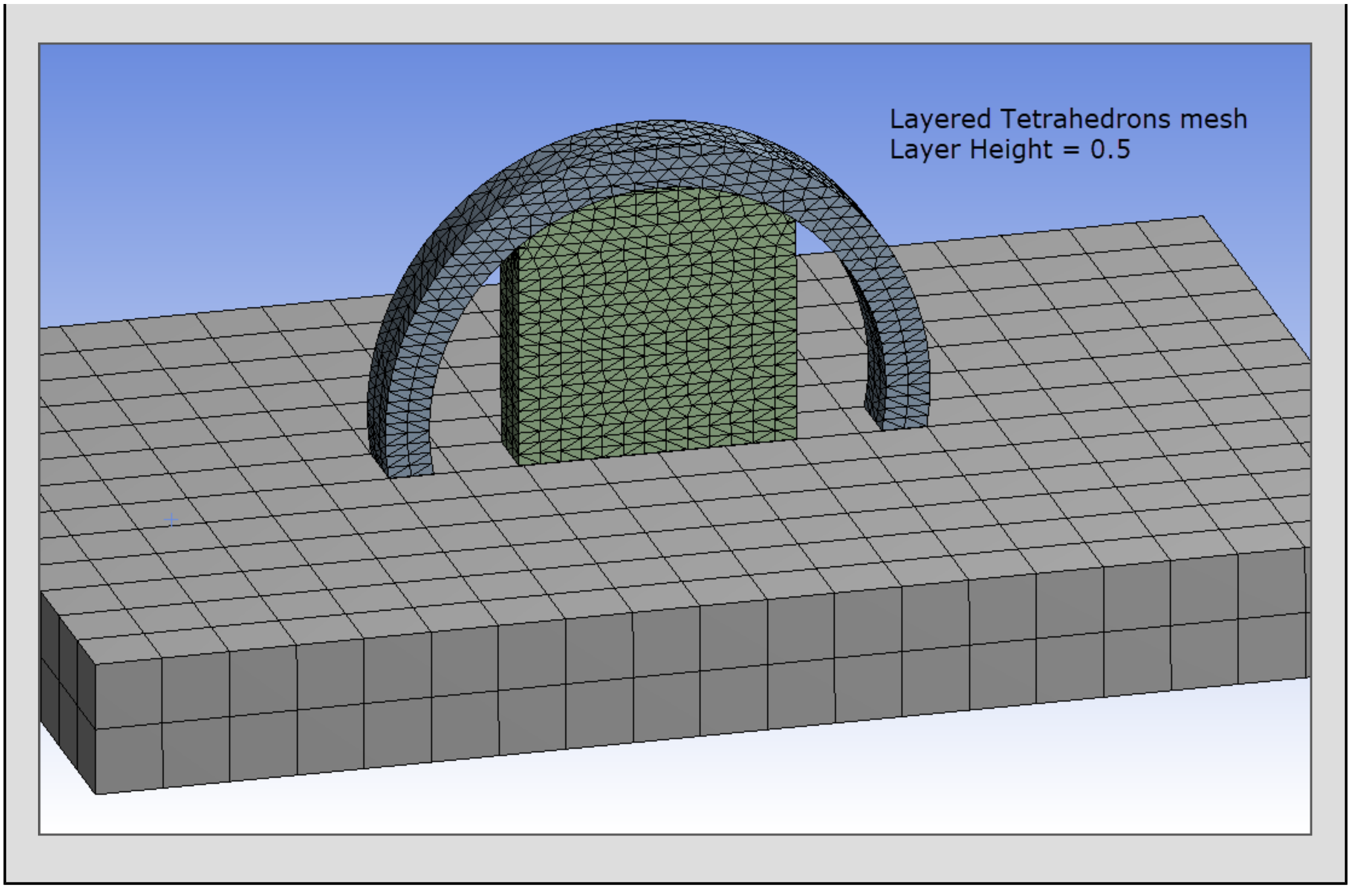
Understanding the approach that the layered tetrahedrons mesher uses may be useful if you are not satisfied with the meshes being generated. After experimenting with the mesh settings described here, if you are still not happy with the tetrahedrons mesh, consider adjusting Relative Tolerance, Inflate Relative Tolerance, and Sliver Triangle Height. Care must be taken to ensure that the layer slices cut the geometry in such a way as to avoid thin slices. Thin slices may cause the mesher to struggle or result in poorly shaped elements. Usually adjusting the mesh Layer Height or increasing Relative Tolerance will lead to a higher quality mesh. Refer to the Layered Tetrahedrons Method Control in the *Meshing User's Guide* for further information.

4. Next, set mesh controls for the base plate that will produce a much coarser mesh. Highlight the **Mesh** object and right-click **Insert** > **Sizing**. A Sizing object is created under the Mesh object and becomes active.

   a. Select the base plate body and hit **Apply** in the Details view.

   b. Under Definition, change the **Element Size** to the desired value, something that will result in a fairly coarse mesh.

5. Finally, highlight the **Mesh** object and right-click **Generate Mesh**. If you don't like the generated mesh you can easily go back and change element sizes and settings, then right-click Mesh and select Update.



5.7. Identify and/or Generate Supports

Go directly to procedural steps. (p. 88)

Supports may be imported with your geometry, you may import them as a separate `.stl` file, or you may generate them directly as elements in Mechanical, or some combination of the three.

- **Predefined Supports** are support bodies that are imported with your part geometry and you will need to identify them as such.
- **STL Supports** are supports imported as `.stl` files or transferred in automatically from Additive Prep using the transfer to Workbench feature. STL Supports are generally volumeless (that is, not watertight), thin-walled structures created with Additive Prep or other support-creation tools. The thin, intricate support walls that may include perforations are made of many small facets. We use a voxelizing technique, similar to that used in Additive Print, to account for this. The mesh is generated with cubic elements that are internally divided into subdivisions for sampling the presence of material to determine the overall densities of the elements. These, in turn, are used for the material knockdown factors.

  STL Support files must be in millimeters. Furthermore, with STL Supports, element size is set by the mesh criteria used for the part. It is for this reason that you must mesh the part before, or at the same time as, meshing the STL Support. If you used a Cartesian mesh, the element size uses the value you specified for Build Element Size. (Element size may be slightly smaller than the Build Element Size in certain cases.) If you used a layered tetrahedrons mesh for the part, the element size will be the value you specified for Layer Height.

- **Generated Supports** are available only for parts meshed with Cartesian mesh (with voxelization option equal to yes or no). These supports generated by the Mechanical application are either automatically detected or user-defined. For automatic detection you specify an overhang angle (the default value is 45° to the horizontal X-Y plane) under which supports will be created. For user-defined supports you select individual element faces under which supports will be created. Supports are generated as elements vertically straight down from the overhanging portion of the build to the base, or to a lower portion of the model if it is in the way.

We Treat Supports as Homogenized Solids

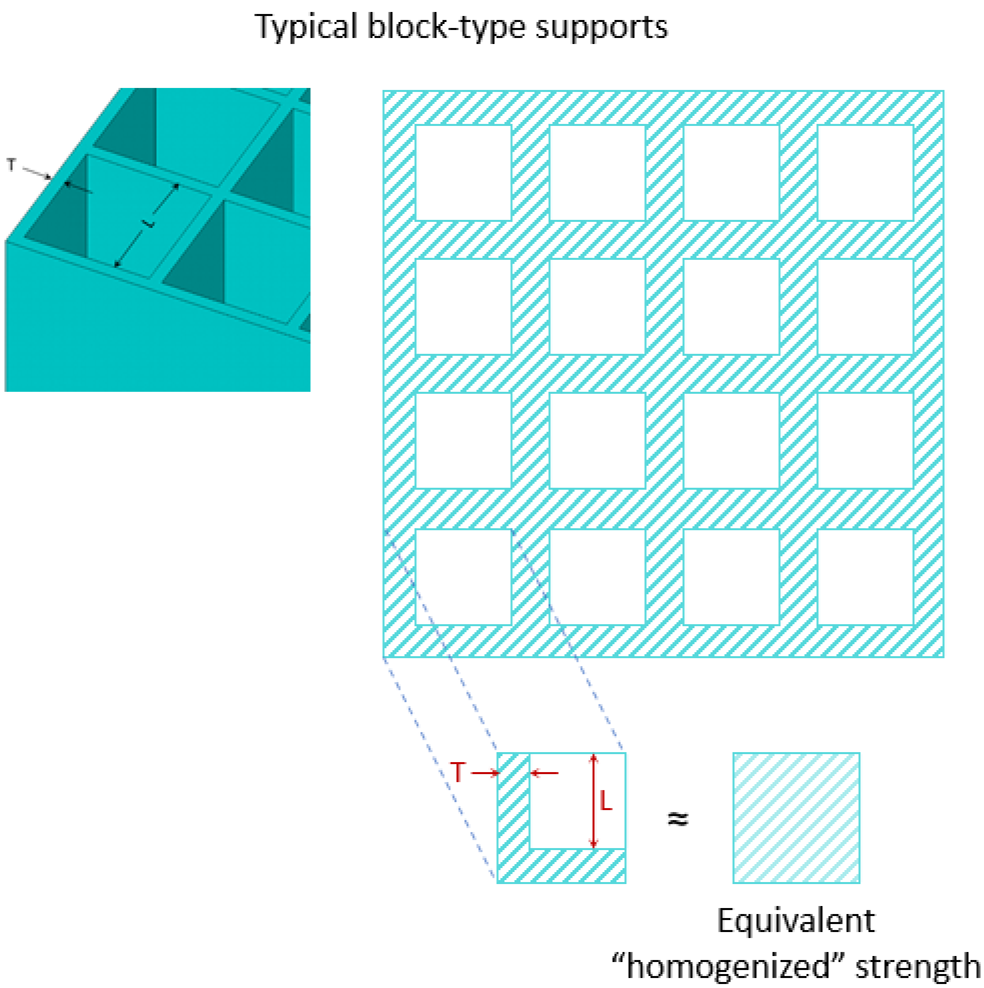
In additive machines, supports are printed with the same material as the part but as thin walled structures with less mass than the part. In the simulation we model the supports as an equivalent "homogenized" solid rather than as thin-walled structures. Regardless of whether the supports are predefined bodies, automatically generated, or imported as `.stl` files, their properties will be scaled down to account for this homogenization technique. Affected properties are elastic modulus, shear modulus, yield strength, density, and thermal conductivity.

Scaling down properties is done automatically for STL Supports. For Predefined Supports and Generated Supports, you will do this in one of three ways:

- Specifying an overall multiplication factor. This factor is the ratio of the actual support area to the area of the solid area. For example an overall multiplier of 0.33 will adjust the properties of the supports to be a third of that of the part material.

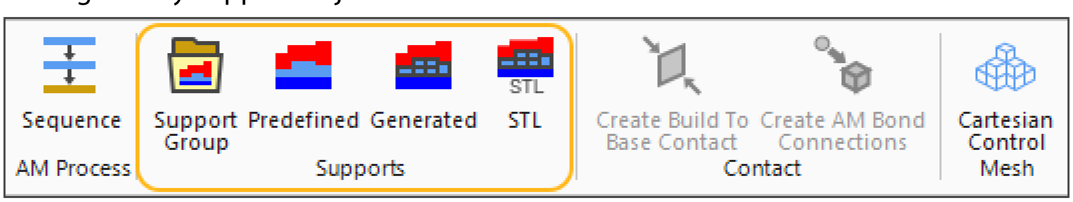
- Specifying individual multiplication factors for each orthotropic direction of each material property.

- Specifying wall thickness (T) and spacing (L) for block-type supports, commonly output from support generation tools. In this method, we calculate the equivalent homogenization factor for you.



Procedural Steps

Supports are identified and/or added using the Support Group in the AM Process context menu. As you add supports, objects are added to the project tree under AM Process. You may want to rename these objects to meaningful names if you have many support groups. If things get confusing as you add supports, useful tools are the Hide Support and Hide All Other Bodies options, accessible by right-clicking on any support object.



Predefined Supports

If support bodies were included in the CAD geometry, you need to identify them as supports so that the Mechanical application is aware of which bodies to adjust properties for, or to remove if support removal steps are specified. To identify these predefined supports:

1. Highlight the **AM Process** object, and then select the **Predefined Support** option on the AM Process context toolbar. Or right-click in the Geometry window and select **Insert** > **Predefined Support**.

2. Select the support bodies (geometry selection) with the mouse, holding the Ctrl key down to select multiple bodies (or click, hold, and drag). Click **Apply**. Or, choose Named Selections.

3. To scale down material properties, in Details under Support Material Settings, choose **Block** or **User Defined** for Support Type. If choosing Block supports, specify Wall Thickness (T) and Wall Spacing (L). If you choose User-Defined for Support Type, you may adjust the material properties by specifying either one overall multiplier or property-by-property multipliers. These multipliers are the ratio of geometric areas of the supports to the areas of the solid geometry, where the areas are the projected areas in the X, Y, and Z directions.

Generated Supports

The Mechanical application's ability to automatically generate supports is available only if the part is meshed with a Cartesian mesh (with voxelization option equal to yes or no). When generating supports, the application can automatically determine their locations or you can specify the support locations.

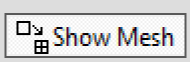
For *automatic detection* of generated supports:

1. Highlight the **AM Process** object, and then select the **Generated Support** option on the context toolbar. Or right-click in the Geometry window and select **Insert** > **Generated Support**.
2. Click the **Support Group** object above Generated Support and change the overhang angle, **Hang Angle**, to an angle between 0 and 90°, or leave it at the default of 45°.
3. Right-click **Support Group** and select **Detect and Generate Supports**. Supports are generated for all bodies that are part of the build. Note that an option exists to Detect Supports only (that is, not generate supports). This is quite useful if you want to see the effect of changing Overhang Angle. If you are satisfied with the location of supports, then select Generate Support Bodies. Another option allows you to detect and generate supports above a particular Z location (Detect above Z location, in Details of Support Group), allowing further control of where supports are generated.

For *user-defined* generated supports, you restrict the regions under which supports will be generated by selecting specific element faces in Scoping Method:

1. Highlight the **AM Process** object, and then select the **Generated Support** option on the context toolbar. Or right-click in the Geometry window and select **Insert** > **Generated Support**.
2. Switch to element face mode of selection , and use Ctrl-left-click to select multiple element faces, or double-click-left to select all the elements on a surface. Click **Apply** in the Details panel.
3. Right-click **Generated Support** and click **Generate Support Bodies**.

Important:

Supports are generated as *elements only* (that is, there are no corresponding geometric bodies created for the new supports). When viewing geometry in the Geometry window you won't see the supports. You will see the support elements when you select the AM Process object (or one of its children), or the Mesh object. Or you can use the Show Mesh toggle button 

to reveal the mesh even when the Model object is active.

More importantly, if you would like to use the supports generated in Mechanical in your final print strategy, you will need to convert the elements to geometric bodies for the `.stl` file required by the printer.

STL Supports

The workflow is slightly different depending on how you add STL Supports. They can be transferred in automatically from Additive Prep or imported from an `.stl` file. Both methods are described in step 1 below.

1. To mesh STL Supports transferred in automatically from Additive Prep:

   a. When supports are transferred automatically from Additive Prep, they are not scoped to any particular body. That is, they are not associated to the part body. In the project tree, click to expand the **Imported Supports** object and then click the **STL Support** object. In the Geometry window, select the body representing the *part* and click **Apply** for the Geometry in the Details view of STL Support.

   To import and mesh STL Supports from a file:

   a. Before inserting the STL Support, be sure the part is meshed and the build geometry is identified in the AM Process object. Highlight the **AM Process** object, and then select the **STL Support** option on the context toolbar. Or right-click in the Geometry window and select **Insert** > **STL Support**.

   b. In the Details view of STL Support, identify the source of the `.stl` file, either a file (default) or a previously imported STL Construction Object. Click **File Name** to navigate to the appropriate folder and select your `.stl` support file.

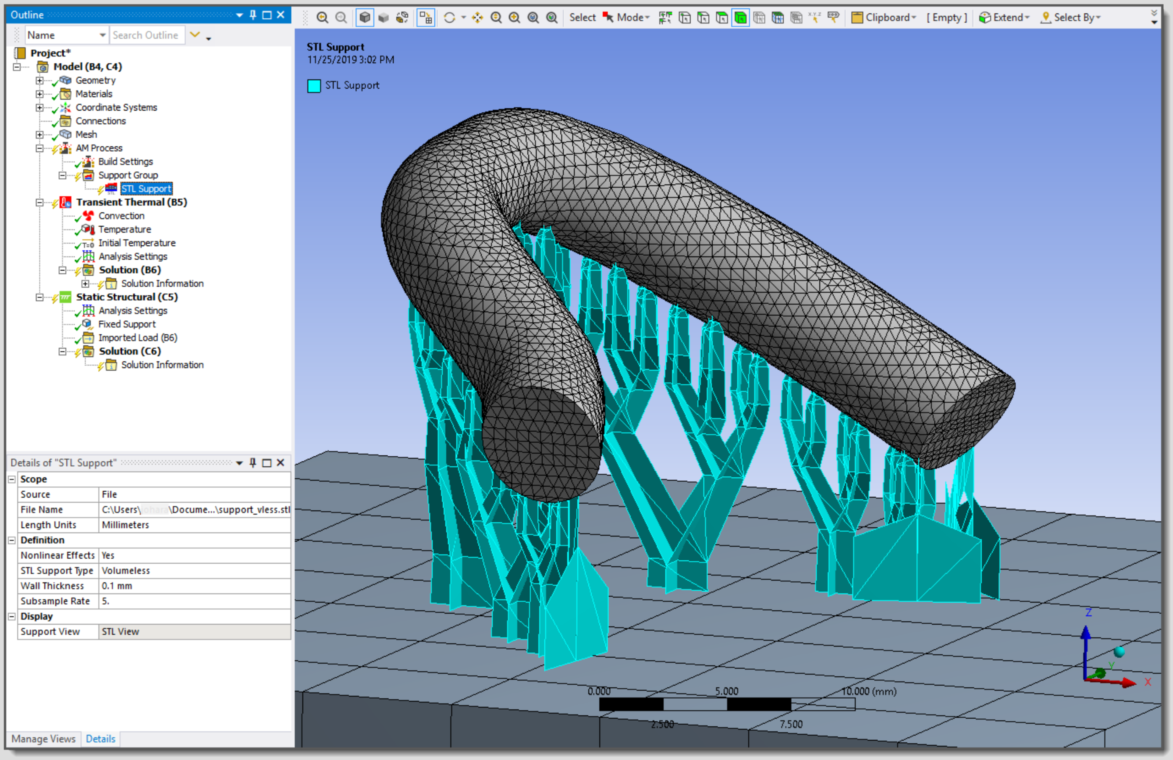
2. In the Details view of STL Support, change the Length Units, as needed. The default is millimeters. Most often, this is what you want.

3. Identify the **STL Support Type** by selecting Volumeless or Solid from the drop-down. Volumeless (default) is for non-watertight supports such as block, heartcell, rod, or line supports created by Additive Prep. Most often, this is what you want. Choose Solid for solid bodies that are watertight, such as custom supports created in Additive Prep.

4. For Volumeless support type, adjust the **Wall Thickness**, as needed, according to the single-bead thickness set by your machine.

5. Adjust the **Subsample Rate**, as needed. For most cases, we recommend using the default value of 5.

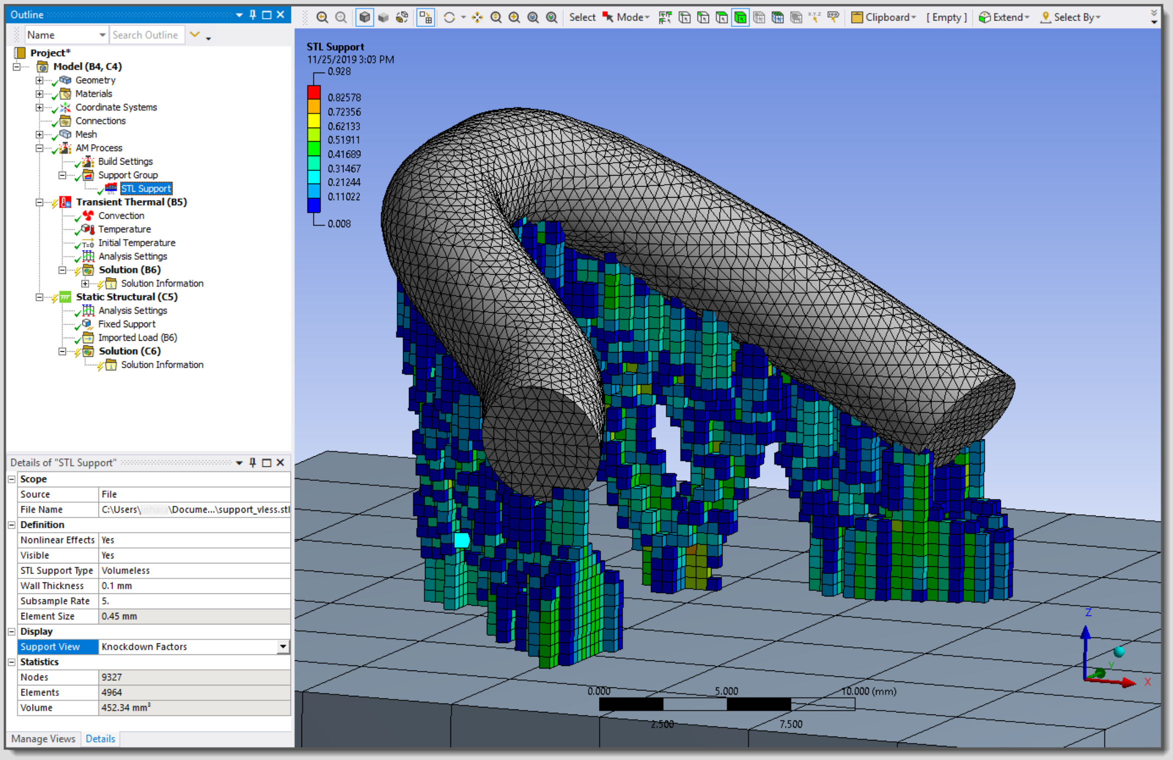
   The following figure shows a curved bar model with tree supports generated in Additive Prep imported as an STL Support file.



The support structure will be meshed with cubic elements. Each cubic element is divided into sampling regions to determine density of support material within that element. These are used as material property knockdown factors. A Subsample Rate of 5 (default) = 5 x 5 x 5 = 125 subdivisions. Subsample Rate affects the accuracy of element density.

6. Right-click the **STL Support** object and select **Generate Mesh** to mesh the support.

7. Use the **Support View** (under Display) to switch back and forth among the view options of STL View, Mesh View, or Knockdown Factors. (Or, the knockdown factors may be turned on using Display Style in the Mesh object Details view.)



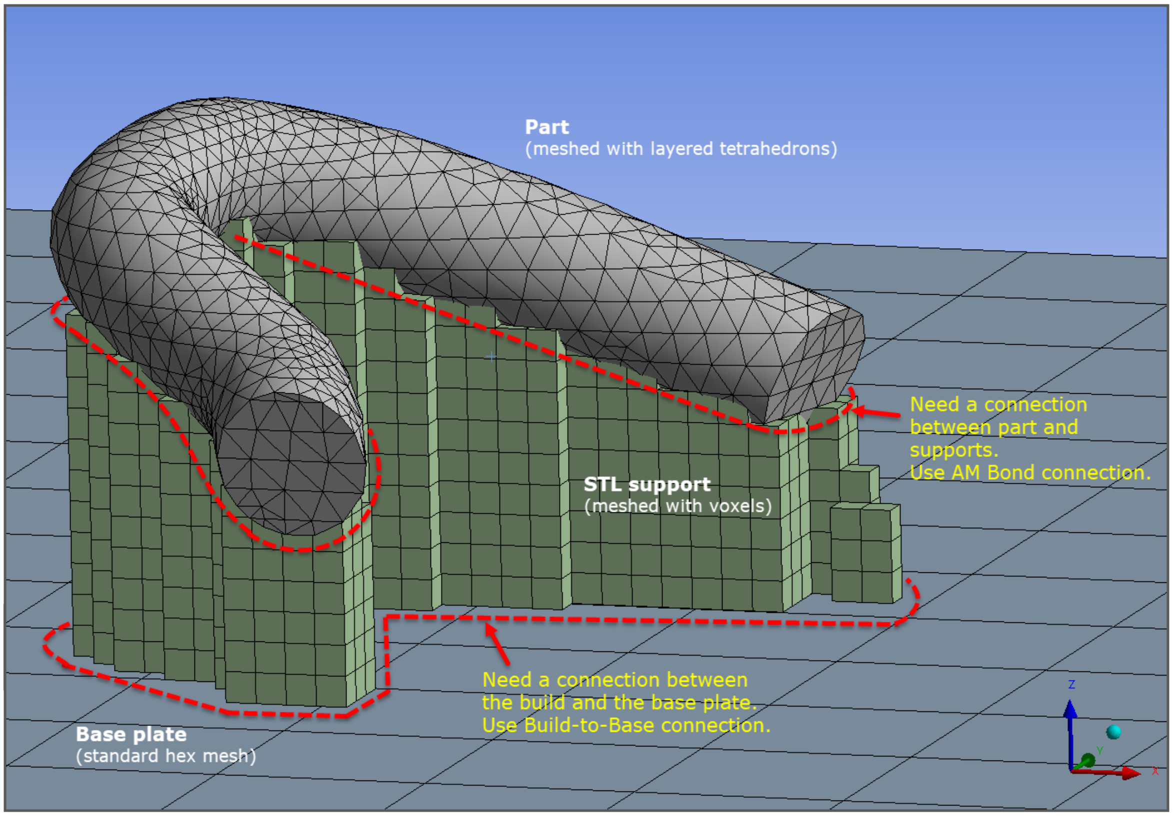
Hint: Look for the tiny icon to the left of an object in the project tree for a clue about its status. For example, if you see a question mark next to an STL Support object, it means you have not scoped it to a part yet (if supports were transferred from Additive Prep) or identified a file name yet (if supports were inserted via an `.stl` file). If you see a yellow lightening bolt, it means you have not generated the mesh for it yet. You will see a green check-mark once you have performed all relevant tasks.

5.8. Define Connections

Go directly to procedural steps. (p. 95)

Connections are the mechanism to ensure that the part, support, and base plate bodies in the simulation are aware of each other and are able to share data (temperatures and displacements) across boundaries. Two connection types that are typically used in AM simulations are Build-to-Base contact connection and AM Bond connection.

- A Build-to-Base connection is a special case of bonded contact between the element faces on the bottom of the build and the element faces on the top of the base.
- An AM Bond connection is used to connect a meshed part to a meshed support when the mesh is non-conformal between them. The internal means of connection is through constraint equations that connect the support nodes to the part elements.



Certain mesh and support combinations in Mechanical result in *conformal* mesh between the part and the supports, in which connections are automatic by definition of a conformal mesh, while other combinations result in *non-conformal* mesh. In these scenarios you will need to create a contact connection between them with an AM Bond.

As depicted in the following table, the part mesh/support combinations that are automatic because of the conformal mesh are scenarios C, E, F, J, and K.

The part mesh/support scenarios that require an AM Bond connection are:

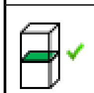
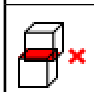
- When the part and/or the Predefined Support is meshed with layered tetrahedrons (scenarios A, D, G, and H)
- When the part is meshed with Cartesian mesh (voxelization option = No) and STL Supports are used (scenario B). STL Supports are automatically meshed with a voxel mesh. In this scenario, we recommend you use the voxelization option to mesh the part instead.

Additive Manufacturing Mesh and Support Options

Support Type	Mesh Type (Part): Layered Tetrahedrons	Mesh Type (Part): Cartesian with voxelization option = No	Mesh Type (Part): Cartesian with voxelization option = Yes [1]
STL Supports [1] • STL Supports are usually volumeless (non-watertight) but can be solid (watertight) and are inserted under the AM Process object. • STL Supports are automatically meshed with the voxelizer regardless of how the part is meshed. • Subsample Rate is used to knock-down properties of supports.	(A) Part is meshed with layered tetrahedrons: • Mesh is nonconformal, connection is not automatic. An AM Bond connection is required between each part and support. Use **Create AM Bond Connections** feature to create all connections.	(B) Part is meshed with Cartesian mesh, voxelization option = No: • Projection Factor of part mesh must be 0 • Mesh is nonconformal, connection is not automatic. An AM Bond connection is required between each part and support. Use **Create AM Bond Connections** feature to create all connections. • Consider using voxelization option = yes instead	(C) Part is meshed with Cartesian mesh, voxelization option = Yes: • Connection between part and support is automatic because of conformal mesh
Predefined Supports • Predefined Supports are solid (i.e., watertight) geometries that are imported with the part geometry. • If support is meshed with layered tetrahedrons, block or user-defined multipliers are used to knock-down properties of support. • If support is meshed with voxelizer, Subsample Rate is used to knock-down properties of support.	(D) Part is meshed with layered tetrahedrons (regardless of mesh type for Predefined Supports): • Requires *unshared* topology between part and support • Mesh is nonconformal, connection is not automatic. An AM Bond connection is required between each part and support. Use **Create AM Bond Connections** feature to create all connections.	(E) Part and Predefined Supports are meshed with Cartesian mesh, voxelization option = No: • Requires *shared* topology between part and support • Connection between part and support is automatic because of conformal mesh (G) Part is meshed with Cartesian mesh, voxelization option = No and Predefined Support is meshed with layered tets • Requires *unshared* topology between part and support • Mesh is nonconformal, connection is not automatic. An AM Bond connection is required between each part and support. Use **Create AM Bond Connections** feature to create all connections.	(F) Part and Predefined Supports are meshed with Cartesian mesh, voxelization option = Yes: • *Unshared* topology between part and support is recommended • Connection between part and support is automatic because of conformal mesh (H) Part is meshed with Cartesian mesh, voxelization option = Yes and Predefined Support is meshed with layered tets • Requires *unshared* topology between part and support • Mesh is nonconformal, connection is not automatic. An AM Bond connection is required between each part and support. Use **Create AM Bond Connections** feature to create all connections.
Generated Supports • Generated Supports are *elements only*, generated straight down from the elements on the meshed part. As such, Generated Supports have no association with any geometric entities. • Block or user-defined multipliers are used to knock-down properties of supports.	(I) Generated Supports are not available if part is meshed with layered tetrahedrons	(J) Part is meshed with Cartesian mesh, voxelization option = No: • Projection Factor of part mesh must be 0 • Connection between part and support is automatic because of conformal mesh	(K) Part is meshed with Cartesian mesh, voxelization option = Yes: • Connection between part and support is automatic because of conformal mesh

[1] The voxelizer does not work on the Linux OS. Therefore, the Cartesian voxelization option and STL Supports are not available on Linux.

Symbols used in the Mesh and Support Options table are shown here.

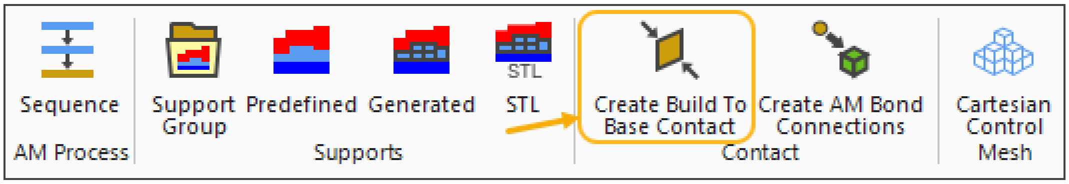
	This mesh/support combination is available
	This mesh/support combination is not available
	Connection between part and support is automatic because of a conformal mesh
	Contact connection between part and support is required

Procedural Steps

Defining connections between entities should be performed after they are meshed. For all scenarios, you must define a connection between the build and the base plate so we will start there.

Connection between the build and the base plate — Build-to-Base:

Highlight the **AM Process** object and then select the **Create Build to Base Contact** option on the context toolbar.

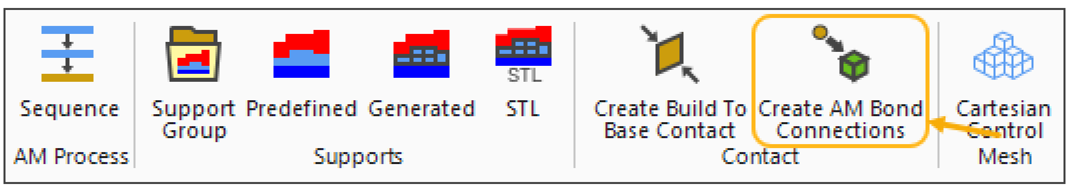


Several things happen. You will see the contact body (red) and target body (blue) views shown in side windows of the Geometry window. The Contact Side is defined as the element faces of the bottom of the build (part and supports). The Target Side is defined as the element faces of the top of the base plate. In the project tree, a Contacts object is added under Connections with a Build to Base object underneath it. Also, named selections have been created that select the element faces at the build-base interface (Build Contact Element Faces and Base Contact Element Faces).

Connections between part and supports — AM Bond:

If you have no supports, or connections are automatic because of a conformal mesh (scenarios C, E, J, and K), skip ahead to the Establish Thermal Analysis Settings (Thermal-Structural System) (p. 113) step.

To create AM Bond connections automatically that will connect all appropriate part and support boundaries, highlight the **AM Process** object and then select the **Create AM Bond Connections** option on the context toolbar.



Several things happen. Named selections have been created for the part and support (such as Build Part and STL Support). You will see the contact body (red) and target body (blue) views shown in side windows of the Geometry window. The Contact is defined as the part body. The Target is defined as the support body. In the project tree, an AM Bond object is added to the tree and renamed to reflect the part and support named selections.

The automatic Create AM Bond Connections option described above works to detect appropriate part and support pairs based on proximity to one another. Be sure to review the contact pairs created by examining the views in the Geometry window and the details of the AM Bond objects in the project tree. If there are multiple parts and/or multiple supports closely packed together on a base plate, it is possible the Create AM Bond Connections action results in incorrect pairings. If you suspect that happened, delete the connections and named selections and create AM Bond contact connections manually, as follows.

To manually create an AM Bond connection that will connect a part to a support:

1. Create a named selection for the part: Right-click the part geometry and choose **Create Named Selection**. Enter a name for the part geometry and click **OK**.

2. Create a named selection for the support:

a. For an STL support, right-click the STL Support object and choose **Create Named Selection of Generated Elements**.

b. For a predefined support, right-click the support geometry and choose **Create Named Selection**. Enter a name for the part geometry and click **OK**.

3. Right-click **Connections** and choose **Insert** > **AM Bond**. Using named selection as the scoping method, choose the *part* named selection as the Contact, and choose the *support* named selection as the Target.

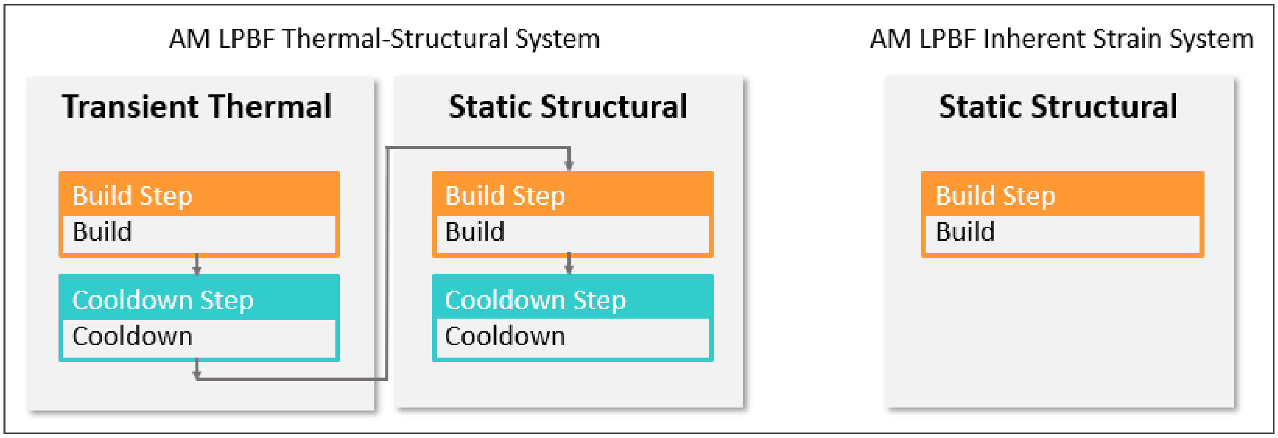
Consider the following guidelines for AM Bond connections:

- Do not use an AM Bond connection to connect a part or support to the base plate. Use the Build-to-Base connection for that.
- One AM Bond connection is required for each part/support combination.
- Only one body should be selected when creating the named selection for the part.
- The identification of contact and target entities as indicated in step 3 above is appropriate for most cases. Reversing the entities identified as contact and target will not only produce a warning message, but will typically produce unexpected and unrealistic deformation. However, if the support mesh is much coarser than the part mesh, say greater than 2:1 proportions, reverse the scoping selection and make the support the contact and the part the target. This is easily done by right-clicking AM Bond and choosing **Flip Contact/Target**.
- The internal constraint equations used to implement AM Bond connections are applied to the nodes of only the *top and bottom* of the supports. This means the connections will be in the Z-direction but not the side-to-side directions.
- The constraint equations will couple the voxel supports to the meshed body even if there is penetration of the support into the part, such as when positive intrusion values are used when generating volumeless supports. You may, though, see anomalies at some of these interfaces, such as temperature higher than melt or lower than ambient, or very localized large displacements at a support-part interface. The overall solution (temperatures, displacements, and stresses) over the build is good and these local anomalies may be ignored.
- You can check the number of constraint equations that are created for each AM Bond connection by viewing the solver output. Under the Transient Thermal system, Solution object, click **Solution Information**. In Details view, Solution Information, choose **Solver Output**. In the worksheet window shown, search on "CE connections."

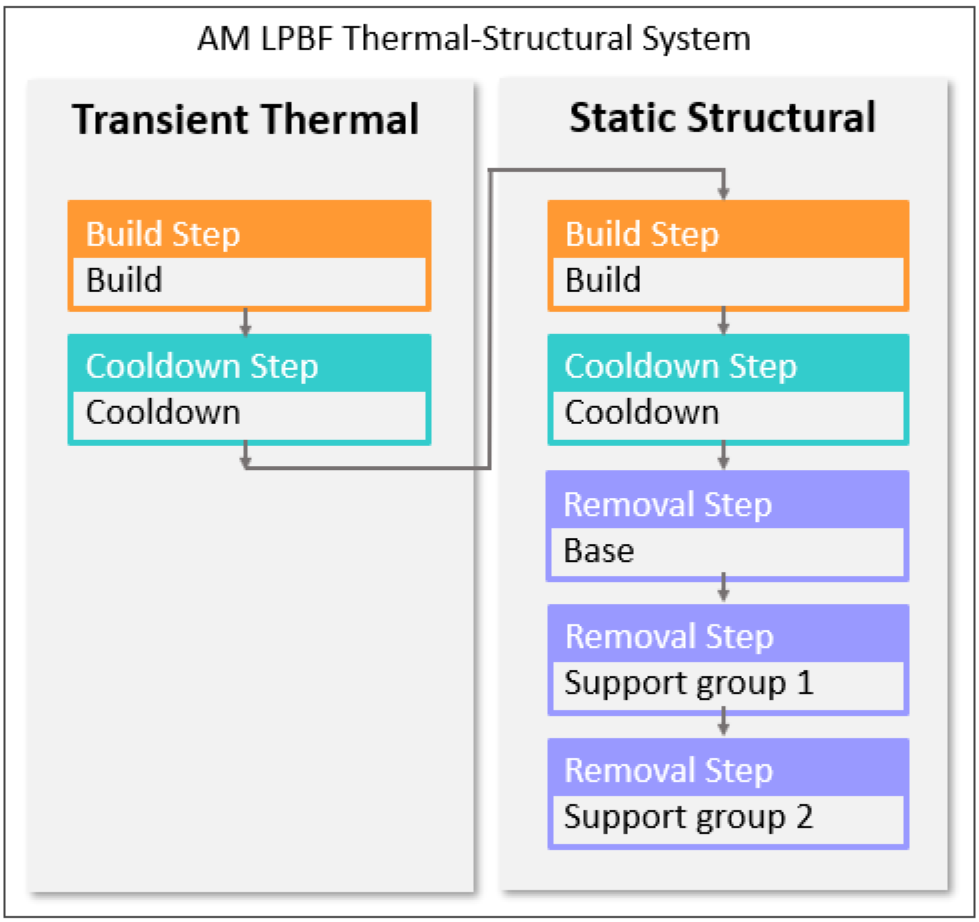
5.9. Define AM Process Steps

Go directly to procedural steps. (p. 100)

The AM process is accomplished through sequential steps that dictate how consecutive solutions are performed in the overall simulation. Minimally, there will always be a build step and usually there is a cooldown step for Thermal-Structural systems. Conceptually, this can be visualized as follows:

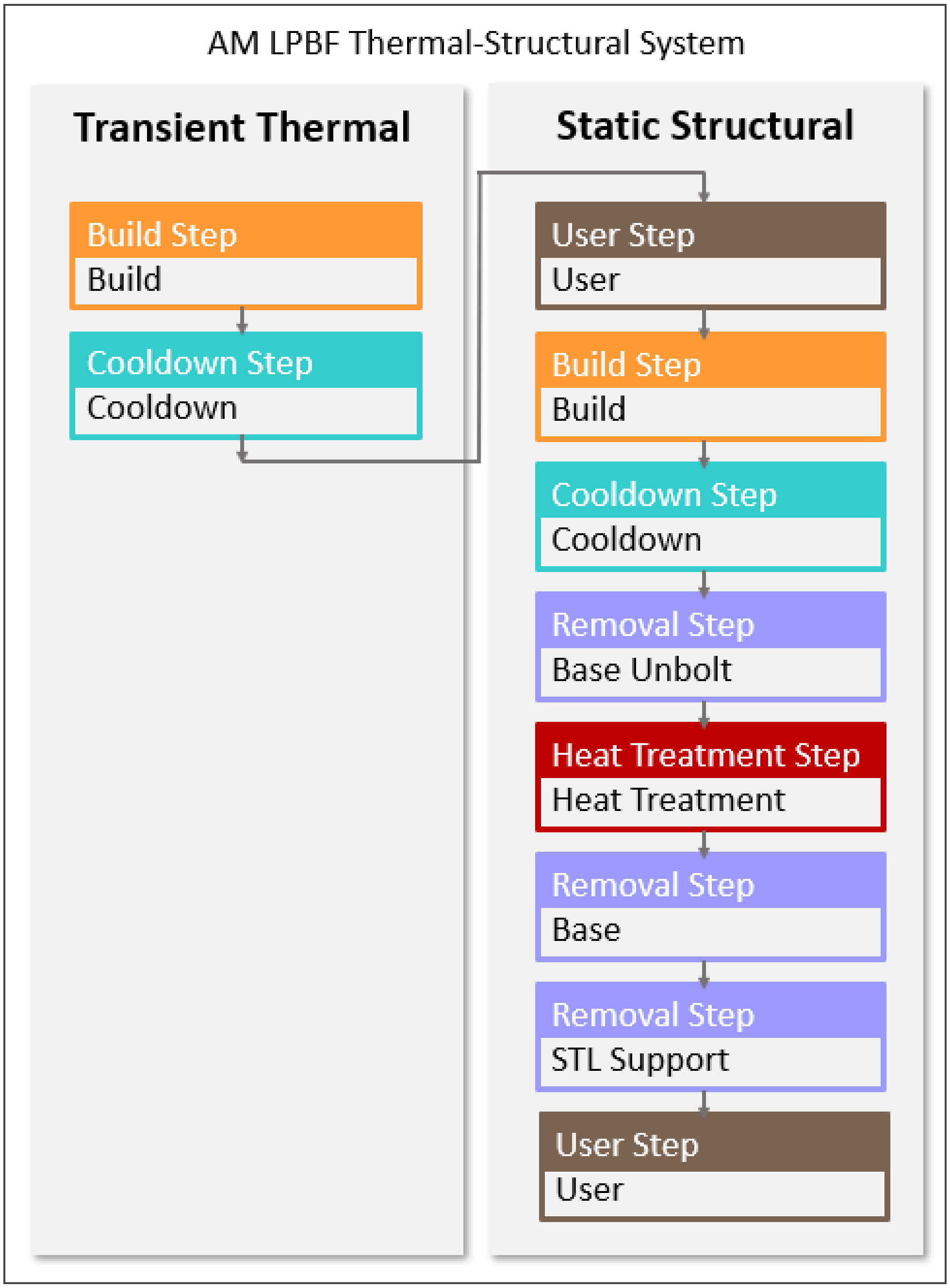


Additional steps may be added to the simulation to account for other phenomena in the AM process, such as the removal of the base plate and/or supports, as shown here for a Thermal-Structural system:



In most cases, base removal is done before support removal. It is important to understand that residual stress and distortion results may be affected by the order in which you remove supports. The Mechanical application can simulate any scenario of support removal.

Other solution steps, such as *base unbolting* and heat treatment steps and user-defined *user steps*, may be added before or after removal steps, as shown here:



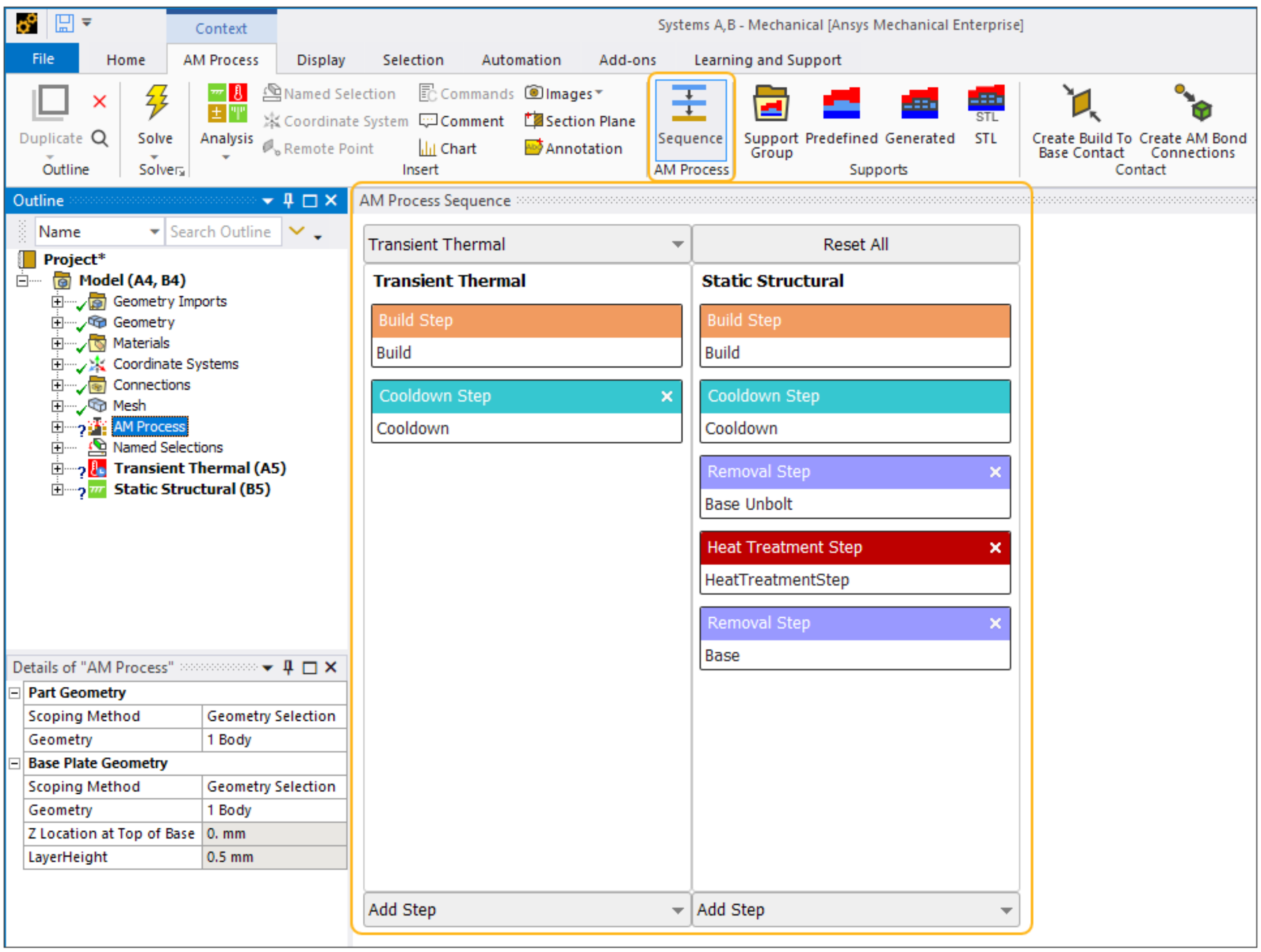
When an additive print is finished, usually the first step is to unbolt the base plate from the machine and transport it for heat treatment or support removal postprocessing. Including this step in the simulation allows more accurate modeling of the deformations in subsequent postprocessing steps. In this step, the constraints for UX, UY, and UZ are removed from the base plate nodes and weak springs are turned on in Analysis Settings. For additional details, see the **AMSTEP** command. Note that unbolting the base prior to heat treatment can be especially important and is recommended to prevent any artificial stress in the base due to fixed boundary conditions and thermal expansion during heating.

A user step includes a solve execution and may be added at various points in the overall sequence. In the application, a user step effectively leaves the AM simulation environment and enters the usual nonlinear "load step" environment. Any valid loading and load step options may be used. One example of a user step is a bolt pretension step before the build.

Procedural Steps

For AM Process Simulations, the Mechanical application provides an alternative to the ordinary step-manipulation in analysis settings by using a custom worksheet called the AM Process Sequence worksheet, or the *sequencer*, as we like to call it. Overall viewing and control of the steps in your simulation is done through the sequencer, shown below. You can access the Sequence button only when the AM Process object is selected in the project tree. You can toggle the display on and off using the button any time the AM Process object is selected.

The sequencer enables you to view and change the steps in the simulation. In the screen image below from an LPBF Thermal-Structural system, the steps in the transient thermal analysis and the downstream static structural analysis are conveniently shown side-by-side. Note that unless you use the LPBF Setup Wizard or a Commands object, using the sequencer is the *only* way to manipulate steps in an AM Process Simulation.



1. Select the **AM Process** object and then select the **AM Process Sequence** option on the context toolbar (or click on Worksheet in the main menu bar).

2. To add a base unbolting step, click **Add Step** on the Static Structural side of the worksheet and select **Base Unbolt**.

3. To add a heat treatment step, click **Add Step** on the Static Structural side of the worksheet and select **Heat Treatment Step**. Additional steps are required to complete the heat treatment step as described in the advanced topic Simulating Heat Treatment after the Build (p. 176).

4. To simulate the removal of the base or supports after cooldown or after heat treatment,

   a. Click **Add Step** at the bottom of the Static Structural side of the worksheet. The dropdown will show the options available for removal, depending on your set-up (Base, Predefined Support, Generated Support, and/or STL Support). You may reorder the steps in the sequence using drag and drop of one step on top of another.

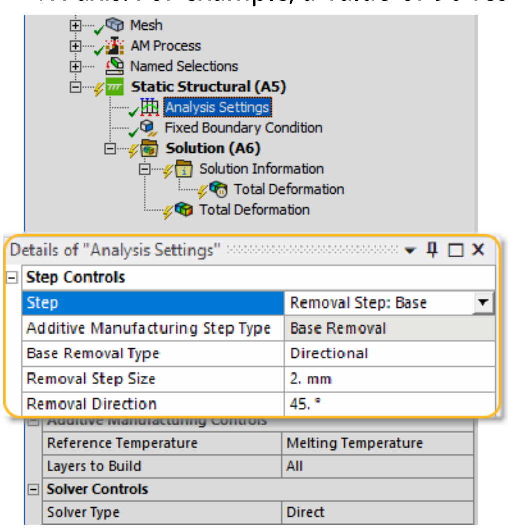
   b. Additional options are available if you choose the base removal step. Under Static Structural, click **Analysis Settings** in the project tree and select the **Removal Step: Base** option from the **Step** drop-down. Choose a **Base Removal Type**, either Instantaneous or Directional.

      - Instantaneous: Simulates instantaneous cutoff of all material at the base (bottom layer of elements only).

      - Directional: Simulates a progressive cutoff of all material at the base (bottom layer of elements only), in which you specify the cut increment and the angular direction for removal from the base.

        Removal Step Size: Distance removed in every cut step. Cannot be 0 or a negative number.

        Removal Direction: Directional cutoff angle on the X-Y plane as measured from the +X axis. For example, a value of 90 results in a cutoff direction in the +Y direction.



5. To add a user step to either analysis, click **Add Step** on the desired side of the worksheet and select **User Step**. You will then need to specify your step conditions, such as an extra boundary condition, etc. If you want to insert a user step, such as for bolt pretensioning, as the *first* step in the sequence before the thermal build step, click **Add Step** and choose **User Step Prior to Build**.

Note:

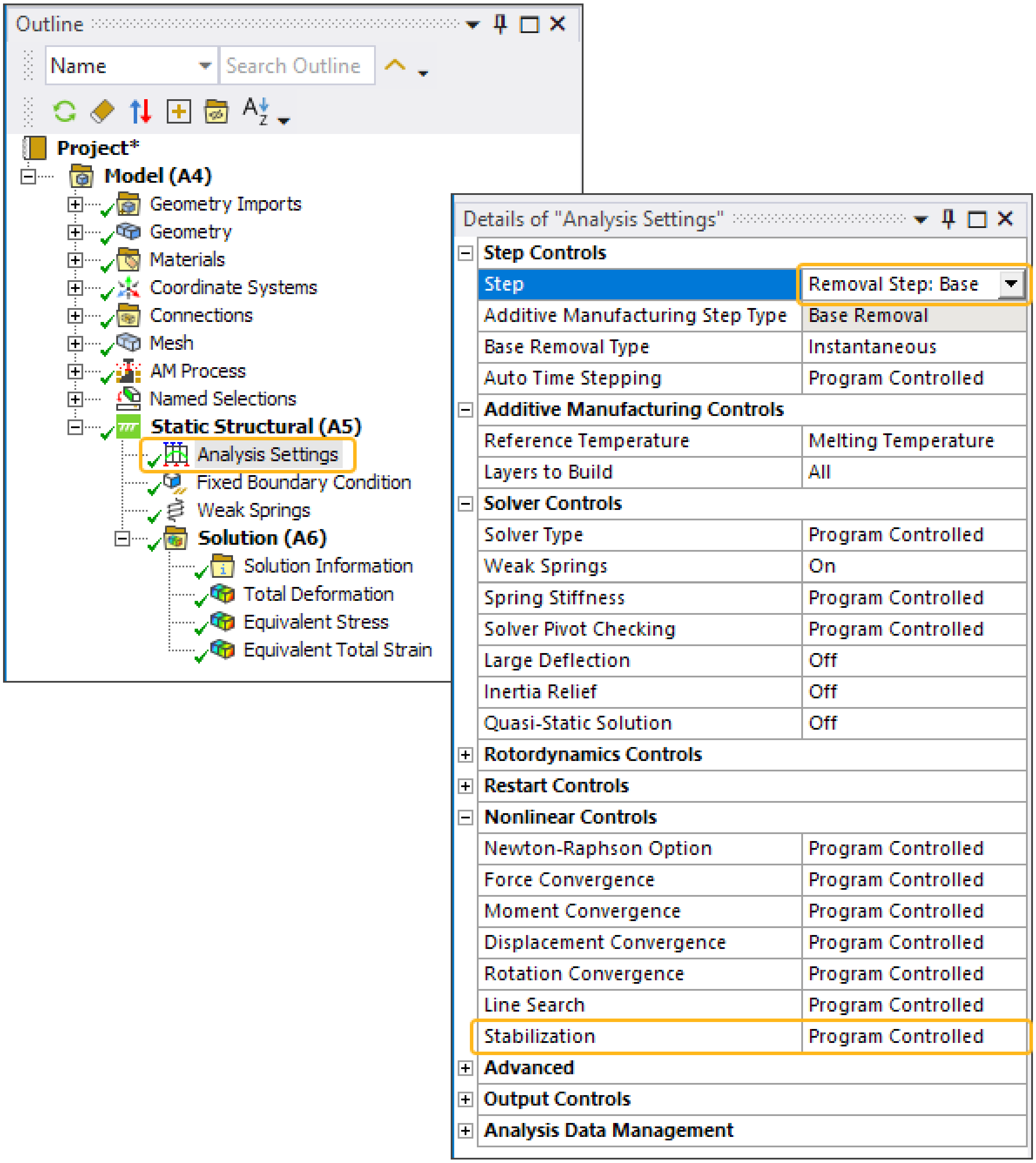
Advanced Ansys users frequently combine Mechanical APDL commands with the automatic execution of models set up in Mechanical for more precise and custom control over a solution. For example, you may insert a Commands object under the Static Structural or Transient Thermal environment objects in the project tree. In the Commands object Details view, there is a useful option to **Issue Solve Command** (either Yes (default) or No). It enables users to better control how the commands in the Commands object are processed relative to the step execution in the sequencer.

Nonlinear Stabilization

The post processing steps of base unbolting and base/support removal are highly nonlinear phenomena. As such, nonlinear stabilization is on by default for these steps to improve convergence. In most cases, stabilization is enough to sufficiently hold the part in place to prevent rigid body motion.

Note the following about stabilization:

- Stabilization is turned on by default only if all AM process steps are set to Program Controlled for Stabilization in Analysis Settings.



- The default behavior is that stabilization is turned on immediately before relevant **AMSTEP** commands and turned off immediately after each relevant **AMSTEP** command so that stabilization is only on for the removal or unbolting step and does not affect heat treatment or user steps.
- Stabilization default values are set as follows (shown with Mechanical APDL commands):
  - For base unbolting: **STABILIZE**,CONSTANT,ENERGY,1.e-004,**ANYTIME**,0.2
  - For base and support removal: **STABILIZE**,CONSTANT,ENERGY,1.e-004,**NO**,0.2

See Stabilization in the *Mechanical User's Guide*, and Using Nonlinear Stabilization and **STABILIZE** in the Mechanical APDL documentation for more information.

5.10. Define Build Settings

Go directly to procedural steps. (p. 109)

In this step, specify the simulation and strain assumptions, process parameters, and conditions related to the machine and the process.

Simulation Settings

Simulation settings include assumptions about the type of simulation and the strain definition.

- **Additive Process**: Laser Powder Bed Fusion. The Laser Powder Bed Fusion (LPBF) process uses thermal energy from a laser or electron beam to selectively fuse powder in a powder bed.

- **Inherent Strain** (Yes or No): An option to use the Inherent Strain method (p. 138). (If you used one of the AM custom systems (p. 67) in Workbench to set up your analysis system, this option is set automatically for you.) If No, the AM simulation uses a linked thermal-structural system in which strains are calculated from material properties and thermal loads. If Yes, an alternative method is used in which strains are based on an experimentally calibrated Strain Scaling Factor. Options under Machine Settings, Calibration Settings, Build Conditions, and Cooldown Conditions differ depending on whether Inherent Strain = Yes or No.

  If **Inherent Strain** = **Yes** (default if AM LPBF Inherent Strain custom system is used):

  – **Inherent Strain Definition**: Assumption about strain behavior reflecting the different ways inherent strain is calculated as an input to the structural solver.

    → Isotropic: Simplifying assumption that a constant, isotropic strain occurs at every location within a part as it is being built. A constant Strain Scaling Factor may be used to scale strain everywhere uniformly to account for calibration of the Mechanical application to a particular machine/material combination.

    → Anisotropic: Uses the same average strain magnitude as isotropic strain, but it subdivides that strain into anisotropic components in the X, Y, and Z directions based on the Global coordinate system.

    → Scan Pattern: Uses the same average strain magnitude as isotropic strain, but it subdivides that strain into anisotropic components based on the local orientation of scan vectors within the part. Scan vectors may be generated internally via a slicing function assuming a rotating stripe scan pattern or input via a build file.

    → Thermal Strain: A method that provides the highest level of fidelity and takes thermal cycling into account at each location within the part.

    - **Thermal Strain Method**: Machine Learning Prediction (only method available at this release)

      Uses a machine learning model prediction of the anisotropic Thermal Strain simulation result from the Ansys Additive application.—Thermal Strain simulations provide the highest level of fidelity by predicting how thermal cycling affects strain accumulation at each location within a part. The simulation follows the full laser path on every layer, and is based on the machine process parameters (power, scan speed, beam diameter, etc.).—The machine learning model has been trained to predict the Thermal Strain result much faster than simulation. It can be one to three orders of magnitude faster than Thermal Strain simulation in Additive Print in calculating the strain that is passed to the structural solver. Speedup increases with part size, scan area, and melt pool size. See Thermal Strain - Anisotropic in the *Additive Print and Science User's Guide*.

Important:

The Machine Learning Prediction feature requires a Structures AI+ license.

- **Machine Learning Model**: A list of materials that were used to train the ML prediction, in particular, the materials validated for the Additive application. Choose the material that most closely matches your material assignment in Engineering Data. ML models may be based on different material properties than those in Engineering Data. The ML models are used to generate loading strains. Materials in Engineering Data are used for the structural analysis.

- **Layer Height**: By default, the program uses the Layer Height specified when meshing as the super layer height. Occasionally you may want to increase the size of the super layer to something coarser than what was used when you created the mesh. Use the Layer Height option here to do that. Specifying a new Layer Height does not affect the existing mesh.

Calibration Settings

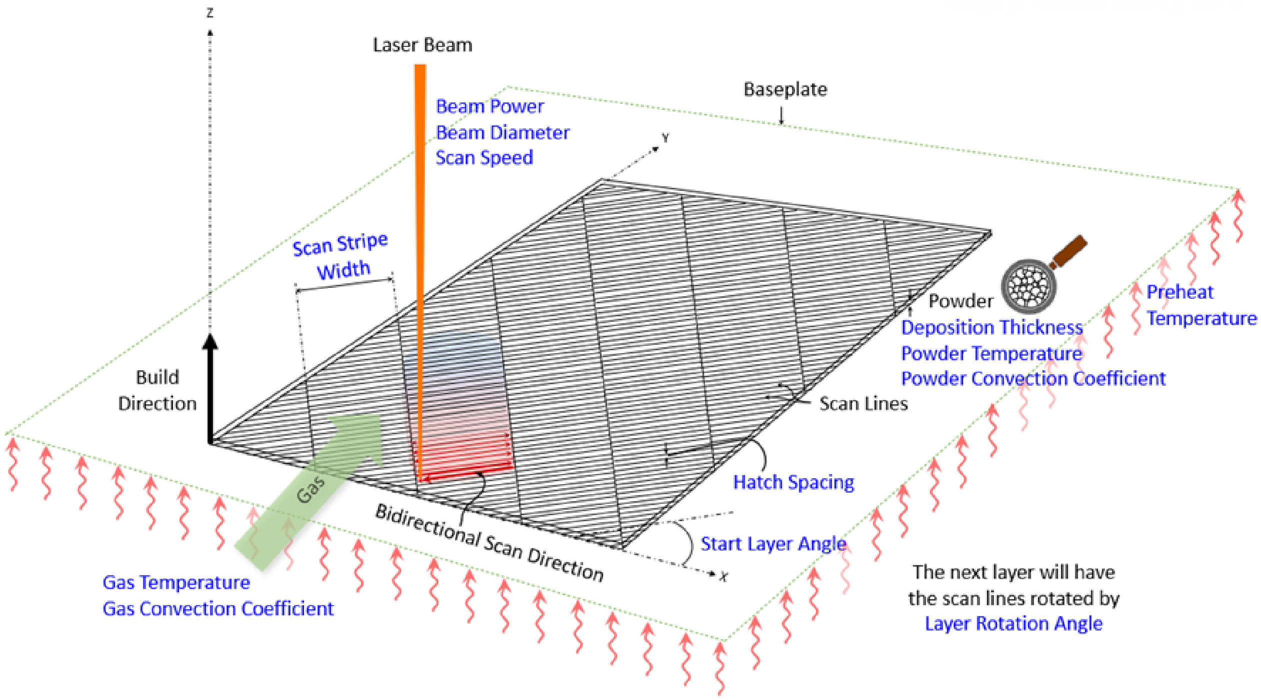
Strain Scaling Factors are optional inputs that scale the inherent strains in an inherent strain simulation, or the thermal strains in a thermal-structural simulation, by a given value. They are usually determined from calibration experiments. Defaults are shown below for the various strain definitions:

Strain Definition	Strain Scaling Factor(s)	Default Value
Thermal-Structural simulation	Thermal Strain Scaling Factor	1
Isotropic - Uniform strain in all directions	Strain Scaling Factor	1
Anisotropic - Strain can be scaled based on the Global coordinate system	Strain Scaling Factor X Strain Scaling Factor Y Strain Scaling Factor Z	1 1 1
Scan Pattern and **Thermal Strain** - Strain can be scaled based on the local orientation of scan vectors (parallel and perpendicular to scanning direction and in the build direction)	Parallel ASC[a] Perpendicular ASC Vertical ASC	1.5 0.5 1

[a] ASC = Anisotropic Strain Coefficient

Machine Settings

Machine settings refer to process parameters for your AM machine setup. Inputs for machine settings are used to calculate the real, physical time duration of the build process so that the cooldown time can be determined. Every simulation except a simple isotropic Inherent Strain simulation uses some, or all, of the parameters shown in blue in the following figure and defined below.



Machine parameters for a typical rotating stripe scan pattern

Input Parameter	Definition
Heating Method	A simulation assumption related to how the material is heated for each new element layer as it is added (made alive). Choose either Melting Temperature (default) or Power. Visible for simulations with thermal solves only (i.e., not Inherent Strain). See Build Settings for details.
Start Layer Angle	The orientation of fill rasters on the first layer of the build. It is measured from the X axis, such that 0 degrees results in scan lines parallel to the X axis. The starting layer angle is commonly set to 57 degrees. Must be between 0 and 180°.
Layer Rotation Angle	The angle at which the major scan vector orientation changes from layer to layer. It is commonly 67 degrees. Must be between 0 and 180°.
Scan Stripe Width	Width of the sections, called stripes, into which the geometry is sliced. The stripes are scanned sequentially to break up what would otherwise be very long continuous scan vectors. Scan Stripe Width is commonly set to 10 mm wide. Must be between 1 and 100 mm.
Hatch Spacing	The average distance between adjacent scan vectors when rastering back and forth with the laser. Hatch spacing should allow for a slight overlap of scan vector tracks such that some of the material re-melts to ensure full coverage of solid material. For Machine Learning strain definition, must be between 60 and 1000 microns.

Input Parameter	Definition
Deposition Thickness	The thickness of added powder material in every pass of the recoater blade. Specifically, use the amount the base plate drops between layers. For Thermal Strain strain definition, must be between 10 and 100 microns.
Scan Speed	The average speed at which the laser spot moves across the powder bed along a scan vector to melt material, excluding jump speeds and ramp up and down speeds. For Thermal Strain strain definition, must be between 350 and 2500 mm/sec and the recommended range is between 500 and 2500 mm/sec.
Beam Power	The power setting for the laser in the machine. Must be between 50 and 700 Watts. The recommended range is between 50 and 500 Watts.
Beam Diameter	The width of the laser on the powder or substrate surface defined using the D4σ beam diameter definition. Usually this value is provided by the machine manufacturer. Sometimes called laser spot diameter. Must be between 20 and 140 μm. The recommended range is between 80 and 120 μm.

- If **Inherent Strain** = **Yes** (default if AM LPBF Inherent Strain custom system is used):
  - **Scan Pattern Definition** (visible if Inherent Strain Definition = Scan Pattern or Thermal Strain): How the scan pattern is defined, either generated using a rotating stripe pattern (default) or input via a build file.
    - → Generated: Start Layer Angle and Layer Rotation Angle as defined above. Scan Stripe Width is also visible if Inherent Strain Definition = Thermal Strain. These inputs define an internally generated scan pattern.
    - → Build File: Machine Type and Build File Path inputs become available with this option. These inputs specify an external build file to be used.
      - Machine Type: Specifies the machine or OEM associated with the build file specified. Options are Additive Industries, EOS, HB3D, Renishaw, Sisma, SLM, and Trumpf.
      - Build File Path: Location of a .zip file containing the scan pattern file(s), and an stl of the part geometry. See Build File Requirements (p. 109).
  - Beam Diameter, Beam Power, Deposition Thickness, Hatch Spacing, and Scan Speed as defined above. If a build file is specified for Scan Pattern Definition, deposition thickness is determined from the file.
- If **Inherent Strain** = **No** (default unless AM LPBF Inherent Strain custom system is used):
  - Hatch Spacing, Deposition Thickness, and Scan Speed as defined above.
  - Dwell Time: The span of time from the end of the laser scan of one layer to the start of the laser scan of the next layer. It includes the time required for recoater-blade repositioning and powder-layer spreading.

- Dwell Time Multiplier: The dwell-time multiplier accounts for more than one part in the build. If they are the same part arranged in the same orientation on the build plate, the multiplier is the number of parts.

- Number of Heat Sources: For multiple-beam printers, specifies the number of lasers. This divides the amount of time it takes to scan a layer by the number of heat sources specified.

Build Conditions

Build conditions are the settings pertaining to the environment in the build chamber around the build as it is being printed, including the preheat temperature. For **Inherent Strain** = **Yes**, only preheat temperature is required and only with Inherent Strain Definition = Thermal Strain. For **Inherent Strain** = **No**, additional inputs are required to account for convection.

During a laser powder bed fusion (LPBF) print process, almost all the heat dissipation is conducted through the part back to the build plate rather than out through the unmelted powder surrounding the part. Many users ignore the small effect of heat loss through powder but you may choose to model it as equivalent heat convection.

The Gas/Powder Temperature option allows the use of Preheat Temperature for Gas Temperature and Powder Temperature.

Input Parameter	Definition
Preheat Temperature	The starting temperature of the build plate. Used when Inherent Strain = Yes and Inherent Strain Definition = Thermal Strain, and when Inherent Strain = No. For Thermal Strain strain definition, must be between 20 and 500 °C and the recommended range is between 20 and 200 °C.
Gas Temperature	Temperature of the gas in the build enclosure.
Gas Convection Coefficient	Convection coefficient from the build to the enclosure gas. The convection is applied only to the top of a newly laid layer.
Powder Temperature	Temperature of the newly added powder.
Powder Convection Coefficient	Effective convection coefficient from the sides of the build to the powder bed. To estimate, divide the conduction property of the powder by a characteristic conduction length into the powder (for example, a quarter of the distance from the build boundary to the build-chamber wall).
Powder Property Factor	A knockdown factor used to estimate the powder properties. The Mechanical application applies the factor to the solid material properties to estimate the properties of the material in its powder state. The powder-state properties are used in the newly added layer during the heating of the new layer (before its subsequent solidification and cooldown) prior to the next layer being applied. The default value is 0.01. This powder knockdown factor is also used if powder is explicitly modeled (p. 194) in the build.

Cooldown Conditions

Cooldown conditions are the settings pertaining to the environment in the build chamber around the build in the cooldown step after the last layer is printed. Available only if **Inherent Strain** = **No**.

- Room Temperature
- Gas Temperature, Gas Convection Coefficient, Powder Temperature, and Powder Convection Coefficient as defined above, with an option to use Room Temperature for Gas Temperature and Powder Temperature.

Procedural Steps

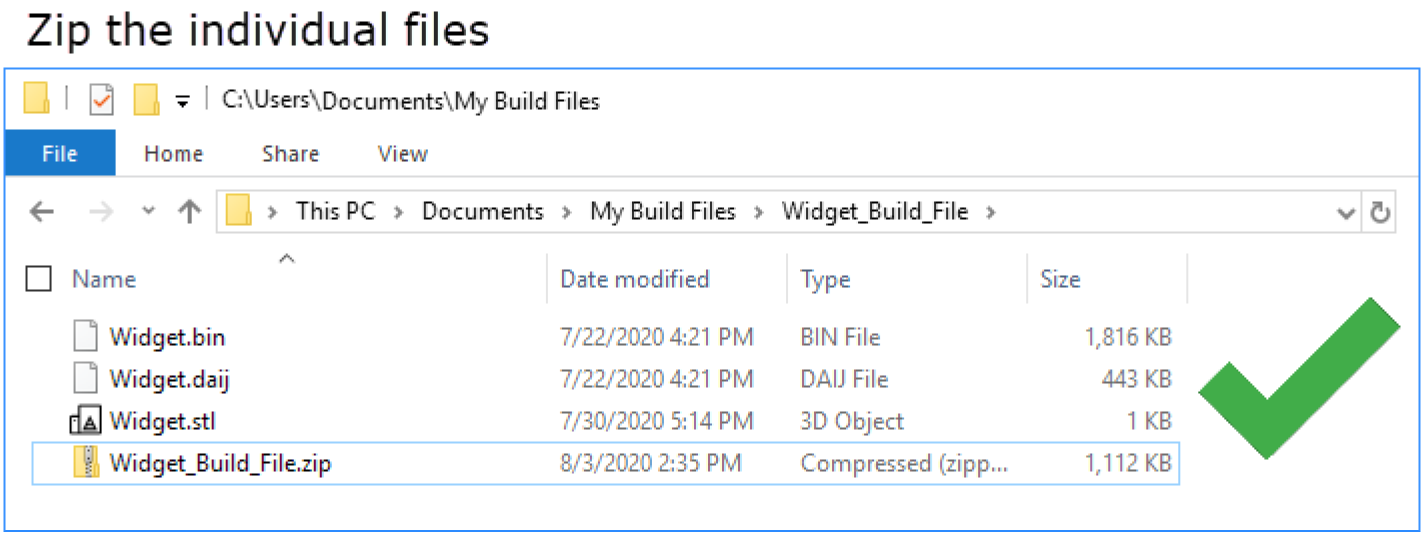
You can go back to Workbench and select the Engineering Data tab at any time to see properties for your chosen material.

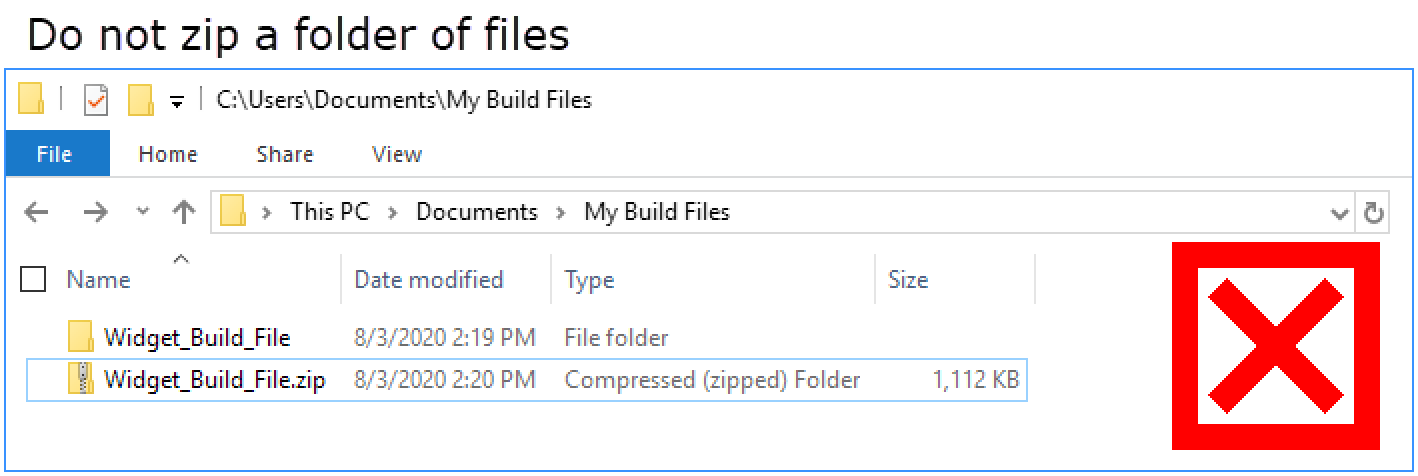
1. Select the **Build Settings** object (under AM Process object).
2. In Details, enter values for all the items under **Simulation Settings**, **Calibration Settings**, **Machine Settings**, **Build Conditions**, and **Cooldown Conditions**. You can load a pre-saved file of build settings if you have one – right-click **Build Settings** and select **Load Build Settings**. (We provide generic sample files for our AM materials in the ANSYS Inc directory, for example: C:\Program Files\ANSYS Inc\v231\aisol\DesignSpace\DSPages\SampleData\AdditiveManufacturing. In this example, v231 indicates Release 2023 R1. Note that the ANSYS Inc directory on your machine may not be on the C drive.)
3. Right-click the **Build Settings** object and select **Save Build Settings** to save your inputs to an `.xml` file that can be reused.

Build File Requirements

Listed here are the general requirements for build files, which are the same as those required for Additive Print. Additional machine-specific requirements are provided in the expandable topics below the general requirements.

- We define a build file as a .zip file containing, at a minimum: one `.stl` file for the part geometry and one machine-specific print file defining the scan vectors. Supported machine manufacturers include Additive Industries, EOS, HB3D, Renishaw, Sisma, SLM, and Trumpf. Ansys may add additional options as we continue to work with more machine partners.
- Build files from Additive Prep are automatically created with a file name of "ansys_additive_print.zip.
- The build file is a .zip file. Do not nest the files to be zipped within a folder, as a folder structure is not readable. Rather, zip the individual files together as shown in the following figure.





General requirements for the part geometry `.stl` file

- `.stl` files within the .zip file must have units of mm and be positioned in the same location as the part in the Mechanical application.
- Multiple parts can be used, but they must all be within one `.stl` file.
- The baseplate should not be included in the part geometry `.stl` file.
- Supports should not be included in the part geometry `.stl` file nor as a separate `.stl` file.

General requirements for the machine file

- One laser head is assumed for the simulation. Build files with multiple lasers are not supported. When you import a build file with multiple lasers, the application handles it differently depending on the machine. In most cases, the application either generates an error or ignores the extra lasers upon import.

- One set of process parameters is used in the simulation. If the build file contains multiple parameter sets, such as different scan speeds and laser powers for the part hatches versus the support hatches, the part hatch parameters will take precedence.
- Only one part layer thickness (Deposition Thickness) is allowed.
- Only one support layer thickness is allowed, and it must be equal to, or a multiple of, the part layer thickness.
- Regardless of how they are defined in the build file, the scan sequence is always simulated from the inside out, that is, from hatch to contour scans. However, the appropriate order is maintained within the hatch area and within a contour. For example, if the build file order is: contour line 1 → contour line 2 → hatch line 1 → hatch line 2, it will be changed to be hatch line 1 → hatch line 2 → contour line 1 → contour line 2.
- Scan vectors marked as contour will not be simulated. The definition of contour/hatch is established by the software that creates the build file. If contour-like scan vectors are marked as hatch they will be simulated and results may not be as expected.
- Build files for simulations that use the Machine Learning Prediction strain definition only support *stripe* scan patterns.

Additive Industries Build Files

A build file for an Additive Industries machine should be a zip file containing:

- Part = `*.stl` file
- Two machine files are required, one `*.daij` file and one `*.bin` file.

Notes:

- *Support scan vectors will be ignored and will not be simulated.* An Additive Industries build file with supports has not been tested.
- Only a stripe scan pattern has been tested.

EOS Build Files

A build file for an EOS machine should be a zip file containing:

- Part = `*.stl` file
- Machine = `*.openjz` file. Mechanical uses version 2.8 of the EOS API.

Notes:

- *You must have an EOSPRINT 2 license, in the form of a dongle or from a license server, from EOS in order to import an EOS build file.*. If you are using a dongle, do not unplug it until the after strains have been generated.
- EOS M100, M290 and M400 single-laser machines are supported.

Known Issues and Limitations

- When preparing the `.openjz` file in EOSPRINT, start height and end height (<height> element) should not be zero since it is never the position of any layer. The build file may fail to generate strains in Mechanical.
- When attempting to import an older EOS build file, the application may reject it since version 2.8 of the EOS API is stricter in regards to the OpenJob format. If this happens, you may see an error.

HB3D Build Files

A build file for an HB3D machine should be a zip file containing:

- Part = `*.stl` file
- Machine = `*.h3d` file. Mechanical uses version 1.0 of the `*.h3d` file specification.

Notes:

- An HB3D build file ignores the last layer when the distance from its Z coordinate to the Z max is less than one layer thickness.

Renishaw Build Files

A build file for a Renishaw machine should be a zip file containing:

- Part = `*.stl` file
- Machine = `*.mtt` file. Mechanical uses version 1.06 of the `*.mtt` file specification.

Sisma Build Files

A build file for a Sisma machine should be a zip file containing:

- Part = `*.stl` file
- Machine = `*.wza` file. Mechanical uses version 3.0.9 of the `*.wza` file specification.

SLM Build Files

A build file for an SLM Solutions machine should be a zip file containing:

- Part = `*.stl` file
- Machine = `*.slm` file. Mechanical uses version 1.10 of the `*.slm` file specification.

Trumpf Build Files

A build file for an Trumpf machine should be a zip file containing:

- Part = `*.stl` file
- Machine = `*.wza` file. Mechanical uses version 3.0.9 of the `*.wza` file specification.

5.11. Establish Thermal Analysis Settings (Thermal-Structural System)

Go directly to procedural steps. (p. 114)

This step requires you to think about your simulation's end goals. What do you want to investigate? And specifically, what data do you want to see from the transient thermal portion of an LPBF Thermal-Structural simulation? Thermal analysis settings enable the customization of various options during the transient thermal solution, including identifying which items to solve for.

The transient thermal analysis will determine the temperature history during the build process. These temperatures will then be used in a static structural analysis to determine the build distortions and stresses. The Mechanical application will automatically determine all the steps and times needed for time integration in the simulation.

Options to consider include:

- **Hotspot** (p. 114) – A significant cause of issues in additive manufacturing can be attributed to overheating during the build. This typically occurs when there are changes in cross sectional area and there is not enough material to pull heat away from a certain region. Overheating can lead to poorly shaped melt pools, affecting the part's material characteristics and porosity. You can check for the overheating issue with simulation by checking the temperature that each layer cools down to before a new layer is added. Add an LPBF Hotspot result item to your project so that the appropriate data is written out during solution. The LPBF Hotspot result tool configures the Export Layer End Temperature option so that the temperature of a layer just before a new layer is applied is written out to a file.

- **Layers to Build** (p. 114) – An option is available to limit the number of layers to build in the simulation, that is, to simulate only a partial build process. This may be useful if you want to examine results in the lower portion of the build if you suspect there will be cracks or blade interference there.

- **Other Output Controls** (p. 114) – For an AM Process Simulation, your results file will grow in size very quickly, so we recommend you keep the default output control settings that will suppress calculation of thermal flux, nodal forces, Euler angles, volume and energy, and other miscellaneous items. Nodal temperatures are stored at all time points by default but you can change that option so that temperatures are stored at the last heating and cooling steps only, or every N number of finite element layers.

Procedural Steps

Usually it is appropriate to leave most analysis settings set to "program-controlled." These settings are determined when you insert the AM Process object into the project tree. There are a couple of settings to note related to an AM Process Simulation, as described below.

1. To solve for hotspots, use the LPBF Hotspot tool in the LPBF Process Add-on.

   Load the LPBF Process Add-on (p. 8) if you have not already done so. From the LPBF Process tab, click the **LPBF Hotspot** button. A new result object, called AM Hotspot, is added in the project tree under Solution in the Transient Thermal environment. (Alternatively, you can right-click Solution and select Insert > AM Hotspot.)

   The LPBF Hotspot result tool automatically sets the Export Layer End Temperature option to Yes in the Analysis Settings object under Output Controls. It will write out to a file the temperature of a layer just before a new layer is applied. Node numbers and x, y, z locations are also written. The output is not written to the results file but rather to a tab-delimited file called `AMResults.txt`.

2. If you want to limit the number of layers to build in the simulation, select the **Analysis Settings** object under the Transient Thermal object and in Details, **Additive Manufacturing Controls**, change the **Layers to Build** to your desired value. Your results file will show results from the beginning layer only through the specified layer. (Hint: If you change the value of Layers to Build and you want to set it back to All, enter 0.) Pay attention to the active step as indicated at the top of the Details panel under Step Controls. The number of Layers to Build in the Cooldown Step will always equal the number of Layers to Build in the Build Step.

3. In Details of **Analysis Settings**, review the **Output Controls** and adjust them according to your needs.

   - To change the option of when to store element results for the Build Step, select **Store Results At** and choose All Time Points, Last Heating and Cooling Steps, or Every N Layers. (Layers refers to finite element layers, not powder deposition layers.)

5.12. Apply Thermal Boundary Conditions (Thermal-Structural System)

In the Build Settings step of our workflow, we have already accounted for convection between the build and the gas in the chamber and convection between the build and the unmelted powder around the part—in both the Build Step and the Cooldown Step. There is one other area of heat transfer to consider and that is associated with the base plate.

During the build process, the plate is typically heated on the bottom to maintain a constant, slightly elevated temperature. You may insert a temperature constraint or convection surface to simulate this during the build step. If the plate is not heated, perhaps apply a room condition convection surface, or set the surface boundary condition to adiabatic.

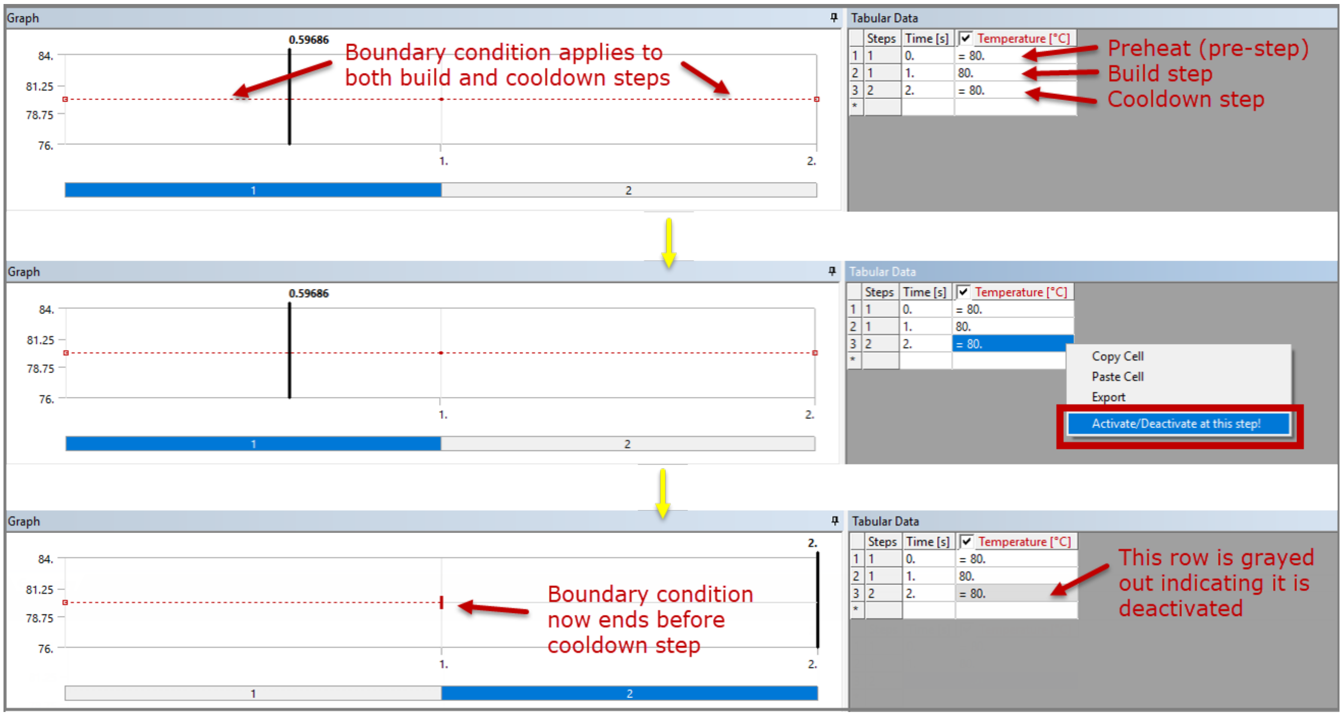
After the print process is complete, the base plate heating is removed and the built part cools to room temperature. This is simulated by a room-temperature convection surface applied to the bottom of the base plate in the cooldown step.

The time duration of the cooldown step is estimated based on the average part temperature at the end of the build, its volume, and material properties. Using the convection values and room temperature, the Mechanical application solves the simple heat transfer equation to get an estimate of the cool down time. It is only an estimate so you can extend it if required or put in a preferred time in analysis settings. One final step is performed to force all the temperatures to room temperature for the subsequent distortion and stress calculation.

Procedural Steps

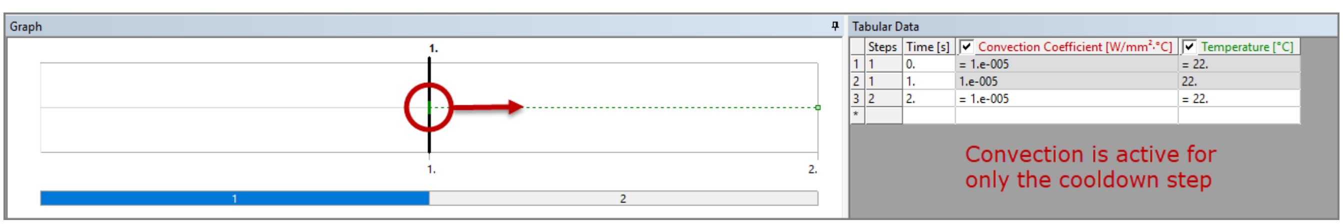
To apply a temperature to the bottom surface of the base plate for the build step:

1. Right-click the **Transient Thermal** object and then select **Insert** > **Temperature**.
2. Select the bottom surface of the base plate (geometry selection) and hit **Apply** in the Details view.
3. In Details under Definition, **Temperature**, enter a temperature value for **Magnitude**. (In our example we use 80°.)
4. In the **Tabular Data** area of the interface, notice the temperature value you just entered is applied for all sequence steps. (The first row of tabular data is considered T=0, the preheat condition.) In the printing process, since the heat is removed after the part is printed, you'll need to remove the temperature boundary condition for the cooldown step of the simulation. Right-click in the third row and click **Activate/Deactivate at this step!** which will deactivate (remove) the elevated temperature for the cooldown step.



To apply room-temperature convection to the bottom surface of the base plate for the cooldown step:

1. Right-click the **Transient Thermal** object and then select **Insert** > **Convection**.
2. Select the bottom surface of the base plate (geometry selection) and hit **Apply** in the Details view.
3. In Details under Definition, **Film Coefficient**, enter a value for the convection coefficient.
4. Under **Ambient Temperature**, enter a value of 22°. (This may be the default.)
5. In the **Tabular Data** area of the interface, select the first two rows, right-click and select **Activate/Deactivate at this step!** which will deactivate (remove) the convection for the build step.



It is important to use separate load objects for each change in thermal boundary conditions, as was done in the example above where temperature and convection boundary conditions were inserted separately. You may be tempted to combine them into one temperature object and specify 80° for the first two rows/steps and 22° for the last row/step, but doing so is incorrect and will cause problems.

Important:

For thermal loads in AM analyses, variation in tabular values within a single load object may cause incorrect temperature profiles during the build stage. Use a separate load object to define each change in thermal boundary conditions.

5.13. Solve the Transient Thermal Analysis (Thermal-Structural System)

Some users prefer to solve the thermal analysis first to observe how the simulation is progressing while others prefer to set up everything and run both the thermal and structural analyses at once. Either way is perfectly acceptable. To run them both at once, simply execute a solve from the static structural side and the transient thermal analysis will be run first automatically.

Since the solution can take significant time to complete, a solution status window is provided that indicates the overall progress of the solution. Other solution trackers and tools allow you to i) view the actual output from the solver, ii) graphically monitor items such as convergence criteria, and iii) view the temperature contour as the build progresses. You can also monitor some result items such as temperature at a node as the solution progresses.

Procedural Steps

1. To set up a plot of overall temperature that you can update throughout the solution, under Transient Thermal, Solution, **Solution Information**, select **Insert** > **Temperature Plot Tracker**.
2. To initiate the solution, under Transient Thermal, highlight the **Solution** object, right-click and select **Solve**.
3. Use the temperature plot tracker occasionally or continuously, as follows:
    - Whenever you want to see the updated temperature, right-click the **Temperature** plot tracker object and select **Update Result**. Or,
    - Right-click the **Temperature** plot tracker object and select **Switch to Automatic Mode**. This will show a continuous, live display as the solution progresses. The plot tracker object must be selected in order to see the live display.

5.14. Establish Structural Analysis Settings

Go directly to procedural steps. (p. 118)

In this step, think about your end goals for the simulation and what data you want to see written out from the structural portion of the simulation. Structural analysis settings allow for the customization of various options during the static structural solution, including identifying which items to solve for.

Options to consider include:

- **Recoater Interference** (p. 118) – A significant concern in additive manufacturing is whether the build will print successfully without experiencing recoater interference (sometimes called blade crash). This phenomenon occurs when the powder recoater blade hits into a portion of the built part that has deformed extensively because of residual stresses. Usually the result is a stopped process and a failed build. If checking for recoater interference is a simulation goal, add an LPBF Recoater Interference result now to your project so that the appropriate data is written out during solution. The LPBF Recoater Interference result tool configures the Export Recoater Interference output control option so that the Z-deformation of a layer just before applying a new layer is written out to a file.

- **High Strain** (p. 119) – When the strain in a part exceeds the elongation a material can withstand, a failure can occur resulting in cracking throughout the part or supports. The LPBF High Strain result tool allows you to identify regions of the part that may be prone to forming cracks during or after the build process by highlighting critical strain values. The LPBF High Strain result tool configures the Export High Strain output control option so that the maximum equivalent strain experienced during the build process is written out to a file.

- **Reference Temperature** (p. 119) – The Reference Temperature is the temperature at which thermal strains do not exist in a material. In the simulation of the AM process, our assumption is that each finite element super layer is added (with the element birth/death technique) at the melting temperature for the material and is initially strain-free. (We set Tref = Tmelt by default.) As the build cools, thermal strains develop. The static structural analysis will use the temperature results of the transient thermal analysis to compute the displacements, stresses, strains, and forces due to these induced thermal strains.

- **Relaxation Temperature** (p. 119) – If you will be simulating a heat treatment process such as annealing after the build, you will probably want to specify a Relaxation Temperature. Lower than the melting temperature, the relaxation temperature is the temperature at which strains begin to soften. (Using a creep model in Engineering Data is an alternative stress relaxation mechanism.) Refer to the advanced topic Simulating Heat Treatment after the Build (p. 176) for details.

- **Layers to Build** (p. 119) – An option is available to limit the number of layers to build in the simulation, that is, to simulate only a partial build process. This may be useful if you want to examine results in the lower portion of the build if you suspect there will be cracks or blade interference there. The number specified here must not be more than the number of layers to build used in the thermal analysis step if you are performing a thermal-structural simulation.

- **Other Output Controls** (p. 119) – Your structural results file will grow in size very quickly, so we recommend you keep the default output control settings that will suppress calculation of contact data, nodal forces, Euler angles, volume and energy, and other miscellaneous items. Consider suppressing the calculation of stresses and strains if you are only interested in distortion, as these data, in particular, will easily increase the size of the results file and it may become unmanageable. Additional controls allow you to specify when to store results for the build step: for all layers, for the last heating and cooling steps, or every N layers.

Procedural Steps

As with the thermal analysis, many analysis settings in the structural analysis are program controlled.

1. To solve for recoater interference, use the LPBF Recoater Interference tool in the LPBF Process Add-on.

   Load the LPBF Process Add-on (p. 8) if you have not already done so. From the LPBF Process tab, click the **LPBF Recoater Interference** button. A new result object, called LPBF Recoater Interference, is added in the project tree under Solution in the Static Structural environment. (Alternatively, you can right-click Solution and select Insert > LPBF Recoater Interference.)

   The LPBF Recoater Interference result tool automatically sets the Export Recoater Interference option to Yes in the Analysis Settings object under Output Controls. It will write out to a file the z-deformation of a layer just before a new layer is applied. Node numbers and x, y, z locations are also written. The output is not written to the results file but rather to a tab-delimited file called `AMResults.txt`.

2. To solve for high strain areas, use the LPBF High Strain tool in the LPBF Process Add-on.

   Load the LPBF Process Add-on (p. 8) if you have not already done so. From the LPBF Process tab, click the **LPBF High Strain** button. A new result object, called LPBF High Strain, is added in the project tree under Solution in the Static Structural environment. (Alternatively, you can right-click Solution and select Insert > LPBF High Strain.)

   The LPBF High Strain result tool automatically sets the Export High Strain option to Yes in the Analysis Settings object under Output Controls. It will write out to a file the maximum equivalent strain experienced during the build process. Node numbers and x, y, z locations are also written. The output is not written to the results file but rather to a tab-delimited file called `AMHighStrain.txt`.

3. Select the **Analysis Settings** object under the Static Structural object and in Details, note the **Additive Manufacturing Controls**. The Reference Temperature is automatically set to the material's melting temperature. (You can see this value in Engineering Data for your chosen material.)

4. If you added a Heat Treatment Step in the AM Process Sequence worksheet (p. 100), you will probably want to specify a Relaxation Temperature. Under Additive Manufacturing Controls, change the **Relaxation Temperature** to **User Specified** and change the **Value** to the appropriate temperature. For additional steps to complete the Heat Treatment Step, see the advanced topic Simulating Heat Treatment after the Build (p. 176).

5. To limit the number of layers to build in the simulation, under Additive Manufacturing Controls change the **Layers to Build** to your desired value. The number specified must not be more than the number of layers to build used in the thermal analysis if you are performing a thermal-structural simulation.

6. Review the **Output Controls** and suppress items not of interest.

   - Turn off calculations of Stress and Strain by changing those options from Yes to No. (Hint: Simply double-clicking in the Yes box changes it to No.)

   - To change the option of when to store results, select **Store Results At** and choose All Layers, Last Heating and Cooling Steps, or Every N Layers.

5.15. Apply Structural Boundary Conditions

Usually, the only boundary condition that must be applied is a fixed support boundary condition on the build plate. This is described in step 1 in the procedural steps below.

If base unbolting and/or removal steps (p. 97) are included in your simulation, the fact that nonlinear stabilization (p. 102) is on by default should be enough to loosely hold parts in place to prevent rigid body motion when the base is removed. However, if you think you need additional constraints, consider using weak springs (first choice) or directly applying fixed boundary conditions on three nodes of the part (second choice) to prevent rigid body motion. These steps are described in Additional Constraint Options for Removal Steps (p. 120).

Procedural Steps

1. To fix the bottom surface of the base plate, right-click the **Static Structural** object and then select **Insert** > **Fixed Support**. Select the bottom surface of the base plate (geometry selection) and hit

Apply in the Details view. If you have a more detailed geometry for the base, such as one with bolt holes that you want to explicitly simulate, apply the boundary conditions to affix the plate accordingly.

Additional Constraint Options for Removal Steps

Weak Springs Option

As a first option, insert a Weak Springs object (p. 245) into an LPBF Static Structural simulation to prevent rigid body motion when the default nonlinear stabilization is insufficient. In most cases, simulations should be able to reach convergence without this object, but it may be necessary on some occasions. When added, the object automatically scopes all nodes in the build at the interface of the parts/supports and the base. It attaches weak springs on the scoping (or a subset of the scoping) to hold the part in place without significantly affecting results.

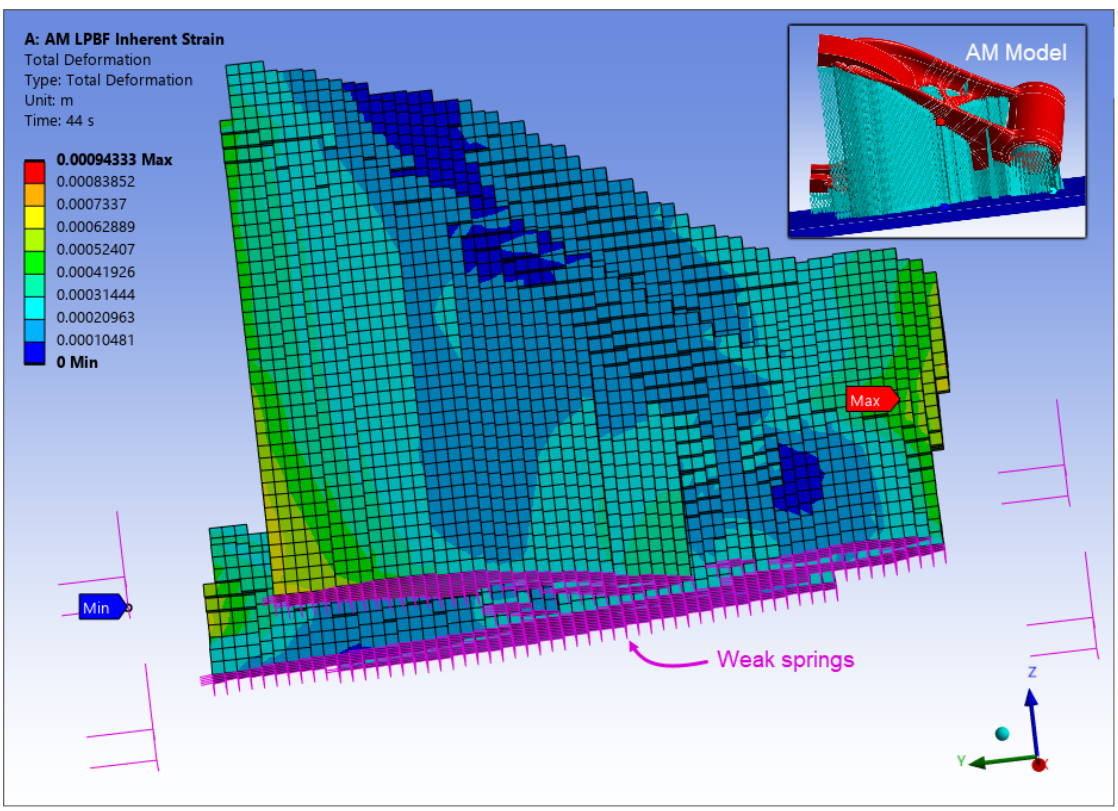
1. To add weak springs, click the **Weak Springs** button in the LPBF Process ribbon. By default, nodes in the build geometries (part and support) at the interface with the base will be scoped to the weak springs object. Weak springs can also be scoped to the base or other non-build geometries (p. 196). You should not scope build elements *above the base* to this object.

2. Weak springs are generated at the solve. To visualize where they are, before solving, do the following:

    a. Under **Analysis Settings** in Solver Controls, set Weak Springs to **On**.

    b. Under Static Structural > Solution, click **Solution Information**, and set **Visible on Results** to **Yes** in the FE Connection Visibility options.

    After doing this, the weak springs should be visible when viewing result objects like Total Deformation.



Not every node scoped will get weak springs applied to it, it is dependent on how you set the Spring Creation (p. 245) setting. Each node that does get springs applied will have three springs, one in the x-direction, one in the y-direction, and one in the z-direction. You should ensure that the stiffness value is low enough that it is not affecting simulation results other than holding the parts in place. It is not recommended to scope weak springs above the interface of the build and base. Weak springs above that interface can affect the build process leading to incorrect results. You will see a warning if you have nodes or faces scoped above the build-base interface.

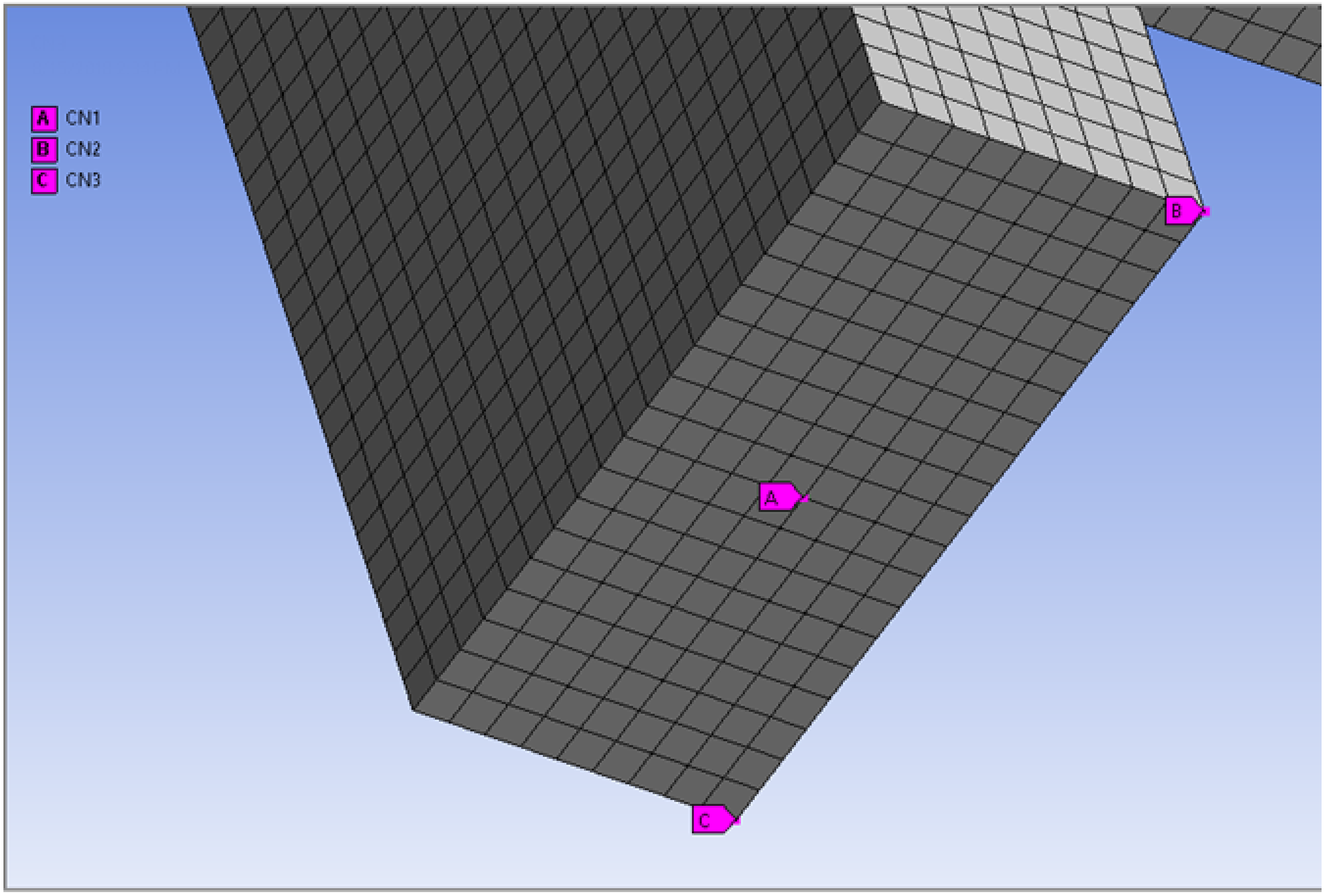
Three Nodes Option

In cases where both stabilization and weak springs are undesired for convergence, consider constraining the part at three nodes to prevent rigid body motion. Be aware that you will be constraining the nodes to their *displaced position at the end of the build*, rather than constraining them to 0. (For example, if the node displaced 0.1 mm during the simulation, the constraint will keep the node at 0.1 mm.) Use a Commands object to do this, as follows:

1. You may want to hide the base body during this process. Right-click the base body and select **Hide Body**.

2. Zoom in to the area of the part where you will apply constraints. Switch to Node mode of selection.

3. Create nodal named selections: Right-click **Model** in the project tree, and select **Insert** > **Named Selection**. Select a node on the bottom of the part, perhaps toward the middle of the part. You should choose nodes that are unimportant to your results. Perhaps spread them out over the bottom of the part but *the nodes must not be co-linear*. Your node selection will affect displacement results but not stresses or strains. Click **Apply** in Details. Right-click the newly created **Selection** in the project tree (under Named Selections) and select **Rename**. Name it to something short and descriptive, such as CN1 (for constrained node 1). Do the same step two more times and rename them to CN2 and CN3.



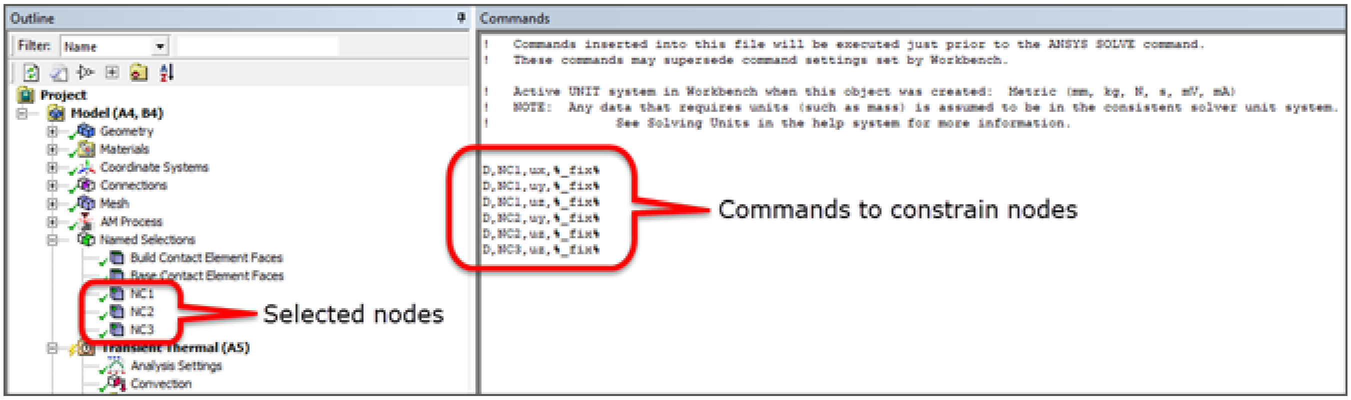
4. Create commands to constrain the nodes: Click the **Static Structural** object and select **Commands** from the context menu. Or, right-click and select **Insert** > **Commands**. In the Commands window that is displayed, copy and paste, or type, the following:

```
D,CN1,UX,%_FIX%
D,CN1,UY,%_FIX%
D,CN1,UZ,%_FIX%
D,CN2,UY,%_FIX%
D,CN2,UZ,%_FIX%
D,CN3,UZ,%_FIX%
```

   Constrained node 1 is fixed in UX, UY, and UZ and holds the part in place. Constrained node 2 is fixed in UY and UZ, and constrained node 3 is fixed in UZ; these prevent rotation. Selection of CN1 can have a big effect on the displacement throughout the part. See the **D** command, which defines degree-of-freedom constraints at nodes, for more information.

5. Match the commands to the base removal sequence step: Select the newly created **Commands** object under Static Structural and, in Details, change the **Step Selection Mode** to **By Identifier**. Change the **Step Number** to **Removal Step: Base**. This ensures the constraints are applied just before the base removal step.

6. Remember to make the base plate visible again if you hid the body. Right-click the base body and select **Show Body**.



5.16. Solve the Static Structural Analysis

You initiate the structural solution in this step. Watch convergence plots or a live tracker of the total deformation as the solution progresses.

Procedural Steps

1. To set up a plot of overall deformation that you can update throughout the solution, under the Static Structural object, **Solution Information**, select **Insert** > **Deformation Plot Tracker**.

2. To initiate the solution, under Static Structural, highlight the **Solution** object, right-click and select **Solve**.

3. Use the deformation plot tracker occasionally or continuously, as follows:

   - Whenever you want to see the updated deformation, right-click the **Total Deformation** plot tracker object and select **Update Result**. Or,

   - Right-click the **Total Deformation** plot tracker object and select **Switch to Automatic Mode**. This will show a continuous, live display as the solution progresses. The plot tracker object must be selected in order to see the live display.

4. To observe convergence plots during solution, click **Solution Information** and in the Details panel, under **Solution Output**, choose from among the selections in the drop-down list, such as **Force Convergence**.

Restarts

Multiframe restarts for additive manufacturing analyses can be set up for any AM step specified by the **AMSTEP** command except the build step. Restarts with additive manufacturing otherwise follow the capabilities outlined here.

5.17. Review Results

For any simulation in Mechanical, once a solution is available you can view a contour plot or animate the results (p. 126). There are solutions at many time points in a transient analysis and you can display the variation of a result item at a location over time. So for a LPBF Thermal-Structural simulation you can view a result at a location on the build over the build history.

In addition to those general results, the following are result items of particular concern for AM simulations:

- Recoater Interference (p. 127) – The LPBF Recoater Interference result tool allows you to identify excessive deformation in the Z direction that may lead to interference with the powder spreading mechanism during printing.
- High Strain (p. 129) – The LPBF High Strain result tool allows you to identify regions of the part that may be prone to forming cracks during or after the build process by highlighting critical strain values.
- Hotspot (p. 130) – The LPBF Hotspot result tool is used to identify areas of overheating that may result in problematic thermal conditions. This result is available for thermal-structural simulations only.
- Deformation Along a Line (p. 131) – At times you may want to obtain results along an edge or other specific location of the part to determine if the part will be within tolerances or to compare to distortion measurements made after the part is built. This technique is used when calibrating Strain Scaling Factors (p. 200), for instance. There are several ways to obtain this result.
- 3D Printing Time Estimate (p. 132) – Get an estimate of the real 3D printing time.

Procedural Steps

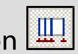
Recall that you used Output Controls under Analysis Options and/or AM-specific result items to control which items are solved for in the simulation. Perhaps you set up results trackers in the solution step to observe the results being processed during the solution. When reviewing results *after* the solution, you will generally need to *evaluate* results to obtain them for viewing. Evaluating results is a way to retrieve them from the data that was stored during solution. Only data items that were solved for may be retrieved in this way.

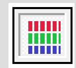
There are many different ways to view results but some of the ones most commonly used for additive manufacturing are described here:

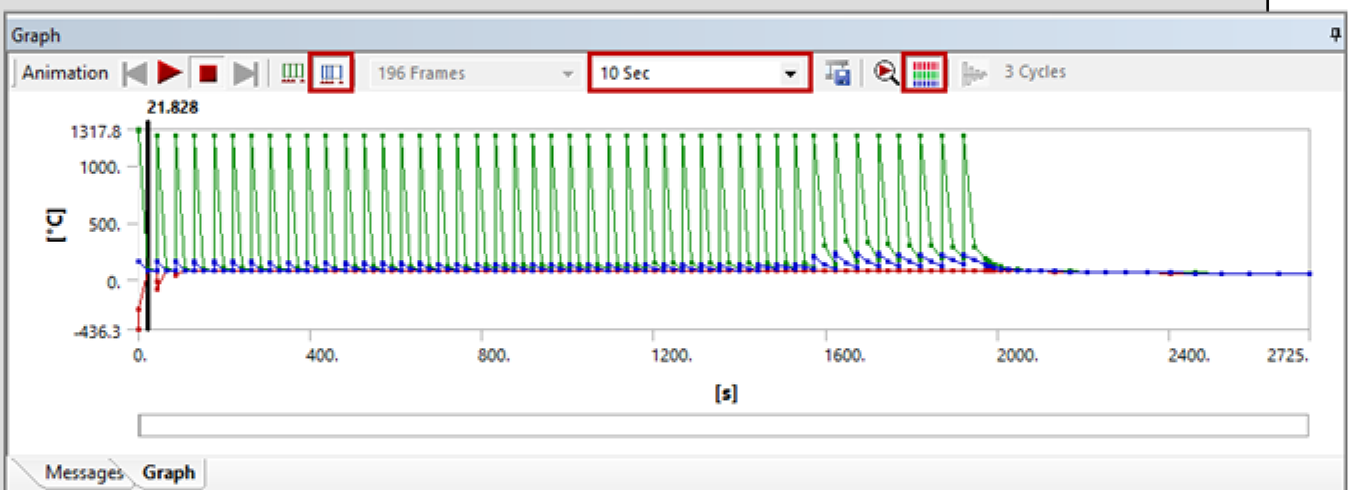
Animate Thermal Results - Temperatures (Thermal-Structural System)

1. In the project tree under Transient Thermal, select the **Solution** object and then, in the context menu, click **Thermal** and then **Temperature**. Or right-click **Insert** > **Thermal** > **Temperature**. A Temperature object is created and becomes active.

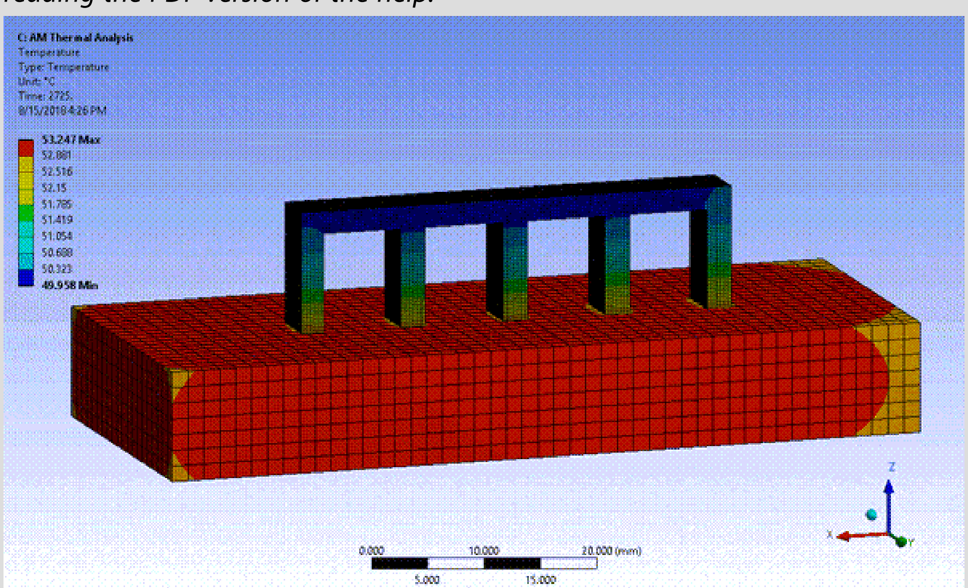
2. Right-click **Temperature** and select **Evaluate All Results**. This retrieves the temperature data and displays it in both graph and tabular form.

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**. For more information about animation controls, see Animation in the *Mechanical User's Guide*.



The following is an animated GIF. Refresh the page to see the animation. View online if you are reading the PDF version of the help.



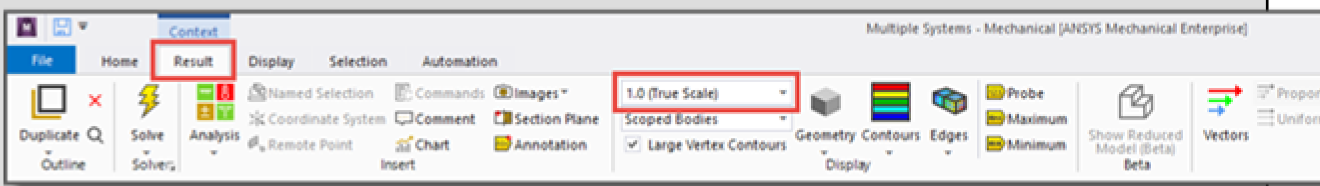
Animate Structural Results - Total Deformation

1. In the project tree under Static Structural, select the **Solution** object and then, in the context menu, click **Deformation** and then **Total**. A Total Deformation object is created and becomes active.

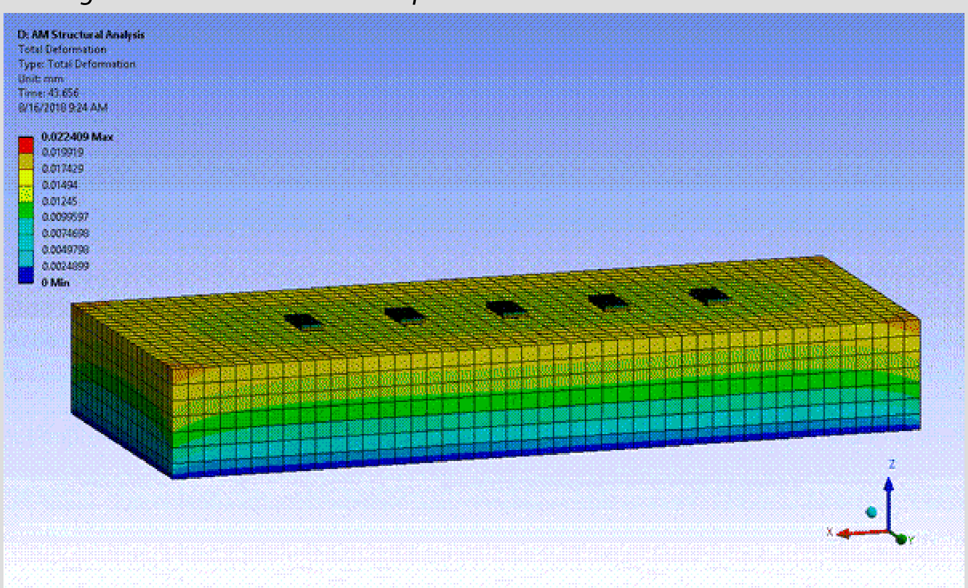
2. Right-click **Total Deformation** and select **Evaluate All Results**. This retrieves the deformation data and displays it in both graph and tabular form.

3. Use the animation controls at the top of the Graph window.

4. It is important (and fun!) to change the magnification scale of the display in the Geometry window so you can get a true sense (or an exaggerated sense) of the deformation in your build. With Total Deformation highlighted in the project tree, from the **Result** context tab, select the **True Scale** option from the drop-down menu. Animate the results using the animation controls. Select other scales from the drop-down menu to see exaggerated results.



The following is an animated GIF. Refresh the page to see the animation. View online if you are reading the PDF version of the help.



Check for Recoater Interference

You need to define some sort of threshold for the level of Z-deformation that is considered to be recoater interference. In the LPBF Recoater Interference Details view, change the **Threshold Definition** to one of these three options:

- **None** - The result tool will show pure Z-deformation in the build, where the value at each point corresponds to the Z-deformation when that material was a new layer.

- **Layer Thickness Based** (default) - The result tool uses the Deposition Thickness (set in the Build Settings object for some simulation types, otherwise set it here) and a new input field, Powder Packing Density, to determine the thresholds for warning and critical deformation. Specifically, if the Z-deformation for any layer is equal to or greater than the Deposition Thickness it is considered a warning. If the Z-deformation is equal to or exceeds the value of Deposition Thickness/Powder Packing Density, it is considered critical. Powder Packing Density may be dependent on the powder particle size, material, spreading mechanism, or other factors.

- **User Defined** - You define your own warning and critical thresholds.

Once you have chosen your Threshold Definition, right-click the **LPBF Recoater Interference** object and select **Evaluate All Results**. The resulting display shows the model with absolute Z-deformation values but with color contours based on the threshold criteria you selected. For the Layer Thickness Based and User Defined options, contours are shown as follows:

- Blue = Safe (Layer Thickness Based: Z-deformation is less than Deposition Thickness)

- Green = Warning (Layer Thickness Based: Z-deformation is equal to or greater than the Deposition Thickness)

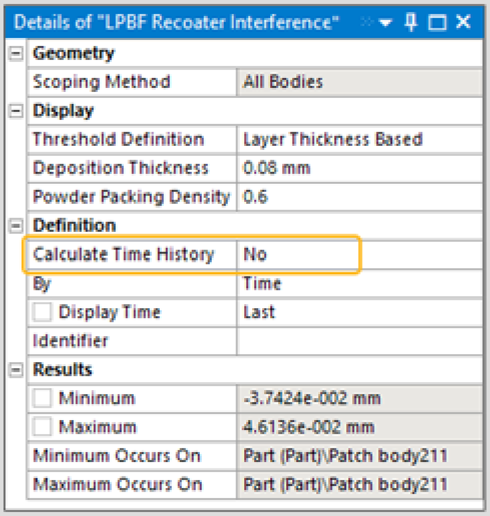
- Red = Critical (Layer Thickness Based: Z-deformation is equal to or exceeds the value of Deposition Thickness/Powder Packing Density)

Important:

The data displayed is based on the Z deformation of each layer right before the next layer is added. It does not represent the "final state" of deformation at the end of the cool-down step.

Note:

To ensure a quick, dynamic calculation of results, Calculate Time History is set to No by default for this result item. Set **Calculate Time History** to **Yes** and then right-click **Evaluate All Results** to see the time history results, such as the layer-by-layer animated build. For large models, the calculation may take some time.


Details of "LPBF Recoater Interference"
Geometry
Scoping Method | All Bodies
Display
Threshold Definition | Layer Thickness Based
Deposition Thickness | 0.08 mm
Powder Packing Density | 0.6
Definition
Calculate Time History | No
By | Time
Display Time | Last
Identifier
Results
Minimum | -3.7424e-002 mm
Maximum | 4.6136e-002 mm
Minimum Occurs On | Part (Part)\Patch body211
Maximum Occurs On | Part (Part)\Patch body211

Check for High Strain Areas

The LPBF High Strain result shows the maximum equivalent strain experienced during the build process. It can help to identify regions at risk of cracking.

Just as you do for the recoater interference result, consider defining a threshold for the level of strain that is considered to be of concern for your material. In the LPBF High Strain Details view, set the **Threshold Definition** to one of these options:

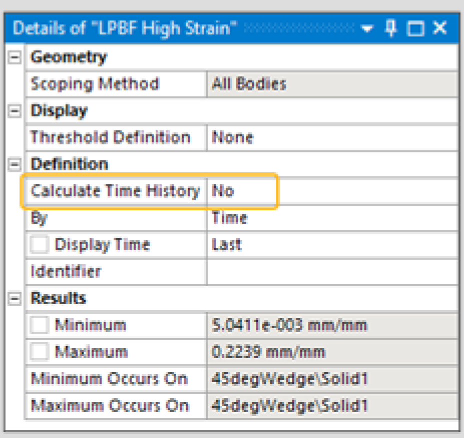
- **None** (default) - The result tool will show pure maximum equivalent strain in the build, where the value at each point corresponds to the strain when that material was a new layer.
- **User Defined** - You define your own warning and critical thresholds.

Once you have chosen your Threshold Definition, right-click the **LPBF High Strain** object and select **Evaluate All Results**. The resulting display shows the model with maximum equivalent strain values. For the User Defined option, color contours are displayed based on the threshold criteria you selected. Contours are shown as follows:

- Blue = Safe
- Green = Warning
- Red = Critical

Note:

To ensure a quick, dynamic calculation of results, Calculate Time History is set to No by default for this result item. Set **Calculate Time History** to **Yes** and then right-click **Evaluate All Results** to see the time history results, such as the layer-by-layer animated build. For large models, the calculation may take some time.



Check for Hotspots (Thermal-Structural System)

The LPBF Hotspot result is applicable for the thermal portion of an LPBF Thermal-Structural simulation. Use the result item after the thermal portion, at least, is solved.

Right-click the **LPBF Hotspot** object and select **Evaluate All Results**. The resulting display shows the temperature of each layer right before the next layer is added. The worst hotspots are going to be the areas with the highest temperature for that layer.

Read-only result data show the minimum and maximum temperature values and where they occur (part or support).

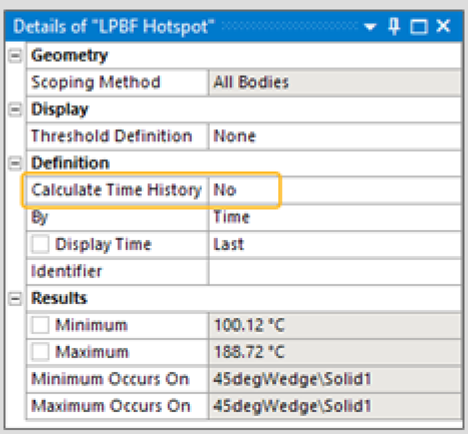
Results are localized (based on nodal values), not averaged across the layer. Use a section plane to reveal hotspots inside the part.

Important:

The data displayed is based on the temperature that the build cools down to at the end of each layer right before the next layer is added. It does not represent the "final state" of temperatures at the end of the cooldown step.

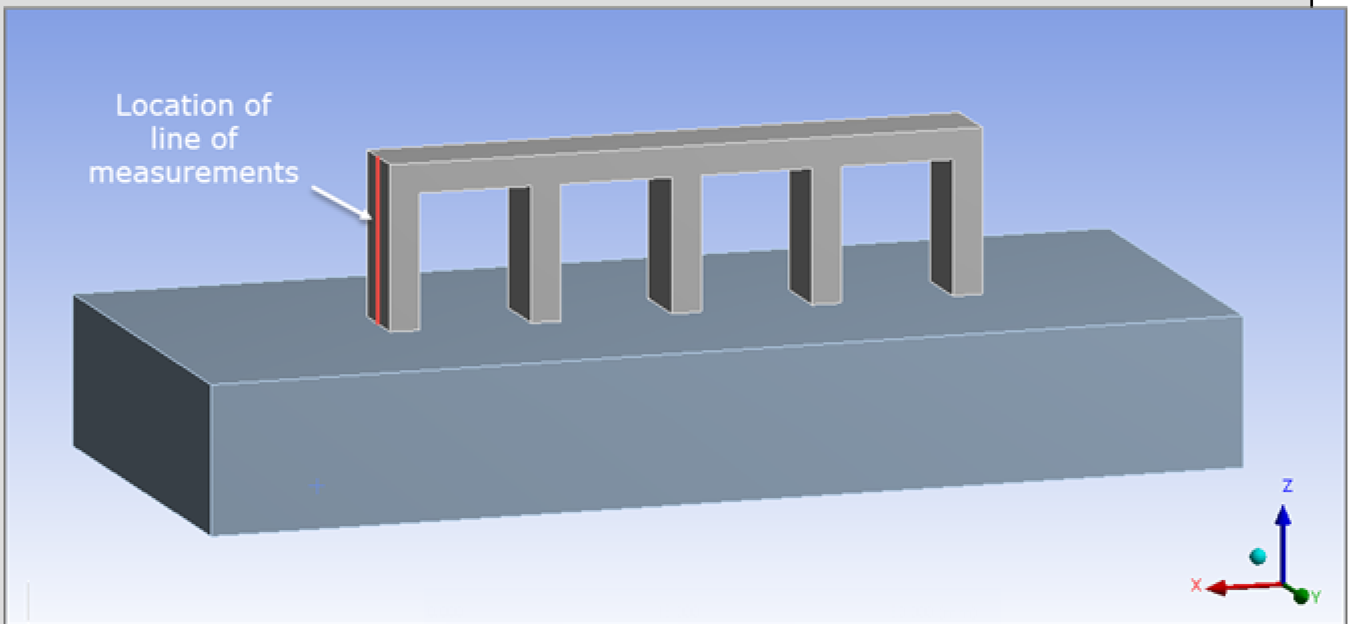
Note:

To ensure a quick, dynamic calculation of results, Calculate Time History is set to No by default for this result item. Set **Calculate Time History** to **Yes** and then right-click **Evaluate All Results** to see the time history results, such as the layer-by-layer animated build. For large models, the calculation may take some time.



Scope a Line of Nodes to Get Results at a Specific Location

Suppose you want to obtain X-direction deformation along a line on a face of your part, such as shown below, at the end of the cooldown step.

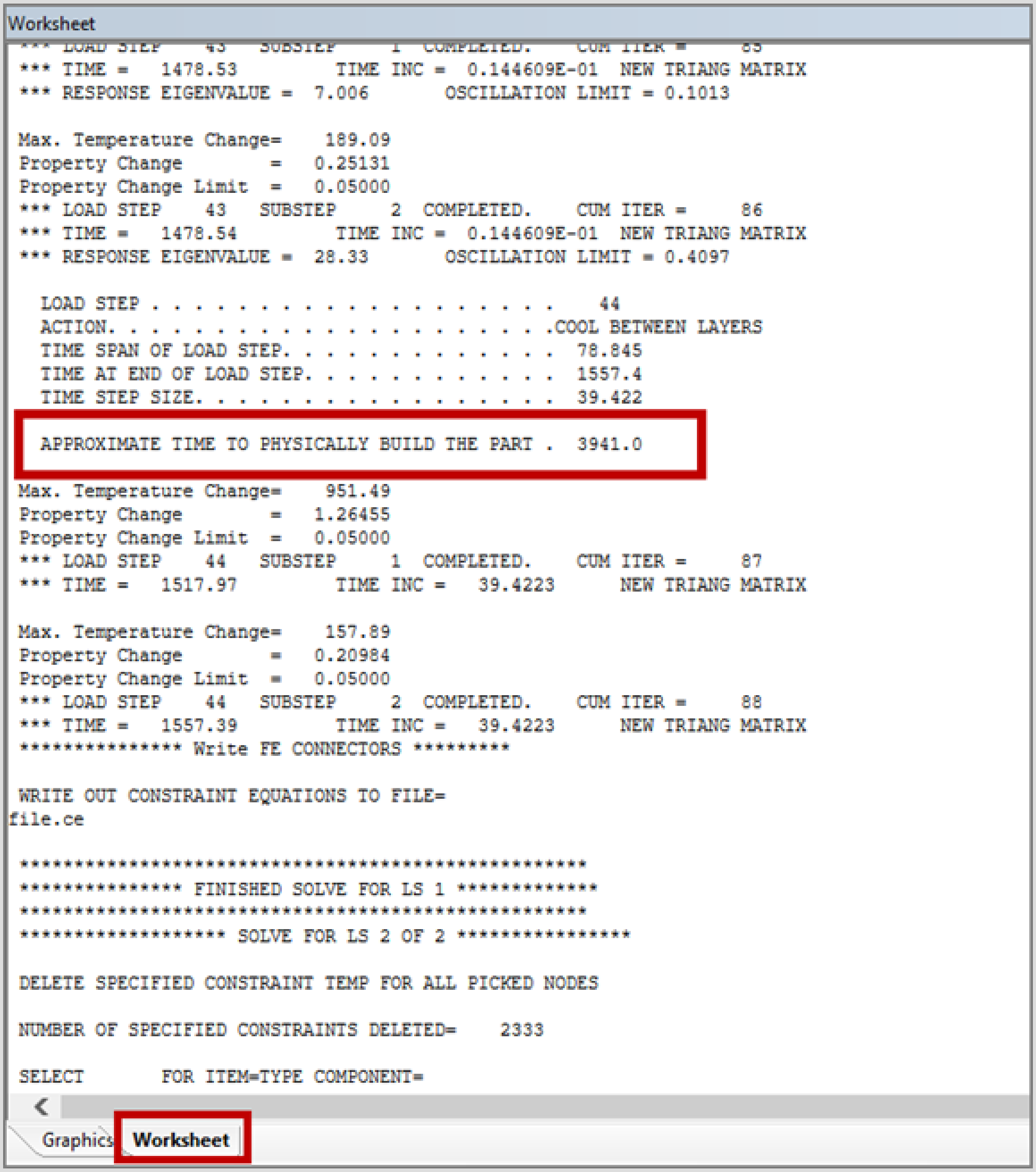


There are several ways to perform this function but we will demonstrate it using a Worksheet. To obtain distortion results along a line on the surface of your part:

1. Create a named selection: Highlight the **Model** object, right-click and select **Insert** > **Named Selection**. Right-click the newly created **Selection** object (under Named Selections) and **Rename** to something meaningful, for example, Line of Nodes.

Get an Estimate of the Actual 3D Printing Time

View the solver output to get an estimate of the real machine build time. It is approximately the transient thermal build step simulation time multiplied by R^(1/3), where R is the number of deposit layers in one element layer. This data is available after the thermal solution. Click the **Solution Information** object and then click the **Worksheet** tab under the Geometry window. In the Details view, be sure that **Solver Output** is selected as Solution Output.



Chapter 6: Advanced Topics

This chapter describes additional topics that you may want to consider for AM Process Simulations, including the following:

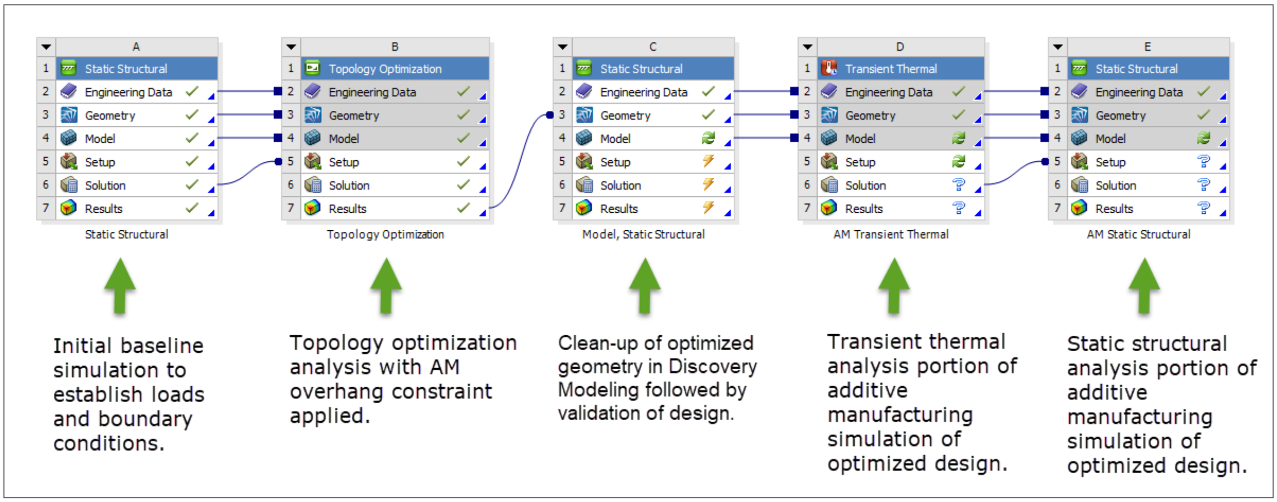
6.1. Using Topology Optimization for Additive Manufacturing

Topology optimization is an exciting technology that allows designers to optimize material layout within a given design space, and for a given set of loads, boundary conditions and constraints, with a goal of maximizing the performance of a product. The optimal shape of a part is often organic and counterintuitive, and difficult or impossible to manufacture using traditional methods. By its very nature, additive manufacturing opens up whole new possibilities for the real-world production of these optimized parts. Along with the potential gains comes challenges unique to the manufacturing process, however. We've seen that the use of supports for overhanging features during the printing process is necessary but costly. If we could optimize our product designs while minimizing the requirement for supports at the same time, we will be taking advantage of the best of both technologies.

The **AM Overhang Constraint** available in Ansys Mechanical's topology optimization tools allows us to do just that. The goal of the overhang constraint is to create a self-supporting structure so that it may be printed without adding supports. We will examine the overall workflow of using topology optimization combined with AM Process Simulation as well as the specific usage of the overhang constraint in this section. For a general discussion of how topology optimization is implemented in Ansys Mechanical, see Structural Optimization Overview.

Topology Optimization and AM Process Simulation Workflow

The general workflow in a linked topology optimization and AM simulation is shown in the following figure.



Step A – Run an initial static structural analysis to establish loads and boundary conditions and baseline results in Mechanical.

Step B – Run the topology optimization analysis with the AM Overhang Constraint in Mechanical.

- Highlight **Topology Optimization**, right-click and select **Insert** > **AM Overhang Constraint**. In Details, change **Build Direction** and **Overhang Angle** to your desired values.

  Parts designed using the AM Overhang Constraint are constrained more and results will include more material than those optimized without the constraint. To allow more flexibility in solving this highly nonlinear problem, we recommend you specify a *range* for the response constraint rather than just a constant value. For example, if you want to reduce the mass in your part by 70%, we recommend you allow the program to use a Percent to Retain range between 25 and 30% for maximum flexibility in the algorithm. Defining a range for response constraint in combination with overhang constraint will frequently require fewer iterations.

- Under Topology Optimization, select **Response Constraint**. In Details, under Definition, choose **Mass** or **Volume** for **Response**. Change **Define By** to **Range** and enter a range of values for **Percent to Retain** Min and Max.

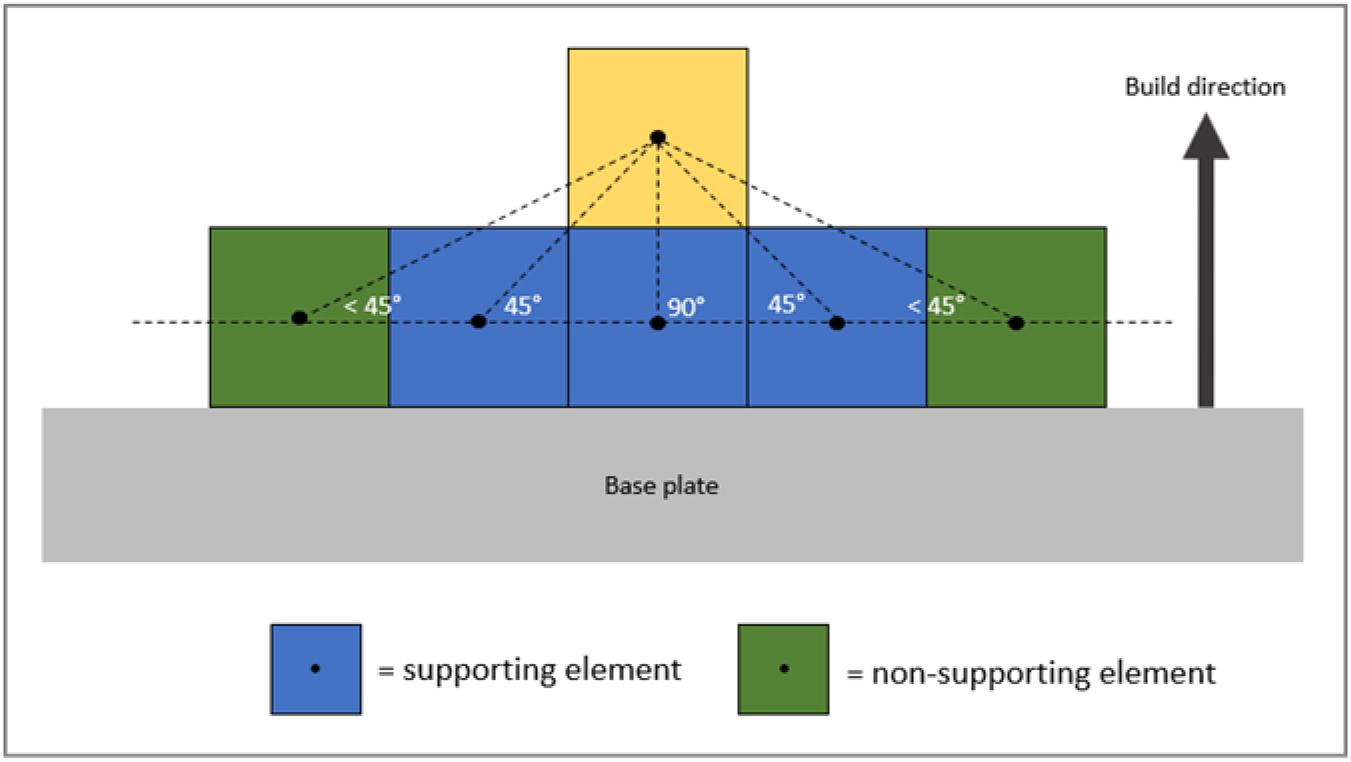
Step C – Clean up the optimized geometry in Discovery Modeling and then validate the design in Mechanical. Search for the Additive Manufacturing section in the Discovery documentation. Also, see Performing Design Validation in the *Structural Optimization in Mechanical*.

Steps D and E – Run the additive manufacturing simulation on the optimized design in Mechanical.

AM Overhang Constraint Methodology

The AM Overhang Constraint is defined by a printing direction and an overhang angle. The printing direction can be one of the global coordinate system axes (either the positive or negative direction).

The overhang angle restricts the state of optimized elements. (Optimized elements are those that are "kept" in the final design and that contribute to the system's overall stiffness matrix; that is, are filled with material.) An element will be kept only if there is a supporting element in the layer below (or above depending on the printing direction) that is also filled with material. An element is called a supporting element of another element if the angle between the line defined by their centroids and the base plate is greater than, or equal to, the overhang angle. The following figure demonstrates the restriction with an overhang angle of 45° in 2D. The printing direction is from the bottom to the top. In order to fill the yellow element with material, at least one of the blue elements has to be filled, too. Which of the blue elements will be kept depends on the state of the surrounding elements as well as the load path. In the final design all remaining optimized elements have supporting elements.



Base Plate

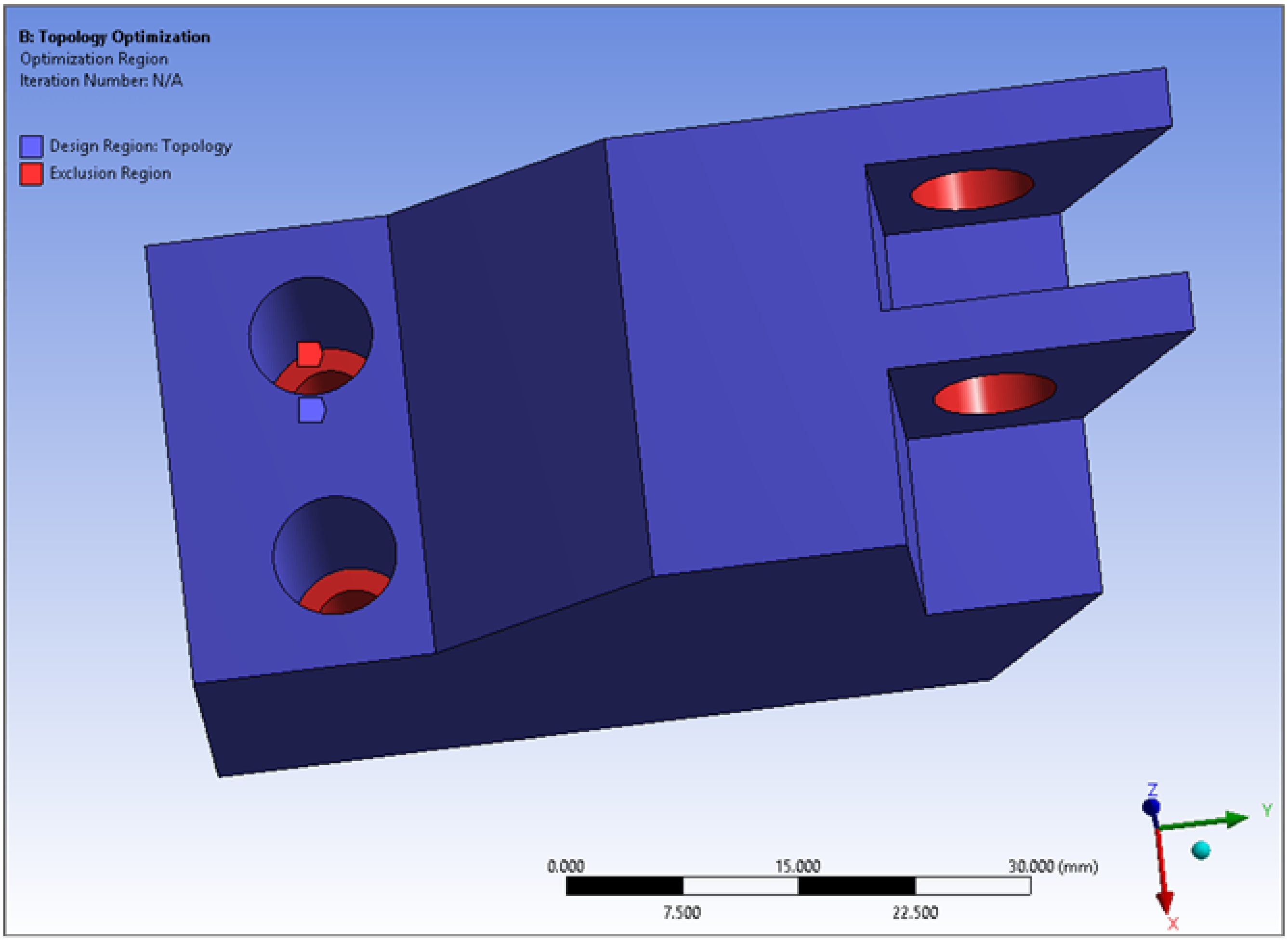
The surface of the base plate is a plane perpendicular to the build direction. The base plate touches the design region from below for positive printing direction (or from above for negative ones). For a design to be printable, it requires a connection to the base plate. The results will depend on the size of the area of contact. Results of the optimization will improve with increased contact area.

Excluded Elements

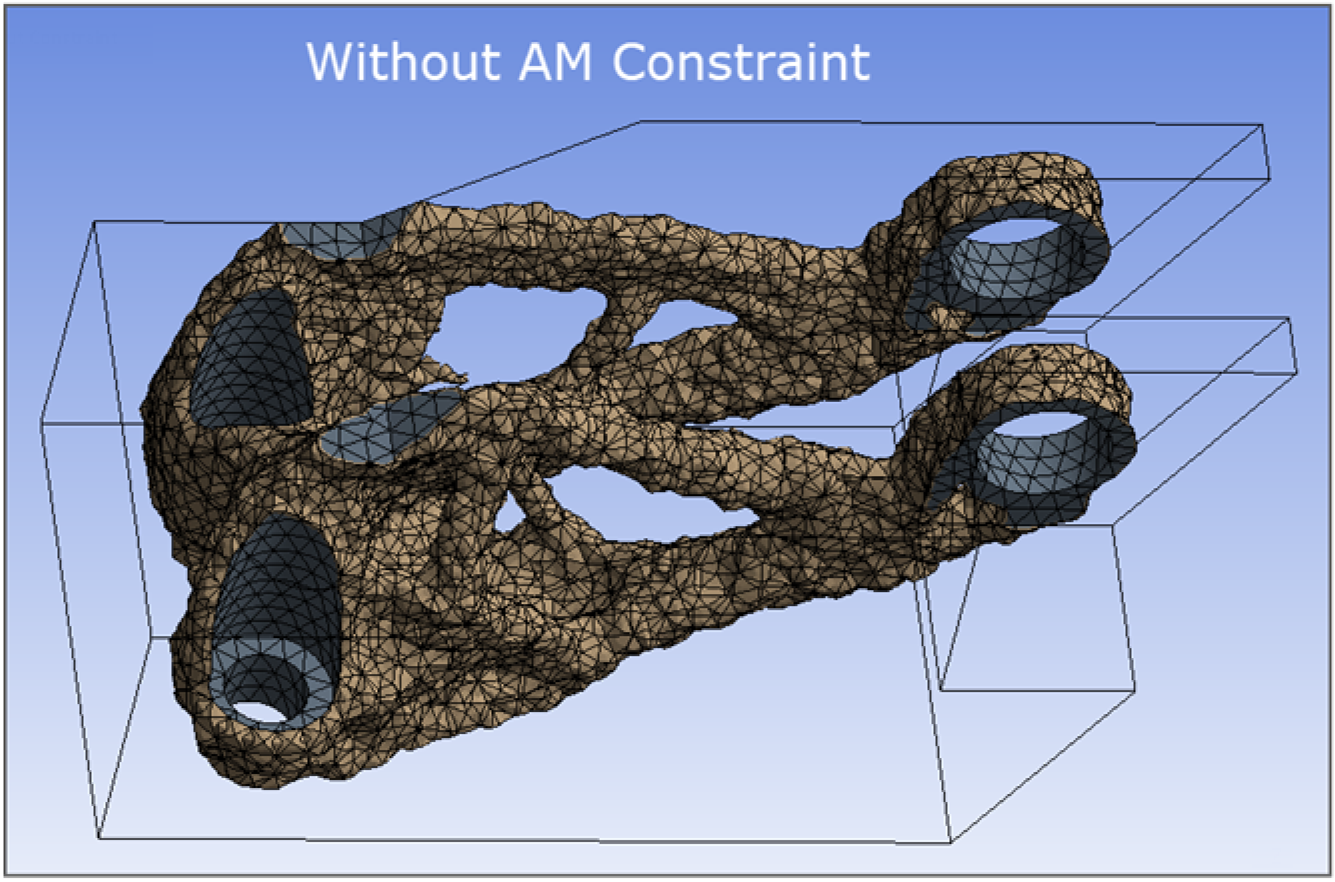
Excluded elements in the design region can lead to designs that do not satisfy the overhang constraint. It cannot be guaranteed that these excluded elements are supported with respect to the overhang angle. Excluded elements are always considered to provide support to optimized elements.

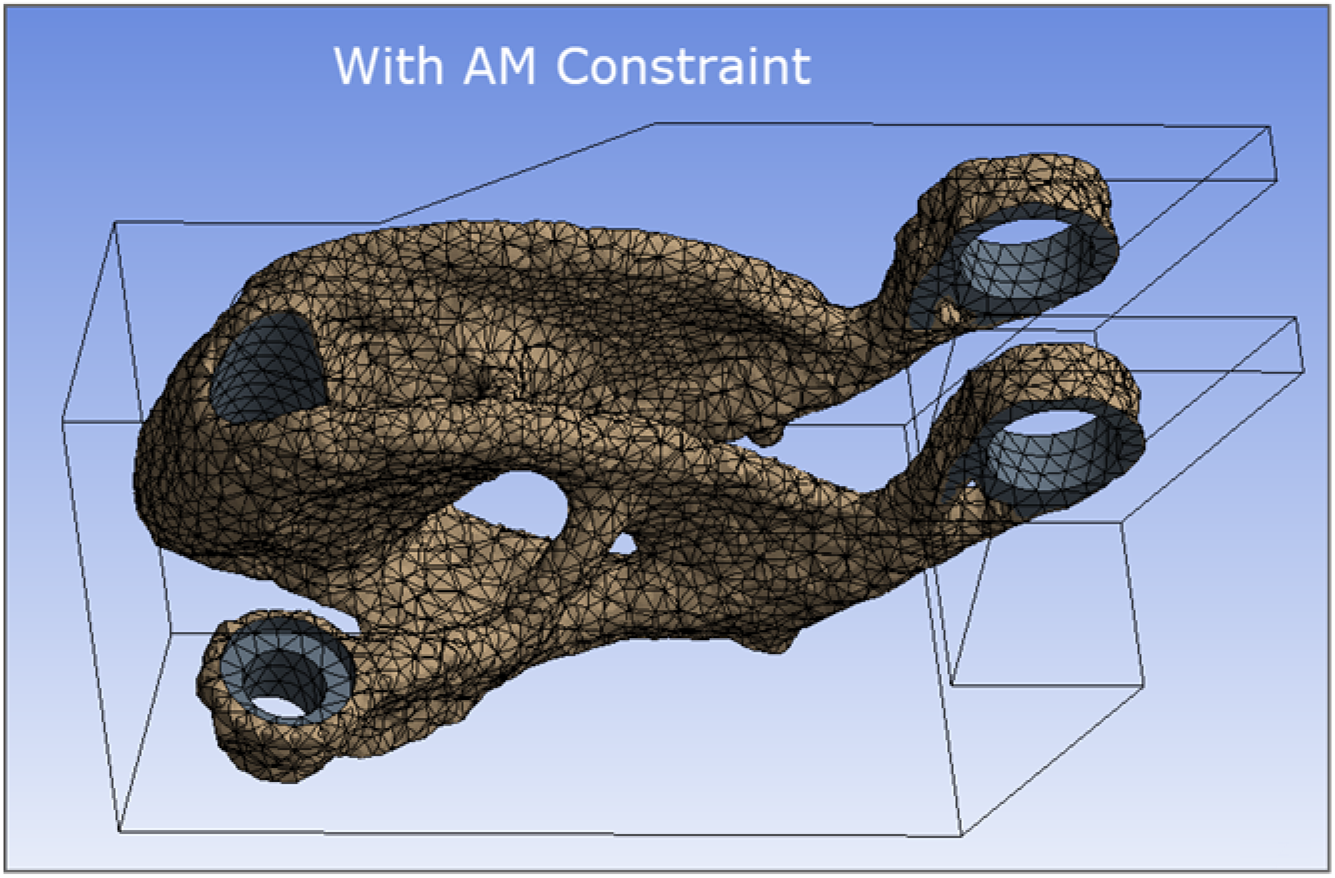
Example of AM Overhang Constraint Used in Topology Optimization

As shown in the following figure, a bracket-type component has two counterbored bolt holes at one end and a through hole going through two flanges at the other. The bolt holes are fixed and a remote force is applied to the two holes of the flanges. These regions are excluded from the topology optimization and are shown in red. The remainder of the geometry is our design region (purple). We want to minimize material on the component while still maintaining its ability to structurally support the loads.



We used topology optimization first without, and then with, the AM Overhang Constraint. We set a mass response constraint range of 10-13% material to retain. When using the AM Overhang Constraint, we set an overhang angle of 45°. The print (or build) direction is from 0 along the positive Z axis.


Without AM Constraint



Comparing the two models, you can see there are several surfaces that will require supports in the first image, especially underneath the branched structures. While not entirely eliminating them, the second model minimizes these surfaces. Note also that the through holes will need to be supported during 3D printing. The hole surfaces were not modified, as they were excluded regions.

6.2. Using the Inherent Strain Method

The Inherent Strain method provides an alternative to a coupled thermal-structural additive process simulation available in Mechanical. With the Inherent Strain method, strains are calculated not from material properties and thermal loads but from the use of a calibration factor, or Strain Scaling Factor. The strain is equal to the Strain Scaling Factor multiplied by yield strength and divided by elastic modulus:

$$\varepsilon = SSF * \frac{\sigma_{Yield}}{E}$$

The Strain Scaling Factor (SSF) is an important factor quantifying the variables unique to each build scenario. It must be experimentally determined for each machine and material combination of interest. We recommend you perform the same calibration procedure used for Ansys Additive, as described in the Additive Print and Science Calibration Guide.

The SSF value is a direct multiplier to the predicted strain values. Using a value of 1 (default) will result in strain magnitudes as calculated by the solver. Some material and geometry combinations result in bulging/expansion rather than shrinkage and so a negative SSF is possible. Values between -1 and 1 will reduce displacement and stress while values outside of that range will amplify them. Using a value of 0 will result in no strain and the final displacement will match the input geometry.

You can define calibration factors that are different in each direction (X, Y, and Z) based on the Global coordinate system by choosing Anisotropic Inherent Strain Definition. The properties of materials will differ in different directions. If Inherent Strain Definition = Scan Pattern or Thermal Strain, individual calibration factors, called anisotropic scaling coefficients (ASCs), may be entered based on the local orientation of scan vectors within the part, that is, parallel and perpendicular to scanning direction and in the build direction.

The steps in a typical additive manufacturing simulation are shown in the table below. Considerations unique to using the Inherent Strain method are described.

Inherent Strain Workflow at a Glance	
Simulation Step	**Considerations for Inherent Strain Method**
1. Create analysis system	Select the AM LPBF Inherent Strain custom system in Workbench.
2. Define Engineering Data	Necessary only if you are using your own user-defined material. (Ansys-supplied sample AM materials are automatically available if you used AM LPBF Inherent Strain custom system.)
3. Attach geometry and launch Mechanical	No special considerations.
4. Identify geometry	No special considerations.
5. Assign materials	No special considerations.
6. Apply mesh controls and generate mesh	No special considerations, other than if you will be using AM octree adaptive meshing (p. 163), in which case Cartesian meshing with voxelization option is required. Adaptive meshing is available only with Inherent Strain simulations.
7. Identify and/or generate supports	No special considerations.
8. Define connections	No special considerations.
9. Define AM process steps	Note there is no cooldown step.
10. Define build settings	Set Inherent Strain = Yes (Automatic if you used AM LPBF Inherent Strain custom system.) Set Inherent Strain Definition to isotropic, anisotropic, scan pattern, or thermal strain. The thermal strain definition uses the machine learning (ML) method. Define process parameters. Required inputs depend on the selected strain definition. Enter strain scaling factors, usually determined by a calibration procedure.
11. Establish structural analysis settings	Review and change output controls as needed. Most items are not stored during solution to reduce result file size. Consider suppressing the calculation of stresses and strains if you are interested only in distortion.
12. Apply structural boundary conditions	Apply a fixed condition to the underside of the base assuming the base is rigid with no distortion.

Inherent Strain Workflow at a Glance	
Simulation Step	**Considerations for Inherent Strain Method**
13. Solve the static structural analysis	No special considerations.
14. Review results	No special considerations.

6.3. Using JAHM Temperature-Dependent Material Data in AM Simulation

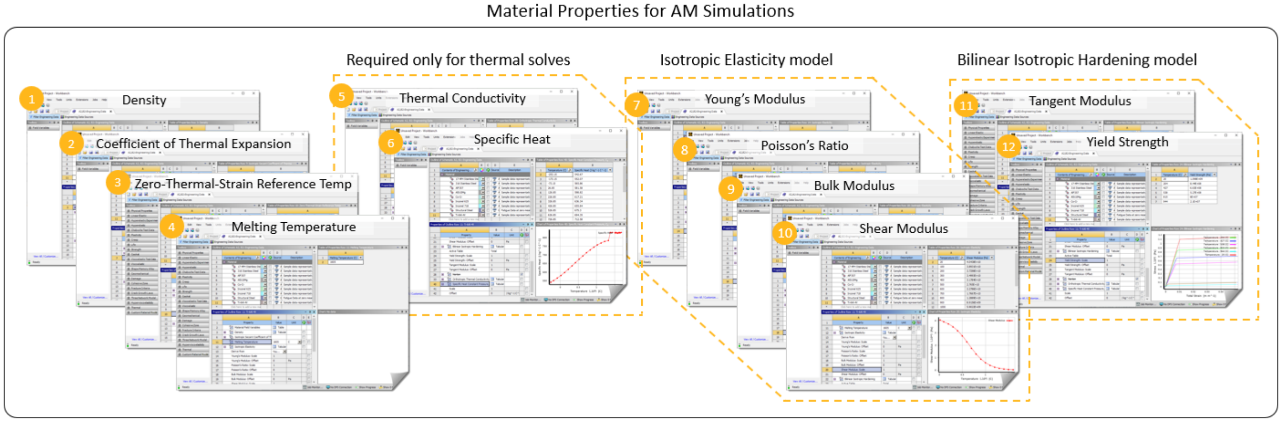
As an alternative to using Mechanical's predefined sample materials for additive manufacturing, you may choose to use a material from the JAHM Curve Data database in Granta MI's material library. JAHM Curve Data provides temperature-dependent curve data for thousands of materials across 27,000+ records. It includes mechanical, thermal, physical, electrical, fatigue, creep, stress-strain, and magnetic properties for materials in a range of physical states (gas, liquid, solid) and covers temperatures ranging from -270°C to over 5,000°C.

When choosing a material from JAHM Curve Data, keep in mind that AM simulations in Mechanical are currently limited to metal materials. Furthermore, the metals included in the JAHM database are not additive specific. That is, the test specimens were not additively manufactured. However, there may be times when you want to use a material from this source. For this reason, we have provided the following procedure and guidance for using JAHM Temperature-Dependent Material Data in an AM simulation.

6.3.1. Review of Required Material Properties

First let's review the material properties required to perform AM simulations in Mechanical. The following figure shows the required material properties, as depicted by screen captures from Workbench's Engineering Data. The selected material happens to be the Ansys Ti-6Al-4V sample material. The 12 properties are: Density; Coefficient of Thermal Expansion; Zero-Thermal-Strain Reference Temperature; Melting Temperature; Thermal Conductivity; Specific Heat; Young's Modulus, Poisson's Ratio, Bulk Modulus, and Shear Modulus for the Isotropic Elasticity model; and Yield Strength and Tangent Modulus for the Bilinear Isotropic Hardening model. Note that Isotropic Thermal Conductivity and Specific Heat are required only when performing simulations with thermal solves, such as AM LPBF Thermal-Structural.

Material Properties for AM Simulations



When using JAHM materials, most of the required data is available on a single material record reflecting "general properties" for a given material, which can be exported as a *material card* from Granta MI and imported directly into Engineering Data. However, a few pieces of data are usually missing from the general properties record and you must provide the missing data. In many cases, the Bilinear Isotropic Hardening model data, from true stress-strain (tension) tensile testing, is not included in a material record that includes all the other required mechanical and thermal properties. This is because true stress-strain is very dependent on the specific manufacturing and heat treatment processes, as well as on tensile test conditions, such as strain rate, hold time, and so on. To handle this, search for a record that contains true stress-strain (tension) data corresponding to your preferred scenario for your chosen material. From that data, you will derive the Yield Strength and Tangent Modulus for the Bilinear Isotropic Hardening model. You may need to refer to other sources to get the single data points of Zero-Thermal-Strain Reference Temperature and/or Melting Temperature.

The following table summarizes the required properties for AM simulation when using a JAHM temperature dependent material.

Material Properties Required for AM Simulation			
Property	**Form**	**Components**	**Source**
Density	Tabular	Temperature, Density	Direct import from Granta MI
Isotropic Coefficient of Thermal Expansion (CTE)	Tabular	Temperature, CTE	Direct import from Granta MI
Zero-Thermal-Strain Reference Temperature	Single data point	Temperature	Other source
Melting Temperature	Single data point	Temperature	Other source
Isotropic Elasticity model:	Tabular	Temperature, Young's Modulus, Poisson's Ratio, Bulk Modulus, Shear Modulus	
Young's Modulus			Direct import from Granta MI
Poisson's Ratio			Direct import from Granta MI
Bulk Modulus			Direct import from Granta MI
Shear Modulus			Direct import from Granta MI
Bilinear Isotropic Hardening model:	Tabular	Temperature, Yield Strength, Tangent Modulus	
Yield Strength			Direct import from Granta MI
Tangent Modulus			*Derived* from true stress-strain curves
Thermal Conductivity[a]	Tabular	Temperature, Thermal conductivity X, Y, Z	Direct import from Granta MI

Material Properties Required for AM Simulation			
Property	**Form**	**Components**	**Source**
Specific Heat[a]	Tabular	Temperature, Specific heat	Direct import from Granta MI

[a] Required only for Thermal-Structural simulations

6.3.2. Workflow Using JAHM Material

The steps for using JAHM temperature-dependent material data in an additive manufacturing simulation are:

1. Obtain JAHM Material Data from Granta MI (p. 142)
   - Search for material record representing general properties (p. 142)
   - Export material card (p. 146)
   - Search for material record for true stress-strain data (p. 147)
   - Save the true stress-strain curve data (p. 151)
   - Use a spreadsheet to derive data for Bilinear Isotropic Hardening model (p. 152)
2. Enter JAHM Material into Engineering Data (p. 155)
   - Import general properties material card (p. 155)
   - Add missing properties (p. 157)
   - Rename and save the new material in Engineering Data (p. 160)
3. Add JAHM Material to a Project and Assign it to a Geometry in a Simulation (p. 162)

6.3.2.1. Obtain JAHM Material Data from Granta MI

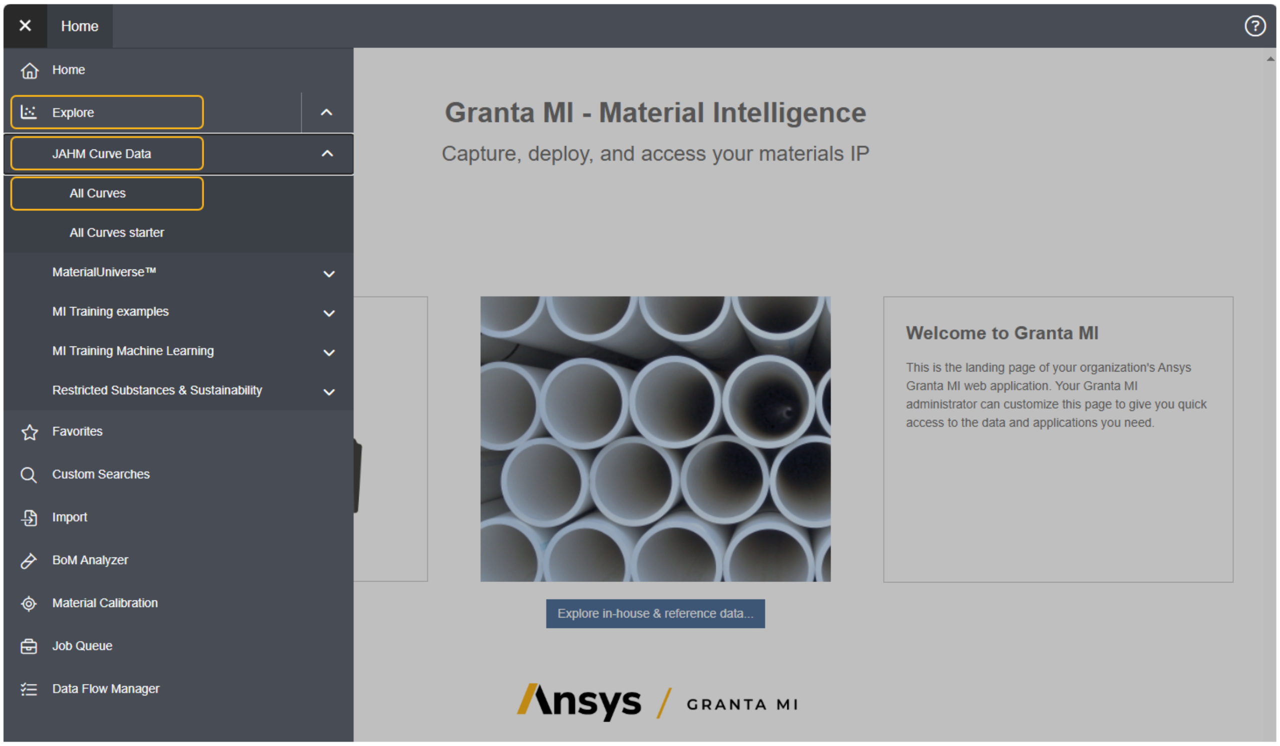
We'll use Ti-6Al-4V as an example material throughout this documentation and we'll use Granta MI Explore to search for the desired JAHM material in Granta MI. We will need more than one material record to get all the required properties.

Search for material record representing general properties

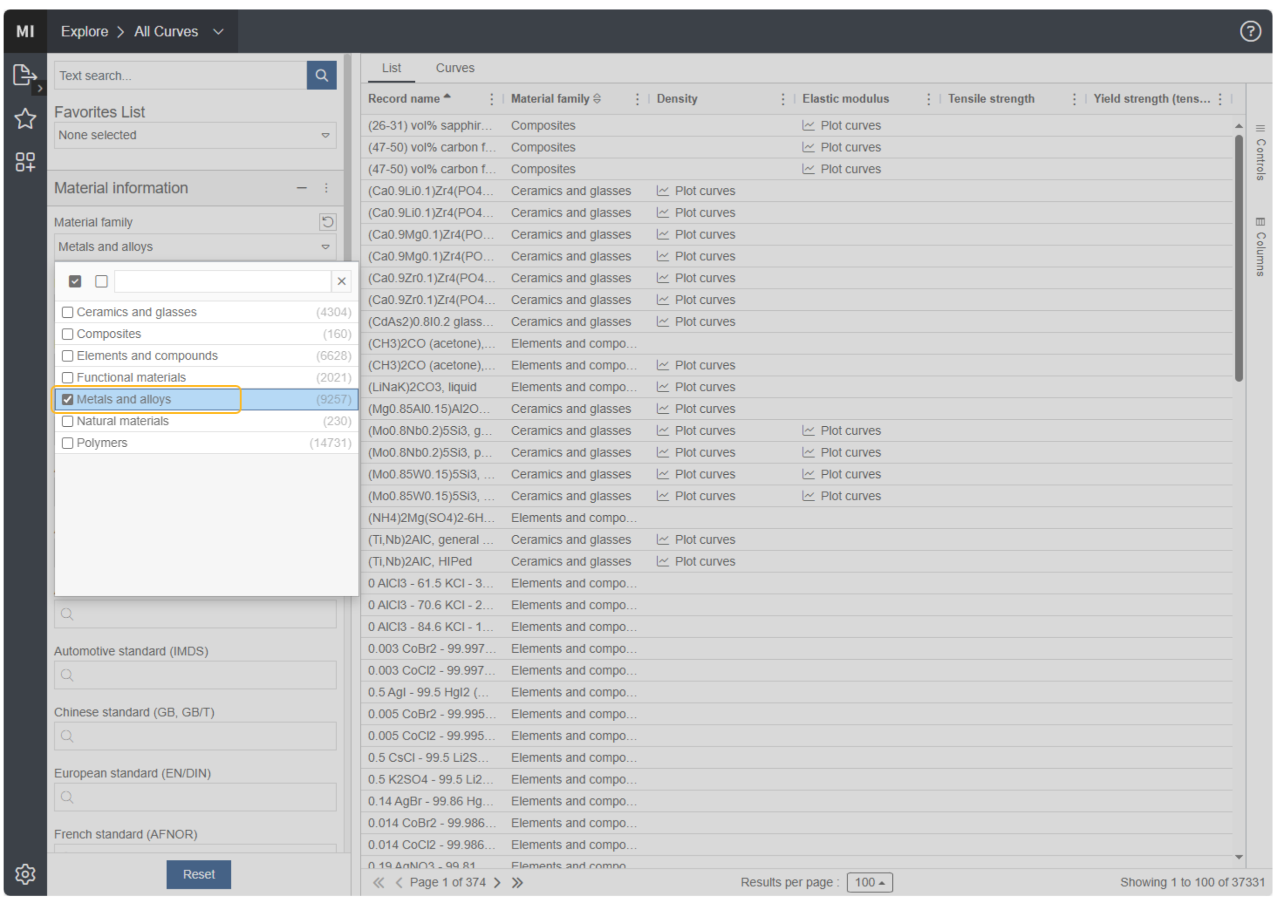
Important:

Either a Granta Selector or a Granta MI license is required to get access to JAHM datasets. If you do not see the features used in this example on your system, check with your Granta MI Administrator, as many features are configurable, such as the Export to Excel feature.

Open Granta MI and, from the **Explore** menu drop-down, choose **JAHM Curve Data** and then **JAHM All Curves**.



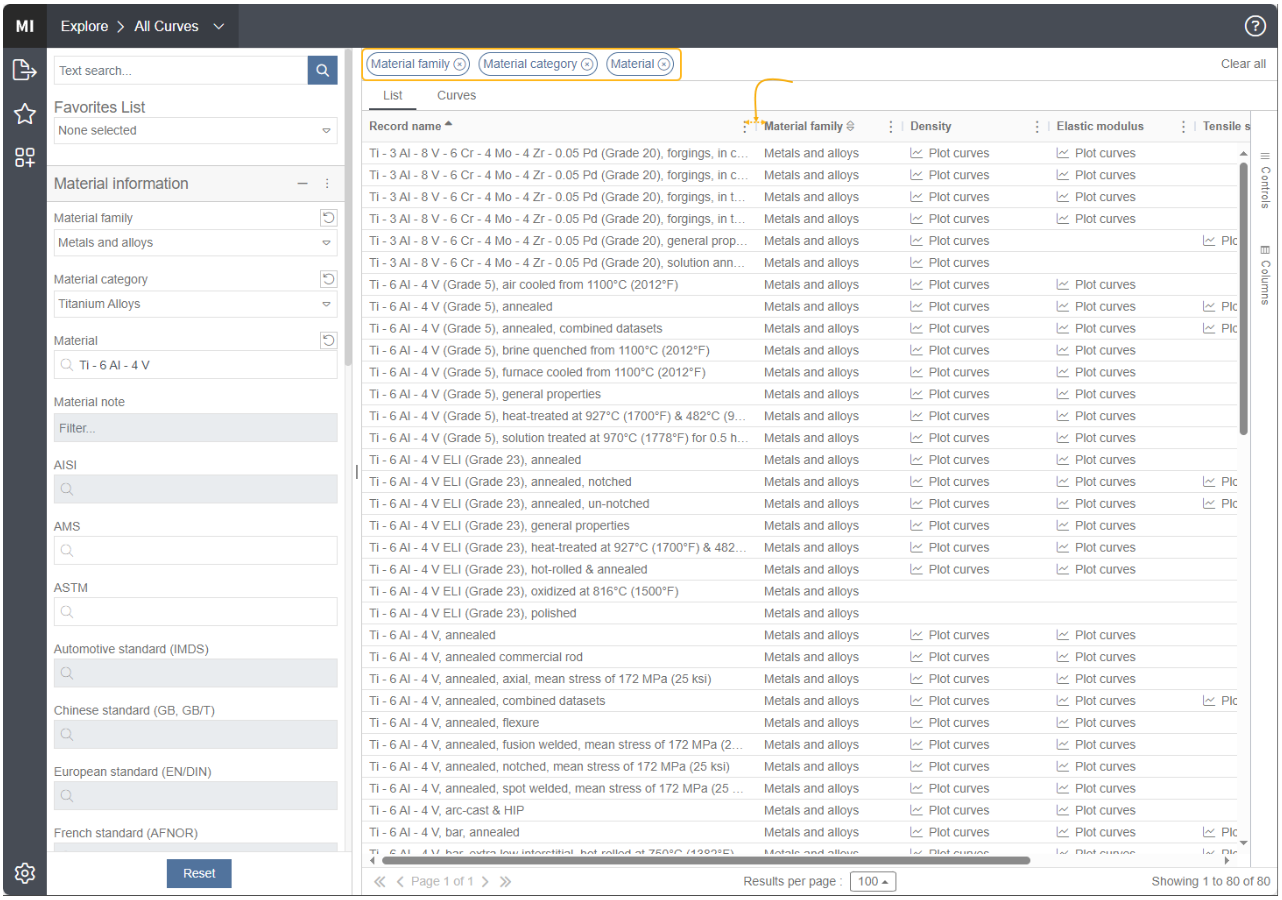
On the left side is a search panel where you enter filter criteria. Under Material information, Material family, choose **Metals and alloys**. After selecting the check box, hit **Escape** or click anywhere in the user interface outside of the check box window.



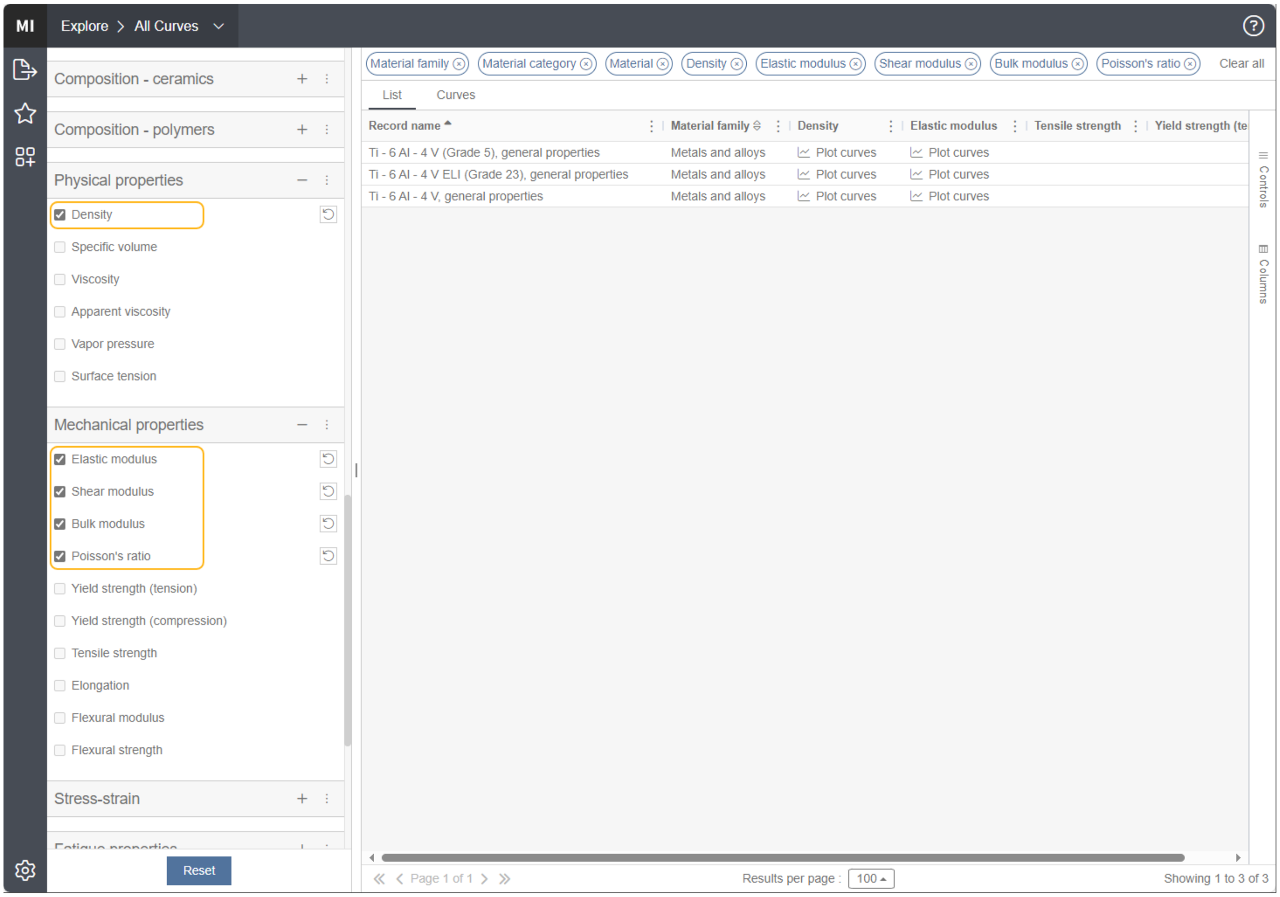
Under Material category, choose **Titanium Alloys**, followed by **Escape**.

Under Material, choose **Ti - 6Al - 4V**.

As you continue to narrow the scope of materials you will see the filter criteria listed at the top of the page. Adjust the column widths as needed to see the information.

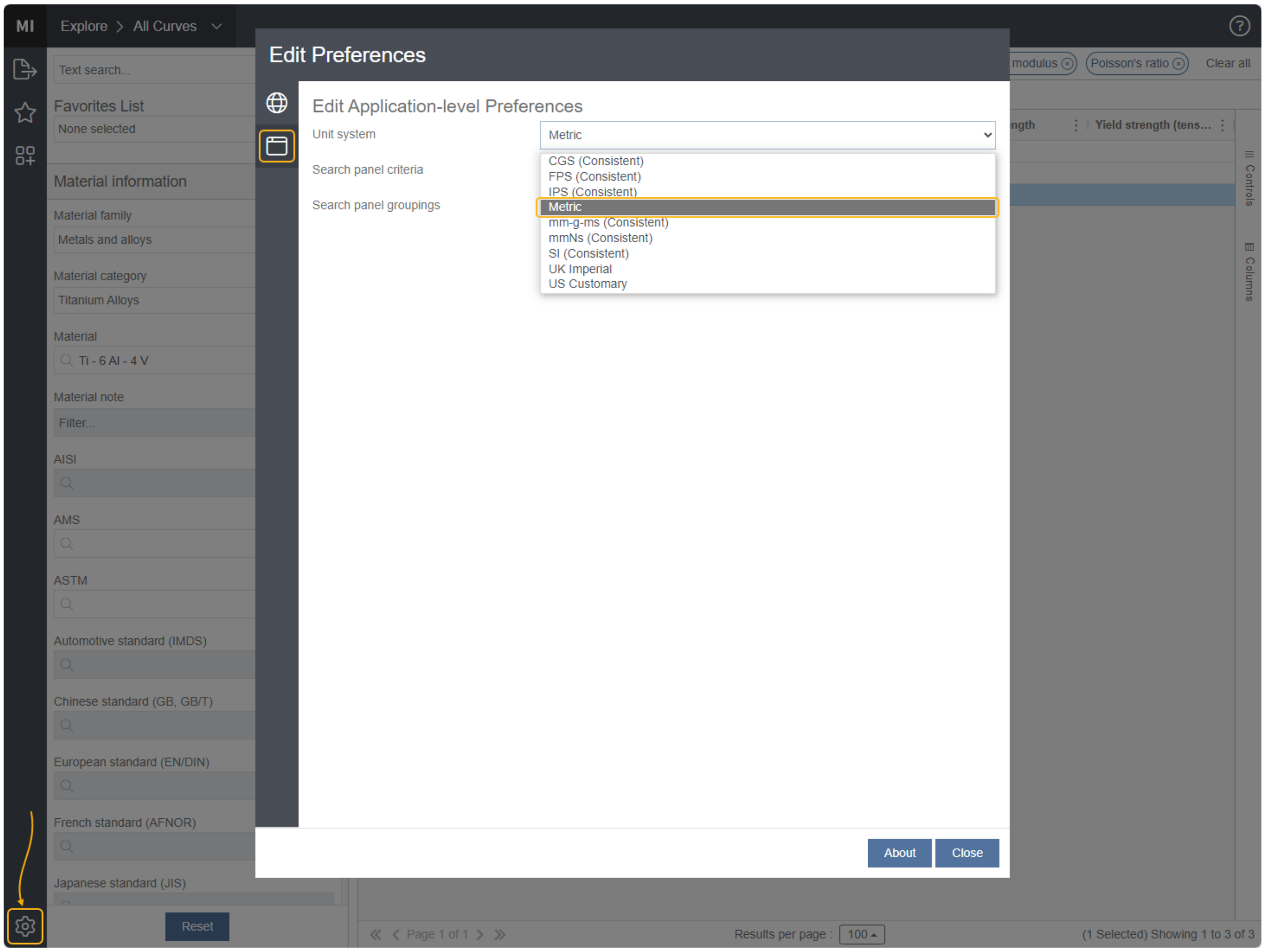


Now we need to narrow the list down to those Ti - 6Al - 4V materials that have density and the properties for the isotropic elasticity model. In the search panel, scroll down to the Physical properties check boxes and choose **Density**. Under Mechanical properties, choose **Elastic modulus**, **Shear modulus**, **Bulk modulus**, and **Poisson's ratio**. If you also try to select Yield strength (tension) and/or, under Stress-Strain, True stress-strain (tension), you will see the message: *"No matches for this criteria in the current search results."* There are no material records that contain all the data. This is because true stress-strain is very dependent on the specific manufacturing and heat treatment processes, as well as on test conditions, such as strain rate, hold time, and so on. To handle this, we will find a second material record later. For now, select only the five properties as shown here.



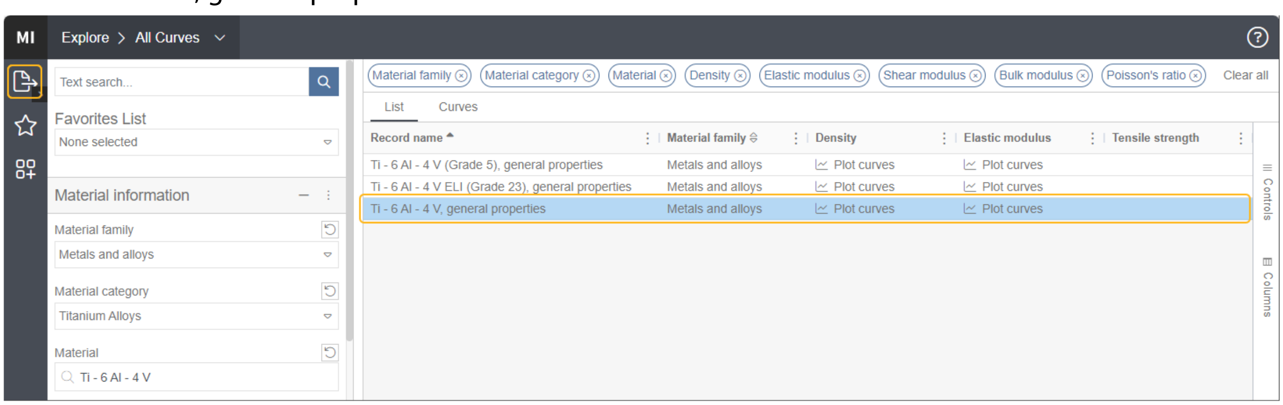
From the search results, we'll choose **Ti - 6Al - 4V, general properties**. You can view the specific properties, plot curves, and so on by double-clicking the record.

You can change the units system under **Preferences**, in the lower, left corner of the interface. Be sure to save the material data in Metric or SI units to be consistent with the default units used in Workbench's Engineering Data for AM simulations in Mechanical.

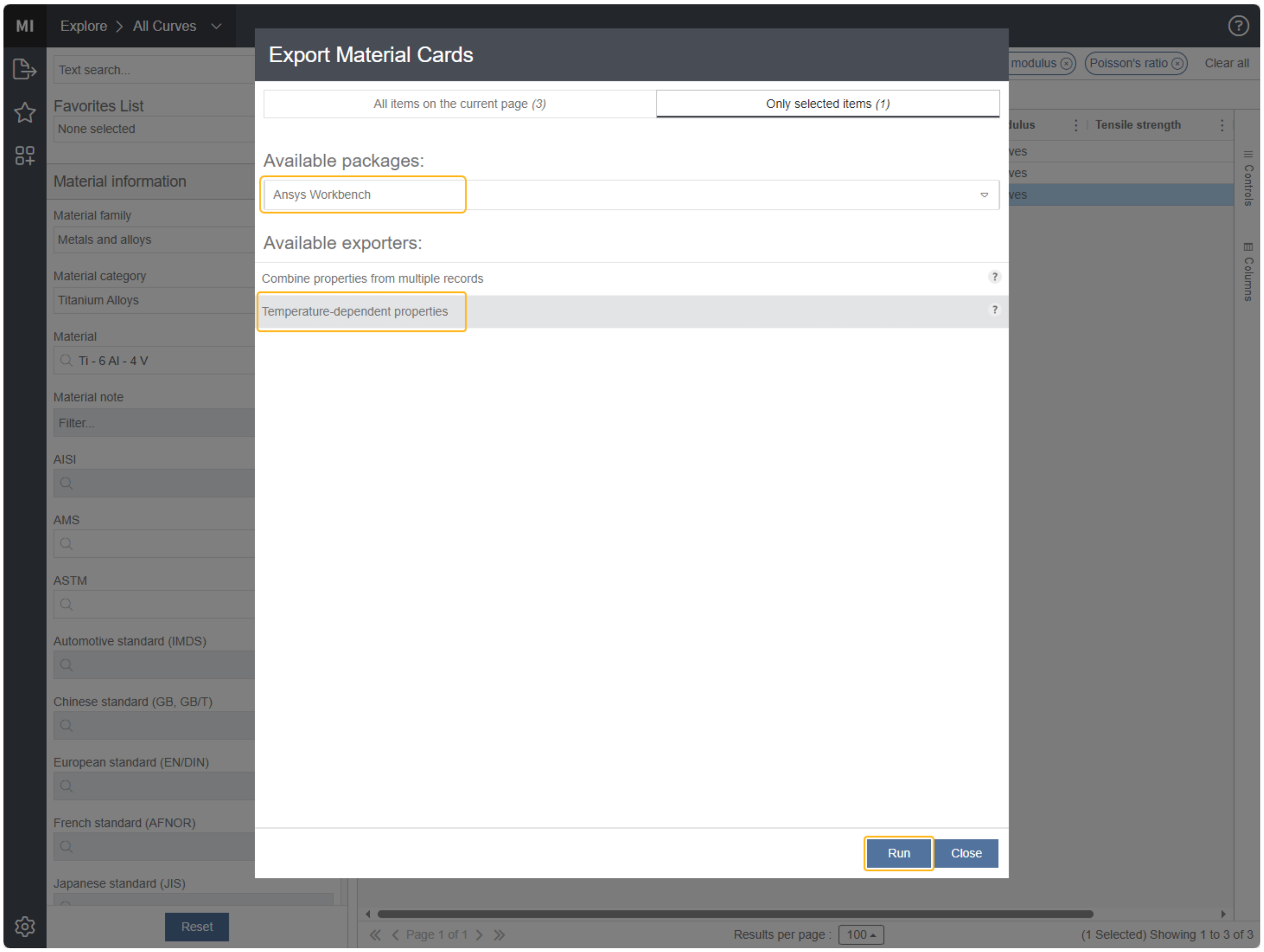


Export material card

A material card is an xml file that contains material data in a format useable by the Engineering Data module. On the left side of the page, click the **Export** icon, **Export Material Cards**, to export the Ti - 6Al - 4V, general properties material card.

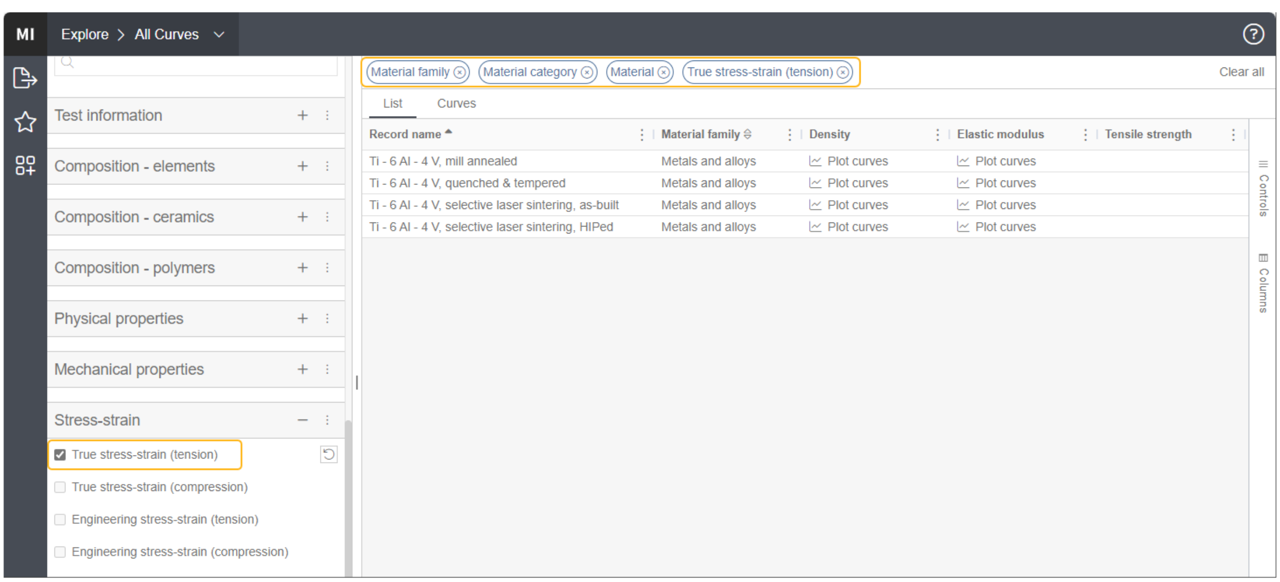


Under Available packages, choose **Ansys Workbench**. Under Available exporters, choose **Temperature-dependent properties**. Then click **Run**. Save the xml file to a location on your computer.

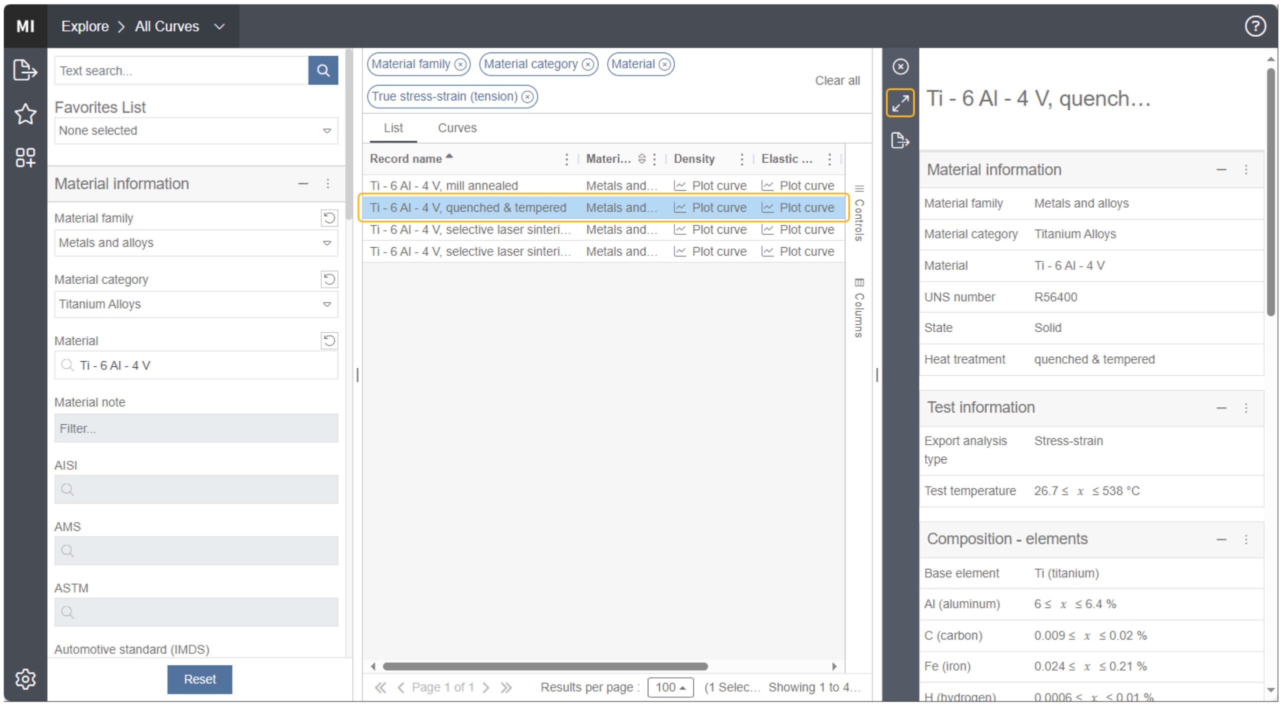


Search for material record for true stress-strain data

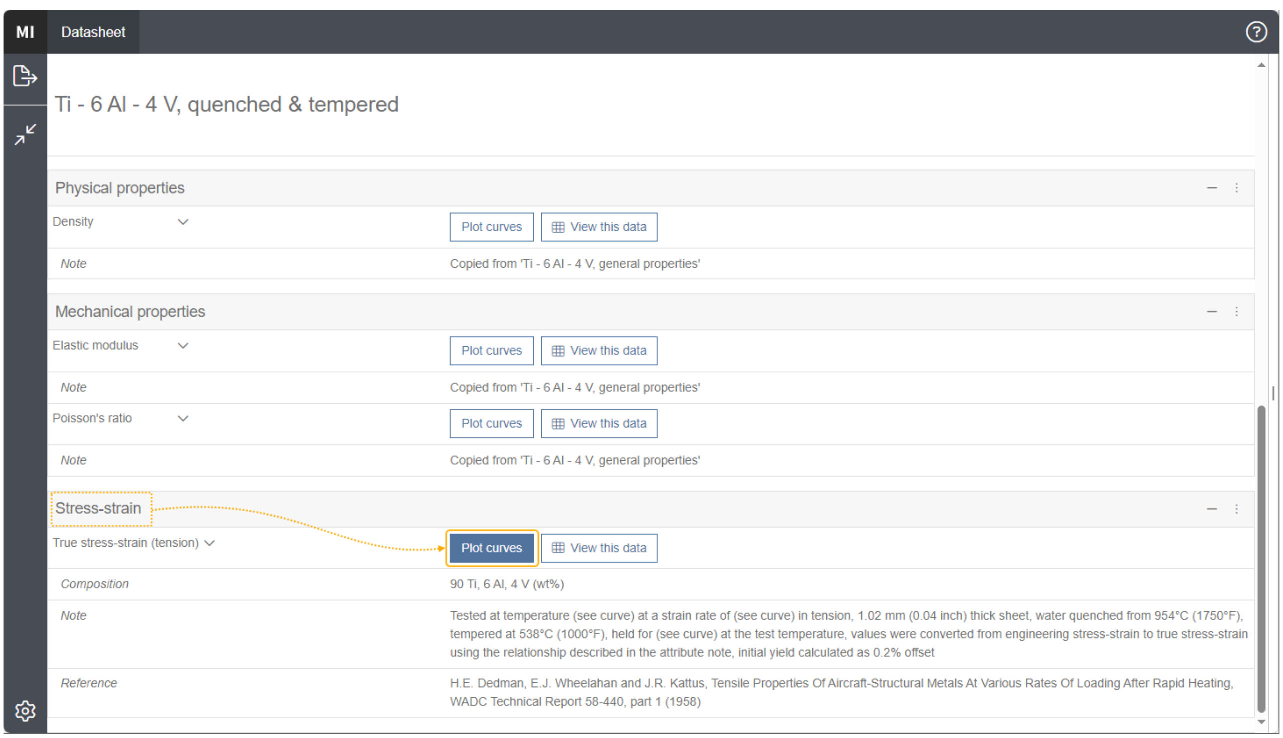
Now we need to obtain the information for the plastic portion of the stress-strain curve. Navigate back to the Explore page and click **Reset** to clear the form. Again search for **Metals and alloys**, **Titanium Alloys**, **Ti - 6Al - 4V**, and this time, scroll down to Stress-Strain and check the box for **True stress-strain (tension)**.

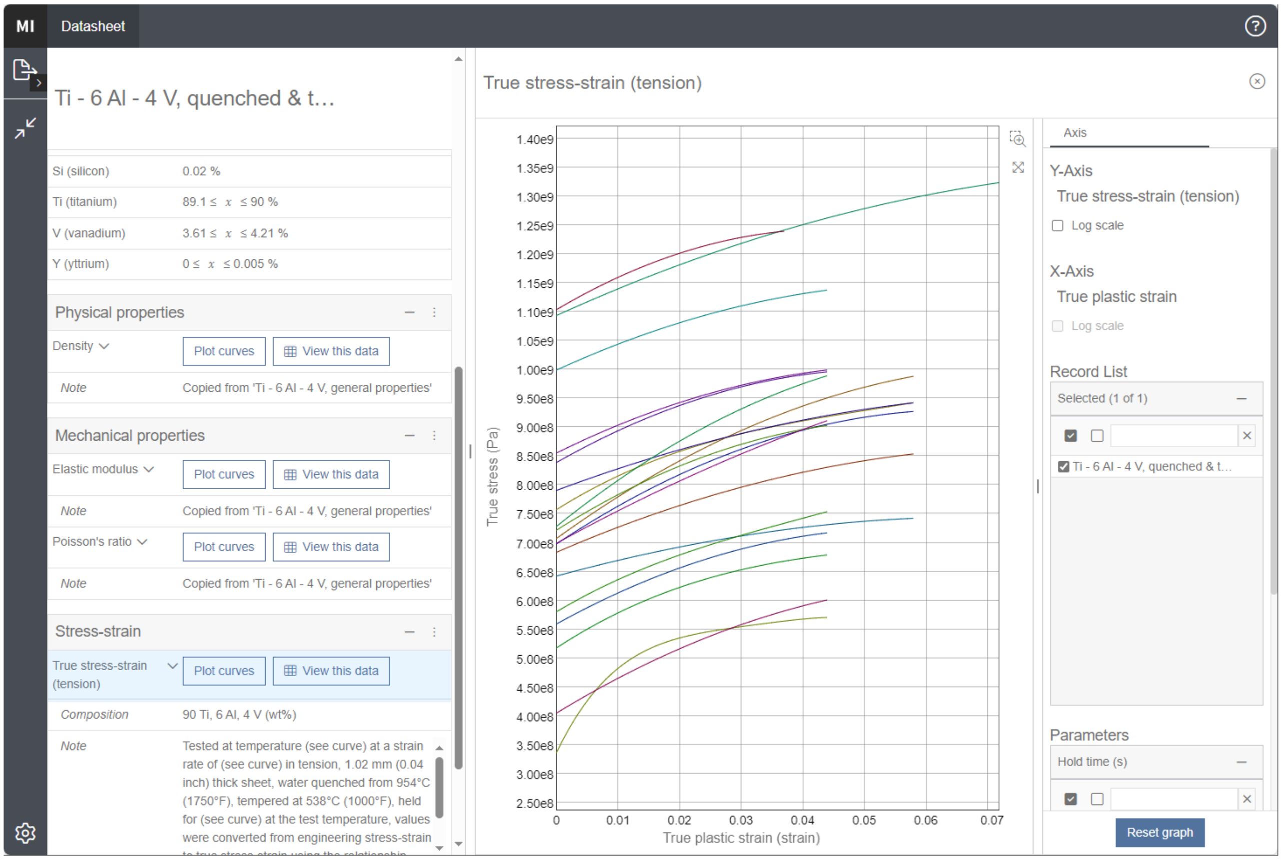


The search results show Ti-6Al-4V material records with various manufacturing processes. Double-click **Ti - 6Al - 4V, quenched and tempered**. In the right side panel that appears, click the arrows icon to **Show full page datasheet**.



Scroll down to the Stress-strain section and click **Plot curves**.



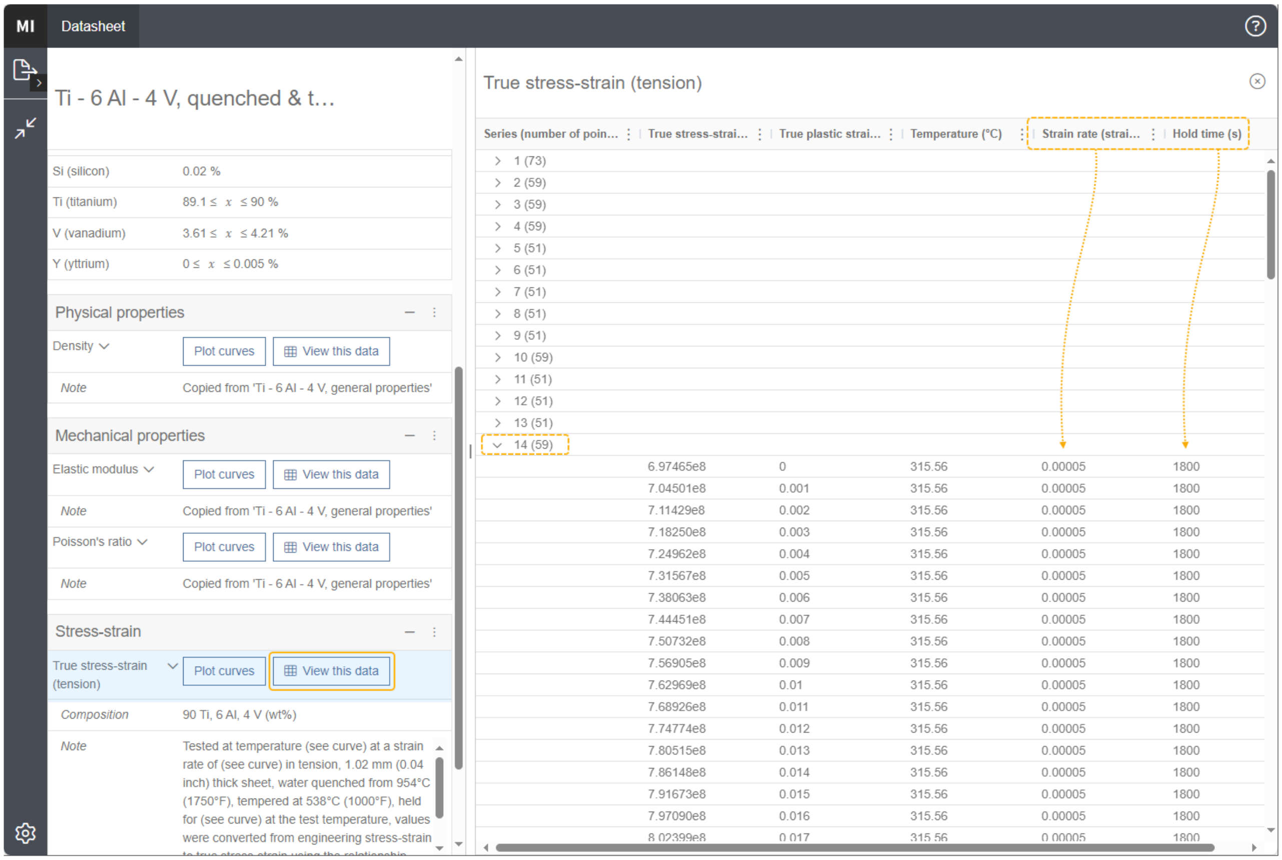


The many curves shown reflect the Ti - 6Al - 4V, quenched and tempered material record with various heat treatment processes and tensile test methods. We will need additional filters to select the conditions we want. We'll choose one that has a fairly slow strain rate and held for a long time. Generally, we would prefer a lower strain rate for a quasi-static tensile testing and a longer hold time for the temperature to homogenize at an elevated testing temperature. You should choose a record that is most appropriate for your situation.

On the right-hand side panel, scroll down to see the parameters of Hold time and Strain rate. Under **Hold time**, clear the first two check boxes and keep the box checked for **1800** seconds, reflecting a half hour hold time. Under **Strain rate**, clear the last two check boxes and keep the check box checked for **0.00005** strain/sec. This leaves three stress-strain curves representing three different temperatures, 204.44°C, 315.56°C, and 537.78°C. Hover over each curve to see the details.

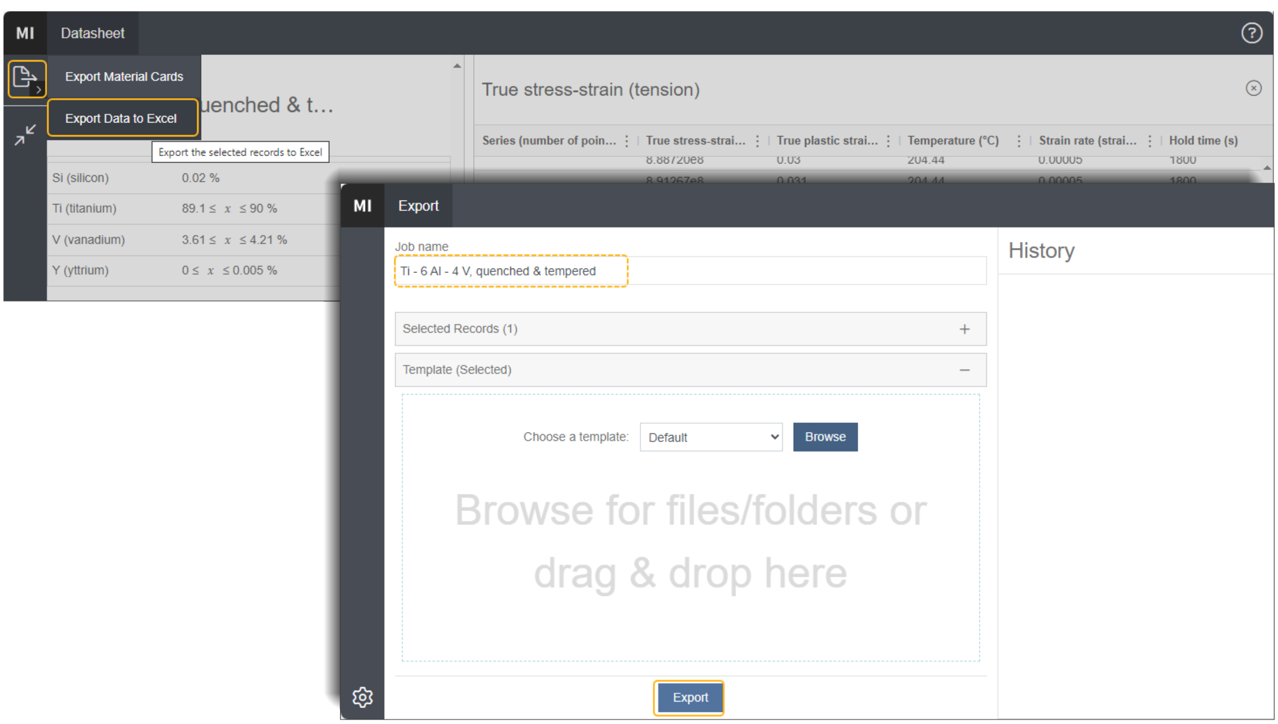
Remember that from the first material card we already have the data for the *elastic* portion of the material's overall stress-strain curve. By looking at the X-axis (True plastic strain) on these curves, we know the graph shows only the *plastic* portion of the material's overall stress-strain curve. For each temperature, the yield point is at the start of the curve, or at a true plastic strain = 0. We will calculate the slope of each curve, which is the tangent modulus.

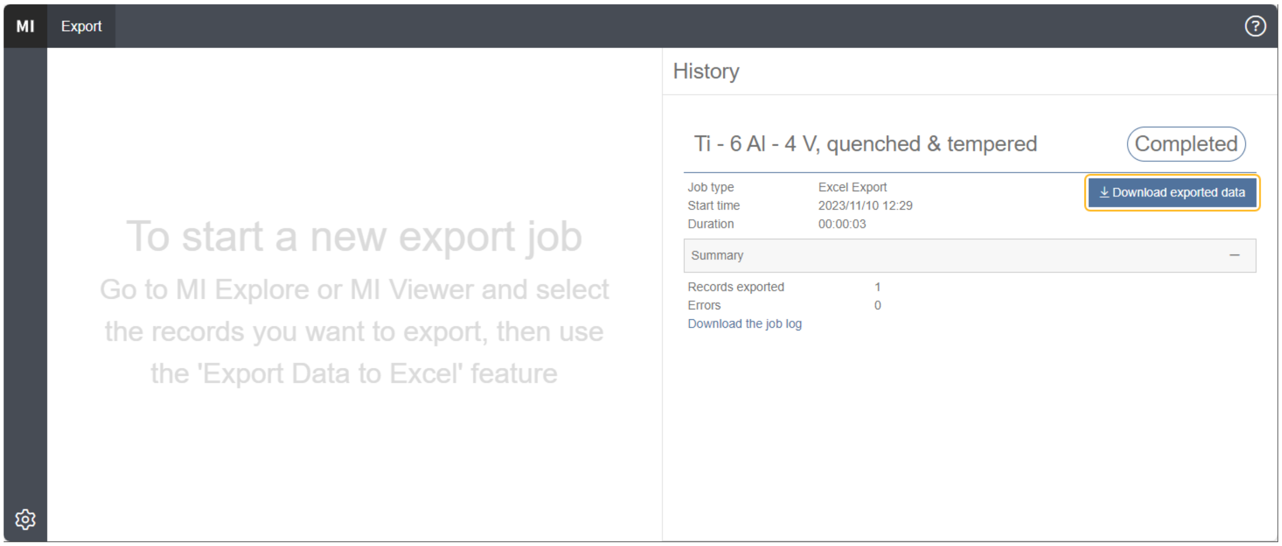
To view the data raw numbers, on the left side panel, under Stress-strain, click **View this data**. Again, this data is from Ti - 6Al - 4V, quenched and tempered with all the processes and test methods. Collapse the data until you find our desired data for 0.00005 strain rate and 1800 hold time. The data for the three temperature curves corresponding to 315.56°C, 537.78°C, and 204.44°C are in Series 14, 15, and 16, respectively. This is the data we must extract.



Save the true stress-strain curve data

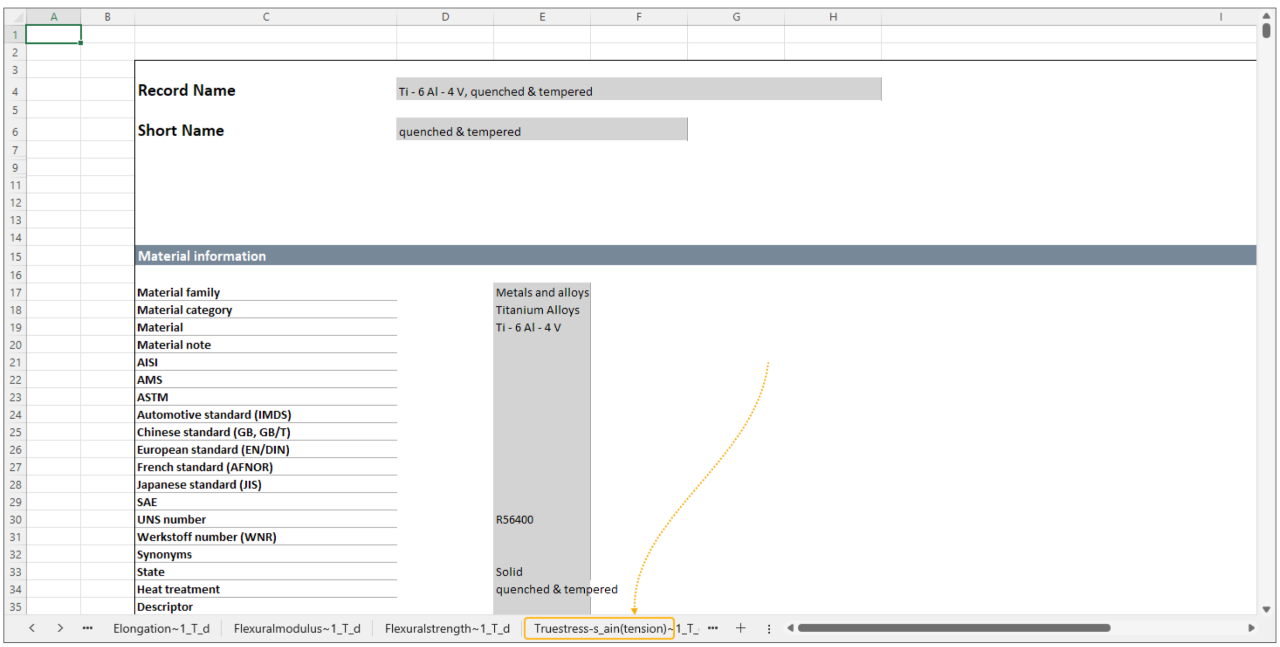
On the left side of the interface, click the **Export** icon and then **Export to Excel** to save the data. On the next page, after the export is completed, click **Download exported data**. It is saved as `Ti - 6AI - 4V, quenched and tempered.xlsx`.



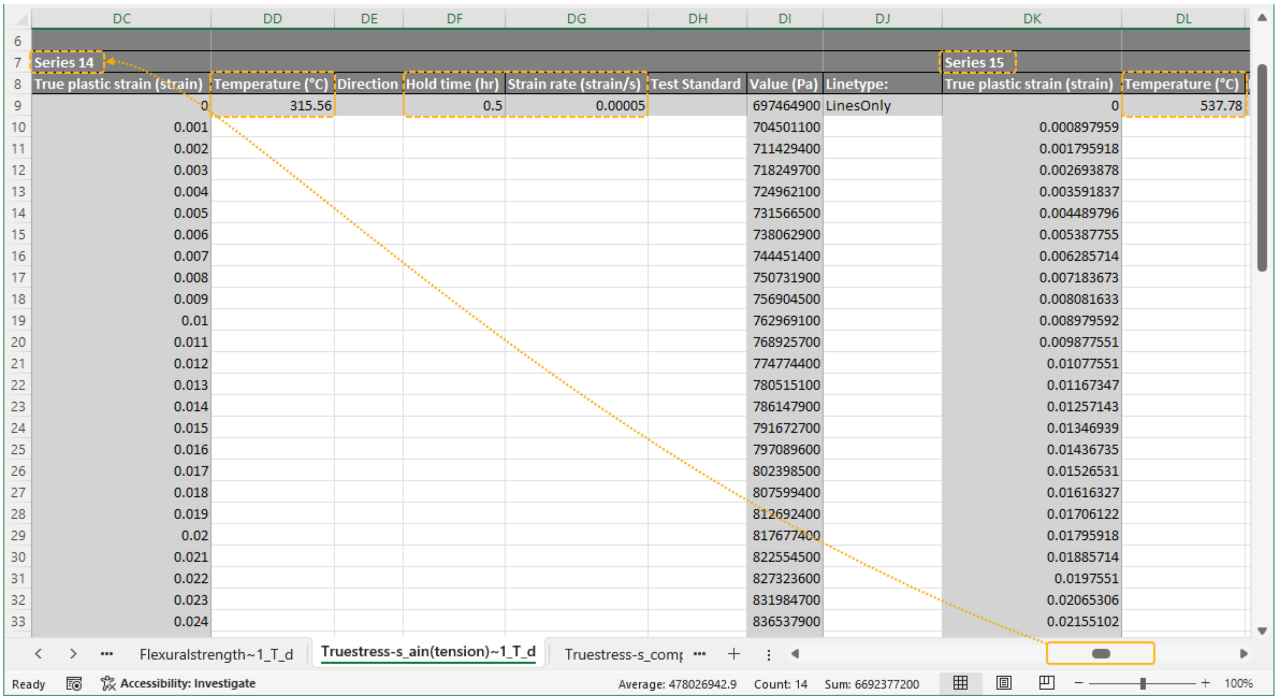


Use a spreadsheet to derive data for Bilinear Isotropic Hardening model

Open the `Ti - 6 Al - 4 V, quenched & tempered.xlsx` file. At the bottom of the page, navigate to the sheet called **Truestress-s_ain(tension)**.

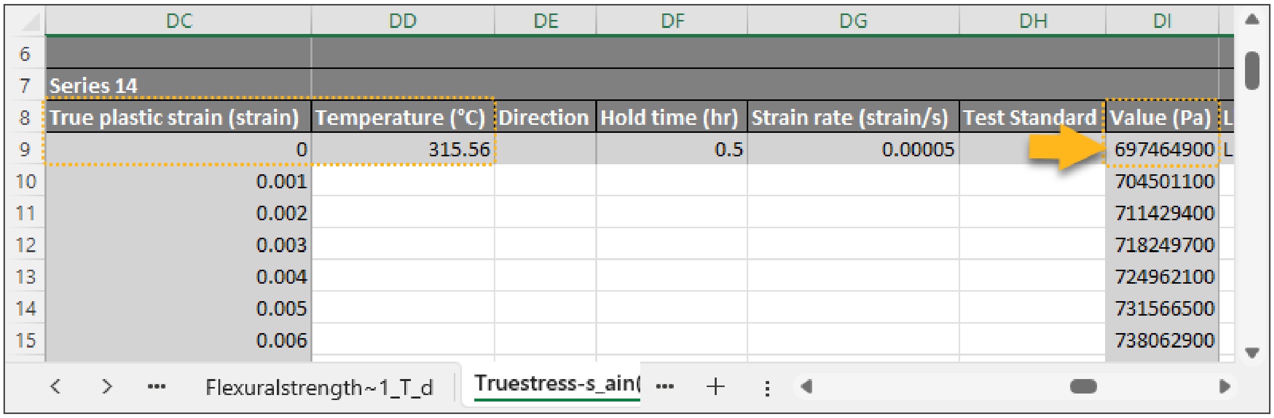


Scroll to the right to find the data in series 14: Temperature = 315.56°C, Hold time = 0.5 hr, and Strain rate = 0.00005 strain/s. Continue to scroll to the right to see the data in series 15: Temperature = 537.78°C, and in series 16: Temperature = 204.44°C.

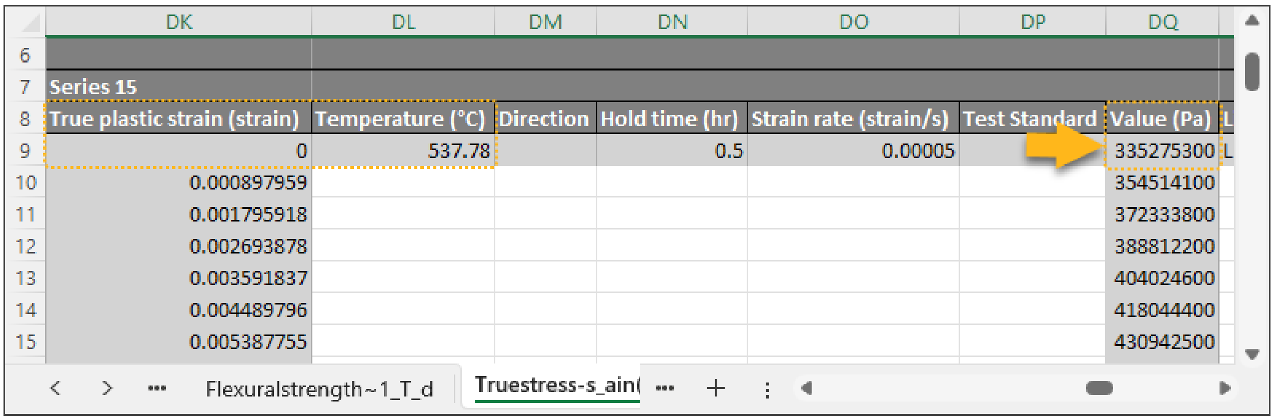


	DC	DD	DE	DF	DG	DH	DI	DJ	DK	DL
6										
7	Series 14								Series 15	
8	True plastic strain (strain)	Temperature (°C)	Direction	Hold time (hr)	Strain rate (strain/s)	Test Standard	Value (Pa)	Linetype:	True plastic strain (strain)	Temperature (°C)
9	0	315.56		0.5	0.00005		697464900	LinesOnly	0	537.78
10	0.001						704501100		0.000897959	
11	0.002						711429400		0.001795918	
12	0.003						718249700		0.002693878	
13	0.004						724962100		0.003591837	
14	0.005						731566500		0.004489796	
15	0.006						738062900		0.005387755	
16	0.007						744451400		0.006285714	
17	0.008						750731900		0.007183673	
18	0.009						756904500		0.008081633	
19	0.01						762969100		0.008979592	
20	0.011						768925700		0.009877551	
21	0.012						774774400		0.01077551	
22	0.013						780515100		0.01167347	
23	0.014						786147900		0.01257143	
24	0.015						791672700		0.01346939	
25	0.016						797089600		0.01436735	
26	0.017						802398500		0.01526531	
27	0.018						807599400		0.01616327	
28	0.019						812692400		0.01706122	
29	0.02						817677400		0.01795918	
30	0.021						822554500		0.01885714	
31	0.022						827323600		0.0197551	
32	0.023						831984700		0.02065306	
33	0.024						836537900		0.02155102	

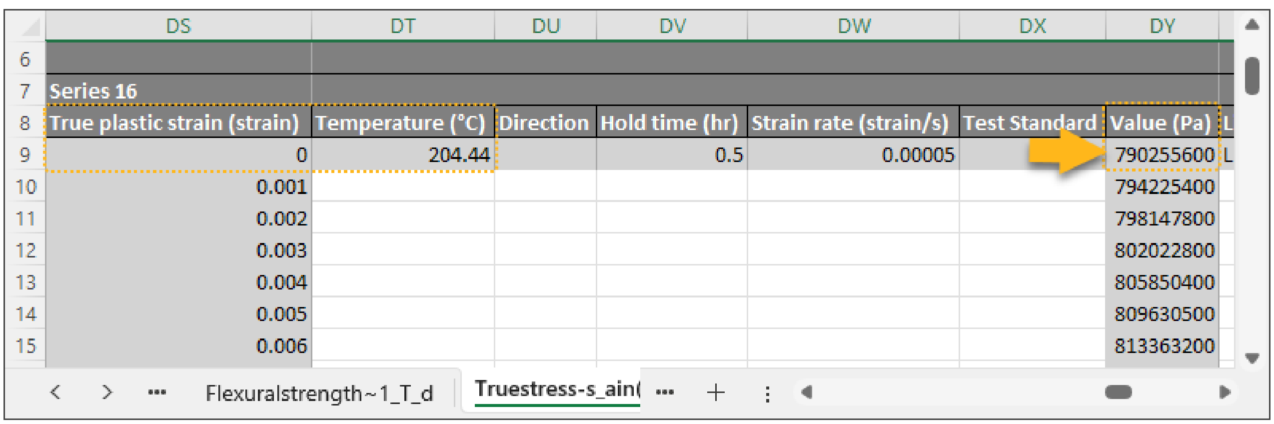
For each temperature, the true stress value at true plastic strain = 0 indicates the Yield Strength. We will enter these values into Engineering Data later.



	DC	DD	DE	DF	DG	DH	DI
6							
7	Series 14						
8	True plastic strain (strain)	Temperature (°C)	Direction	Hold time (hr)	Strain rate (strain/s)	Test Standard	Value (Pa)
9	0	315.56		0.5	0.00005		697464900
10	0.001						704501100
11	0.002						711429400
12	0.003						718249700
13	0.004						724962100
14	0.005						731566500
15	0.006						738062900



	DK	DL	DM	DN	DO	DP	DQ
6							
7	Series 15						
8	True plastic strain (strain)	Temperature (°C)	Direction	Hold time (hr)	Strain rate (strain/s)	Test Standard	Value (Pa)
9	0	537.78		0.5	0.00005		335275300
10	0.000897959						354514100
11	0.001795918						372333800
12	0.002693878						388812200
13	0.003591837						404024600
14	0.004489796						418044400
15	0.005387755						430942500



	DS	DT	DU	DV	DW	DX	DY
6							
7	Series 16						
8	True plastic strain (strain)	Temperature (°C)	Direction	Hold time (hr)	Strain rate (strain/s)	Test Standard	Value (Pa)
9	0	204.44		0.5	0.00005		790255600
10	0.001						794225400
11	0.002						798147800
12	0.003						802022800
13	0.004						805850400
14	0.005						809630500
15	0.006						813363200

For simplicity, we assume the tangent modulus is the slope of the true plastic stress-strain curve. For full documentation on how to calculate the tangent modulus, refer to Bilinear Isotropic Hardening.

$$Plastic\ \ Tangent\ \ Modulus=\frac{Final\ Stress\ Value{-}First\ Stress\ Value}{Final\ Plastic\ Strain\ Value}$$

or

$$E_p=\frac{\sigma_{Final}-\sigma_{Initial}}{\varepsilon_{Final}}$$

Calculate that in the spreadsheet using a formula as follows:

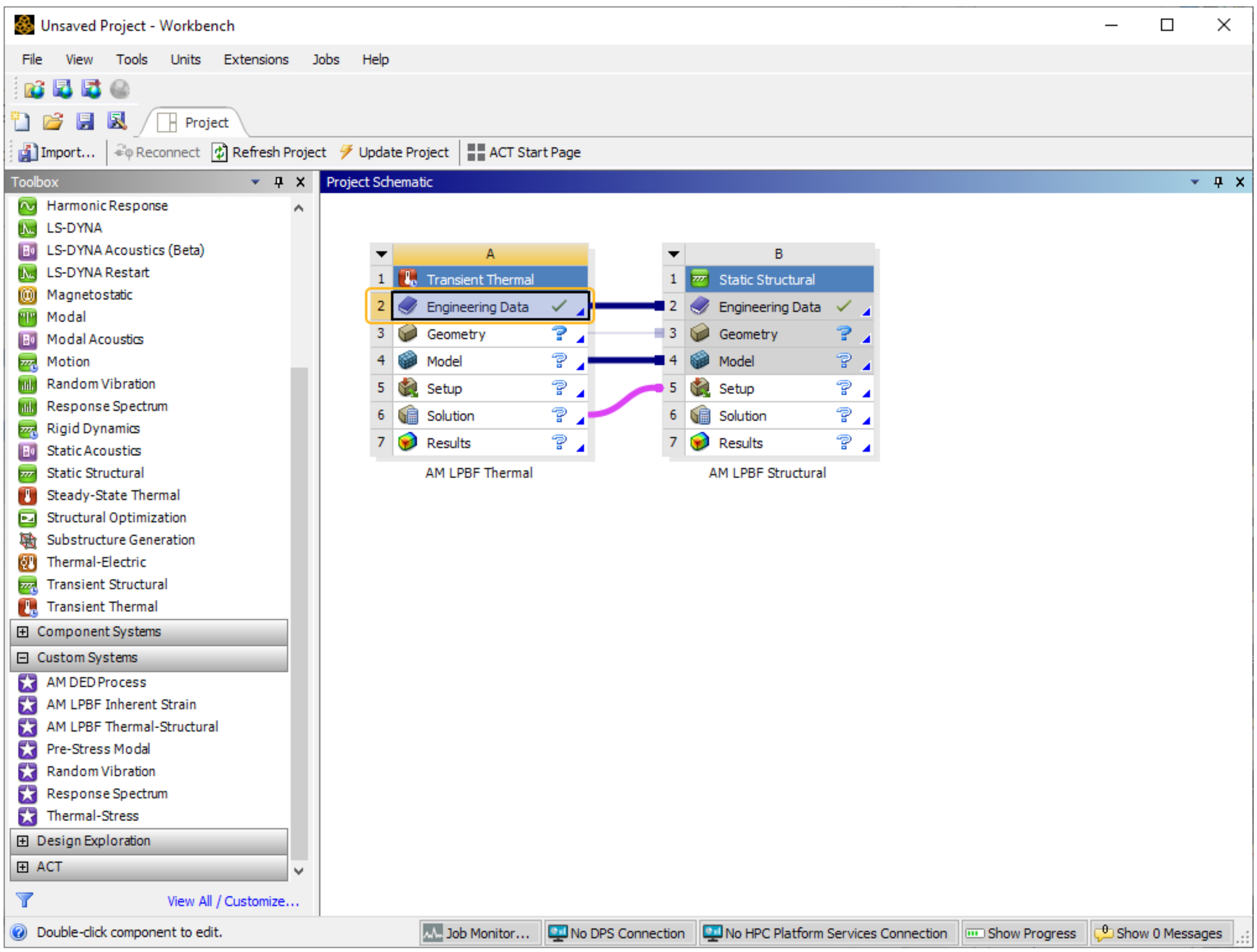
The Tangent Modulus for each temperature curve is highlighted. The following tables summarizes our results.

Series # in spreadsheet	Temperature °C	Yield Strength (Pa)	Tangent Modulus (Pa)
16	204.44	790255600	2618222728
14	315.56	697464900	3959473439
15	537.78	335275300	5347750000

These values will be entered into Engineering Data in the next step. The Yield Strength and Tangent Modulus together comprise the data for the Bilinear Isotropic Hardening model.

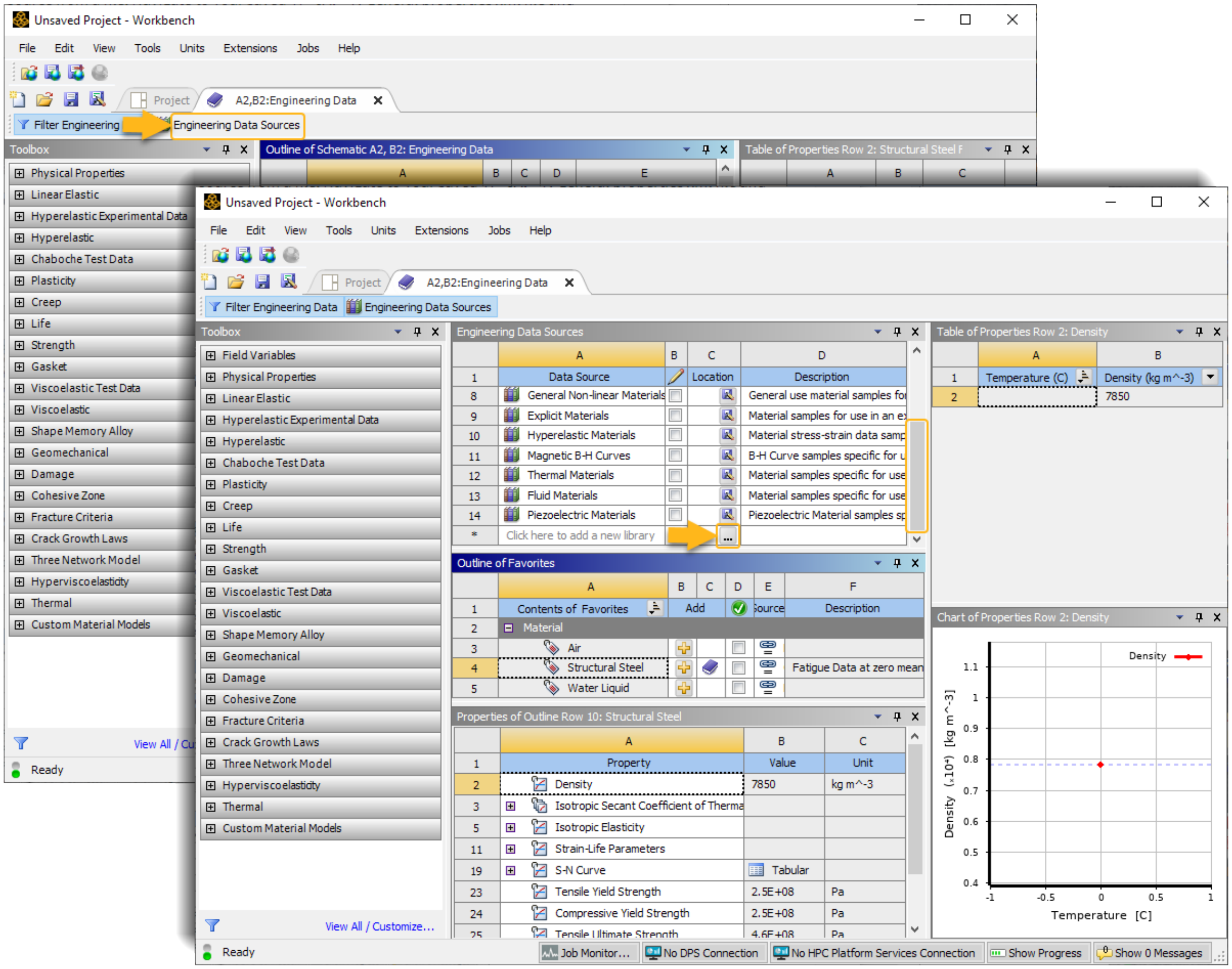
6.3.2.2. Enter JAHM Material into Engineering Data

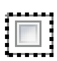
Open Workbench, and add an additive analysis system as you normally do to begin a simulation. Here we added an **AM LPBF Thermal-Structural** system. Double-click **Engineering Data** to open it.

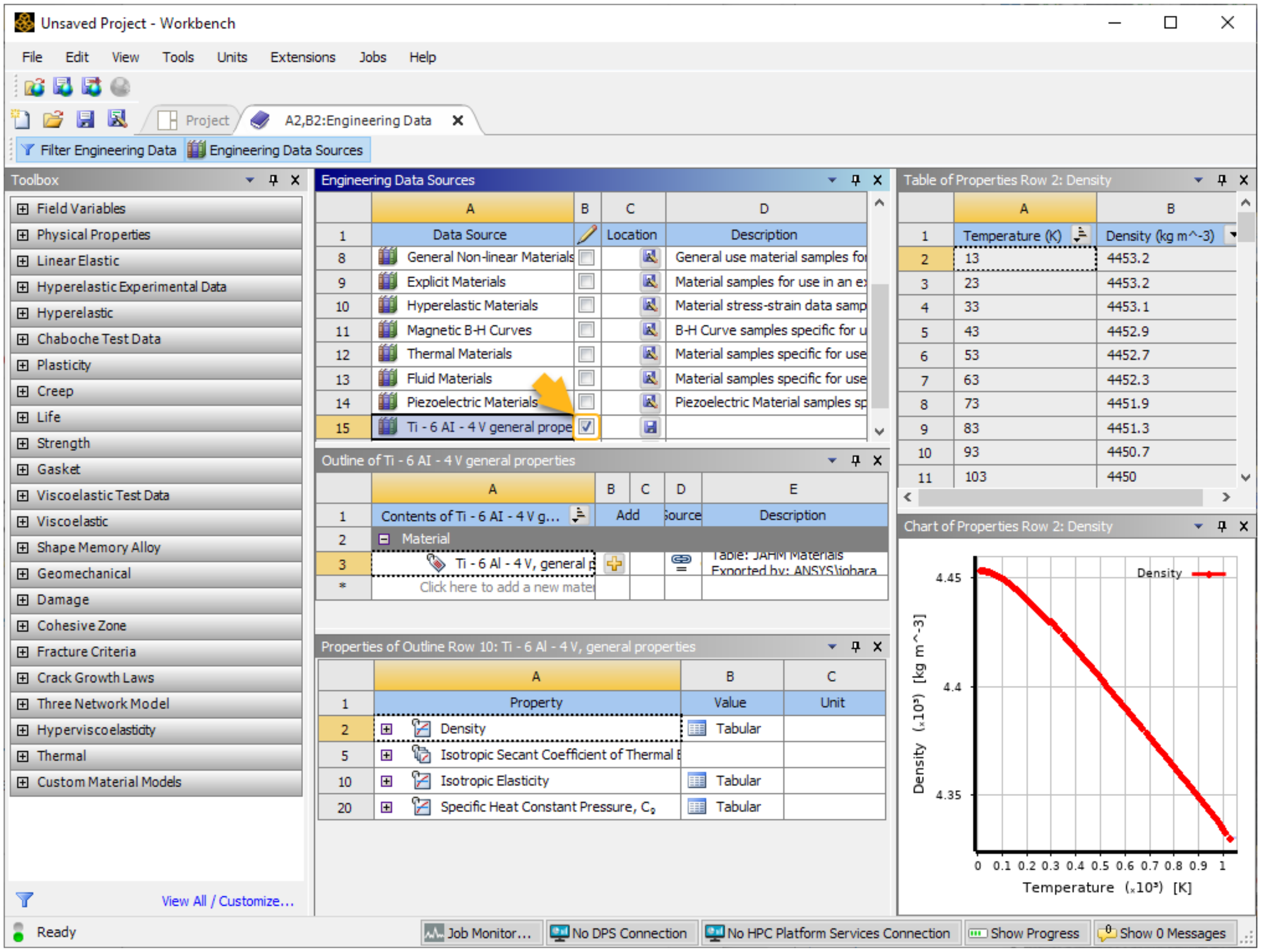


Import general properties material card

Click **Engineering Data Sources** and scroll down to the  button to add an existing data source from a file. Navigate to your saved Ti - 6Al - 4V general properties xml file and click **Open**.



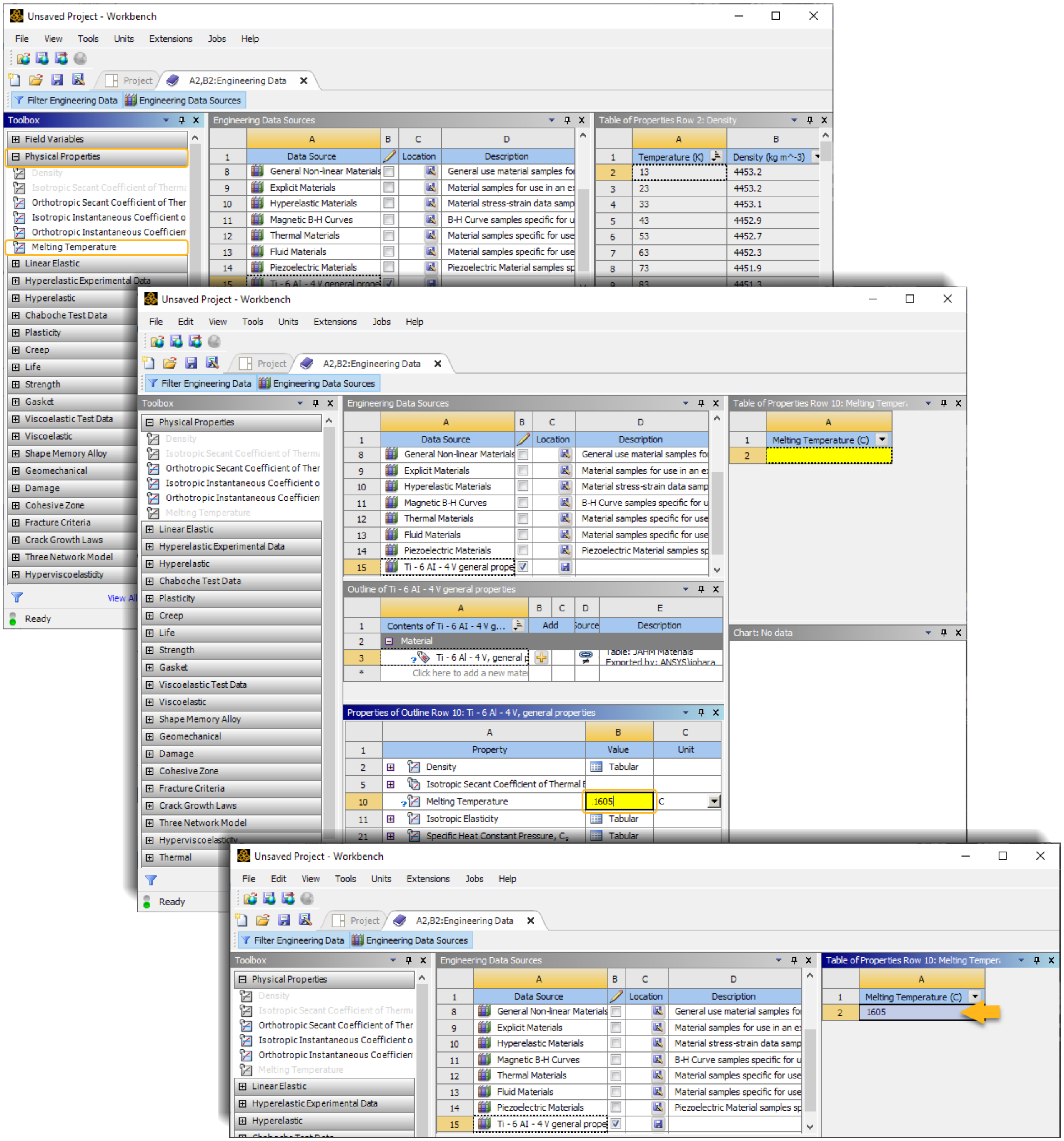
Click the check box  in column B to edit the material.



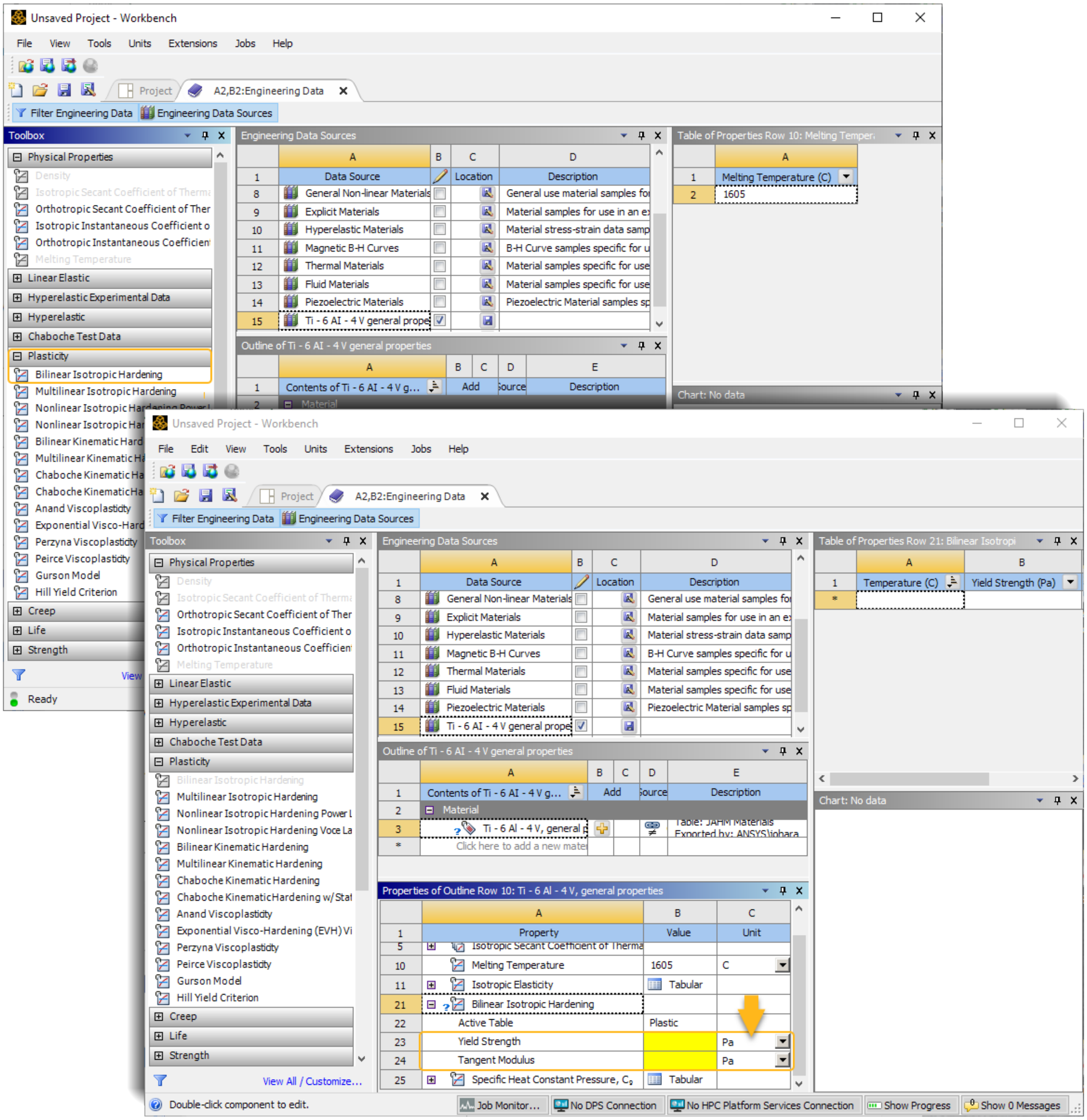
A review of the properties reveals that Melting Temperature and a Bilinear Isotropic Hardening model are missing.

Add missing properties

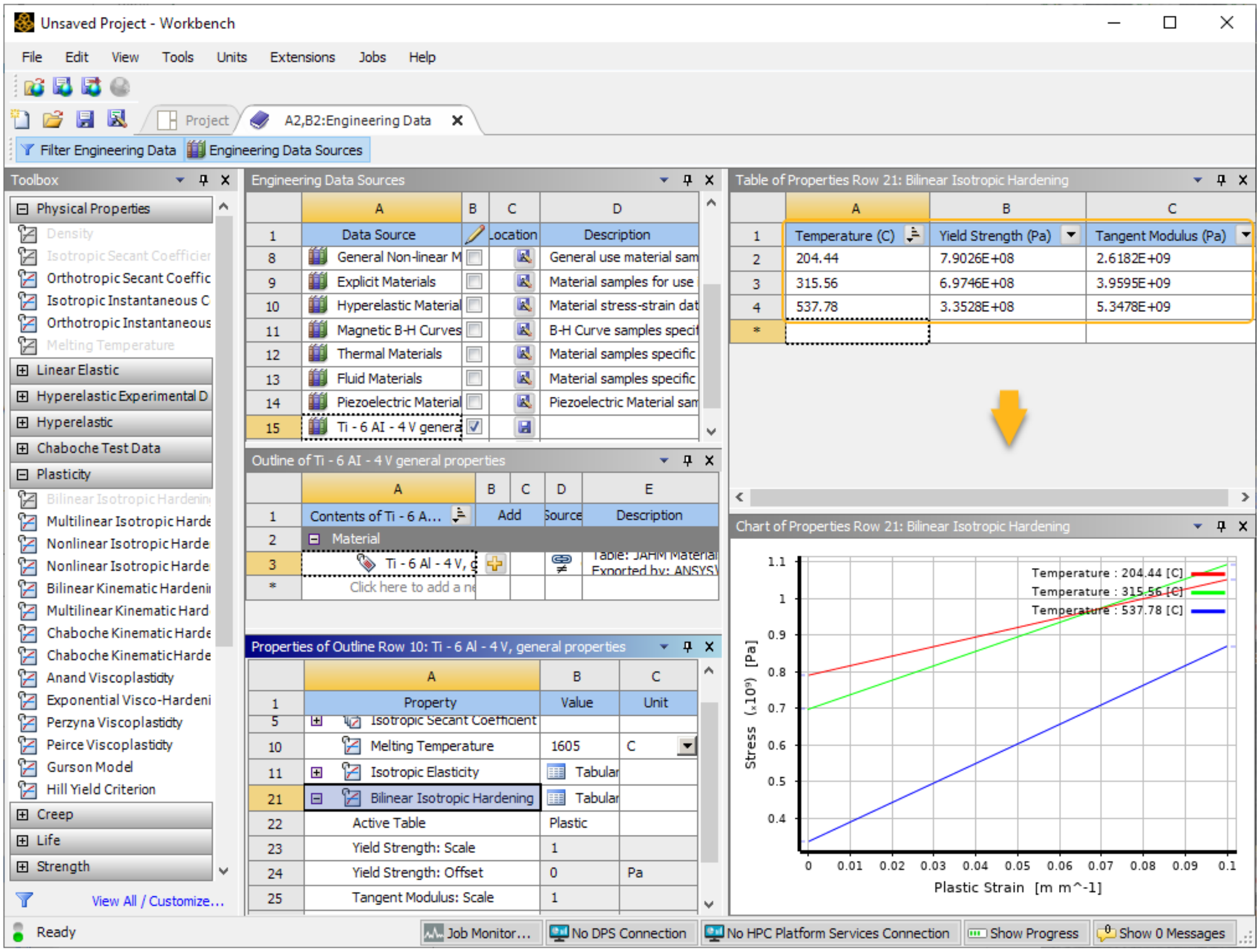
To add the missing Melting Temperature, in the Toolbox on the left side panel, open **Physical Properties** and double-click **Melting Temperature**. This adds the property to the material being edited and shows a yellow highlight for the missing value. Enter **1605** in the highlighted area next to Melting Temperature. The value of 1605°C is the Melting Temperature from the Ansys Ti-6Al-4V sample material.



Do the same to add the **Bilinear Isotropic Hardening** model from the **Plasticity** options in the Toolbox panel. Next to the highlighted input fields, be sure to use the appropriate units; in our example we are using Pascals for both Yield Strength and Tangent Modulus.

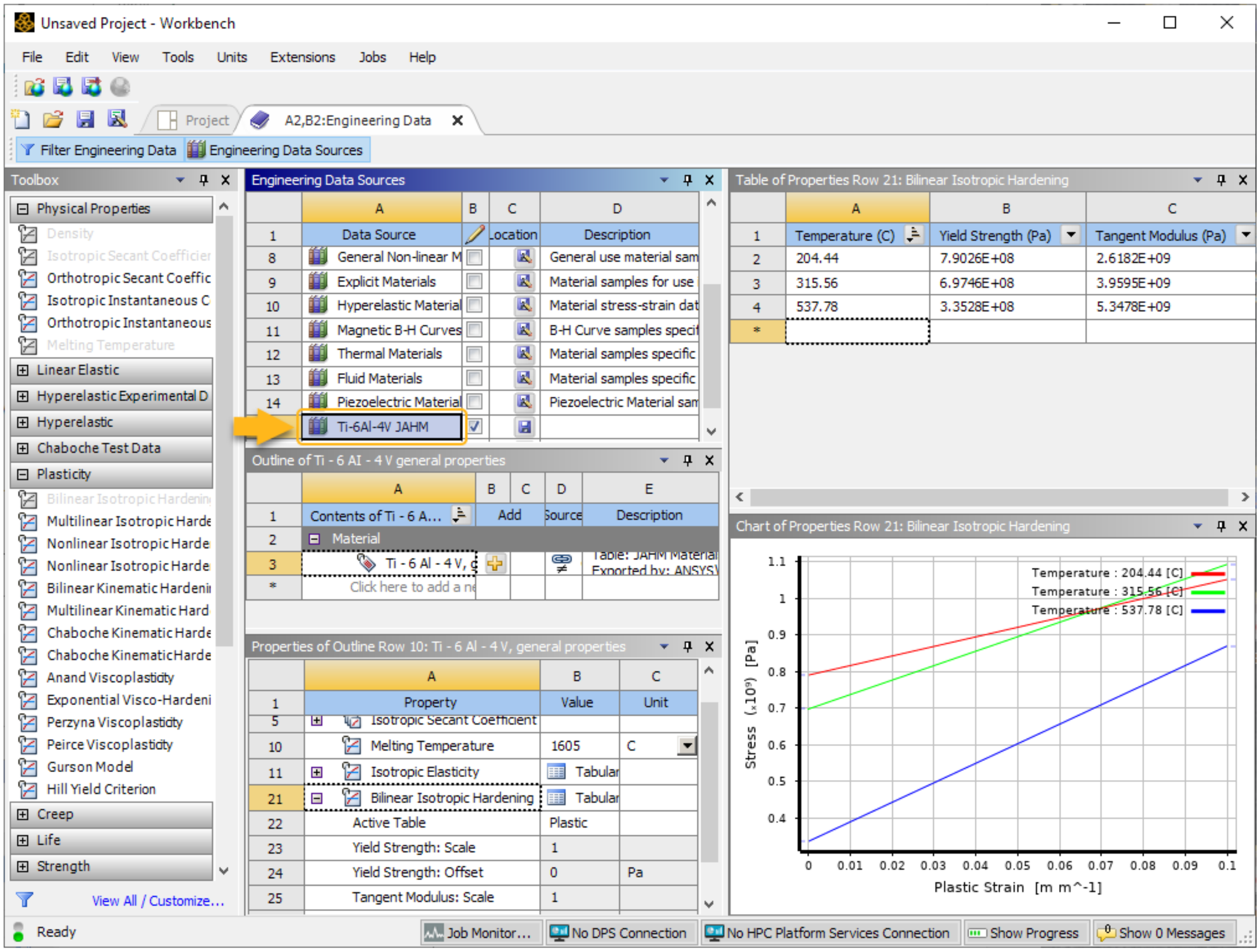


Double-click either highlighted field and then, in the upper right table, enter the values for Temperature, Yield Strength, and Tangent Modulus that you calculated from the spreadsheet.

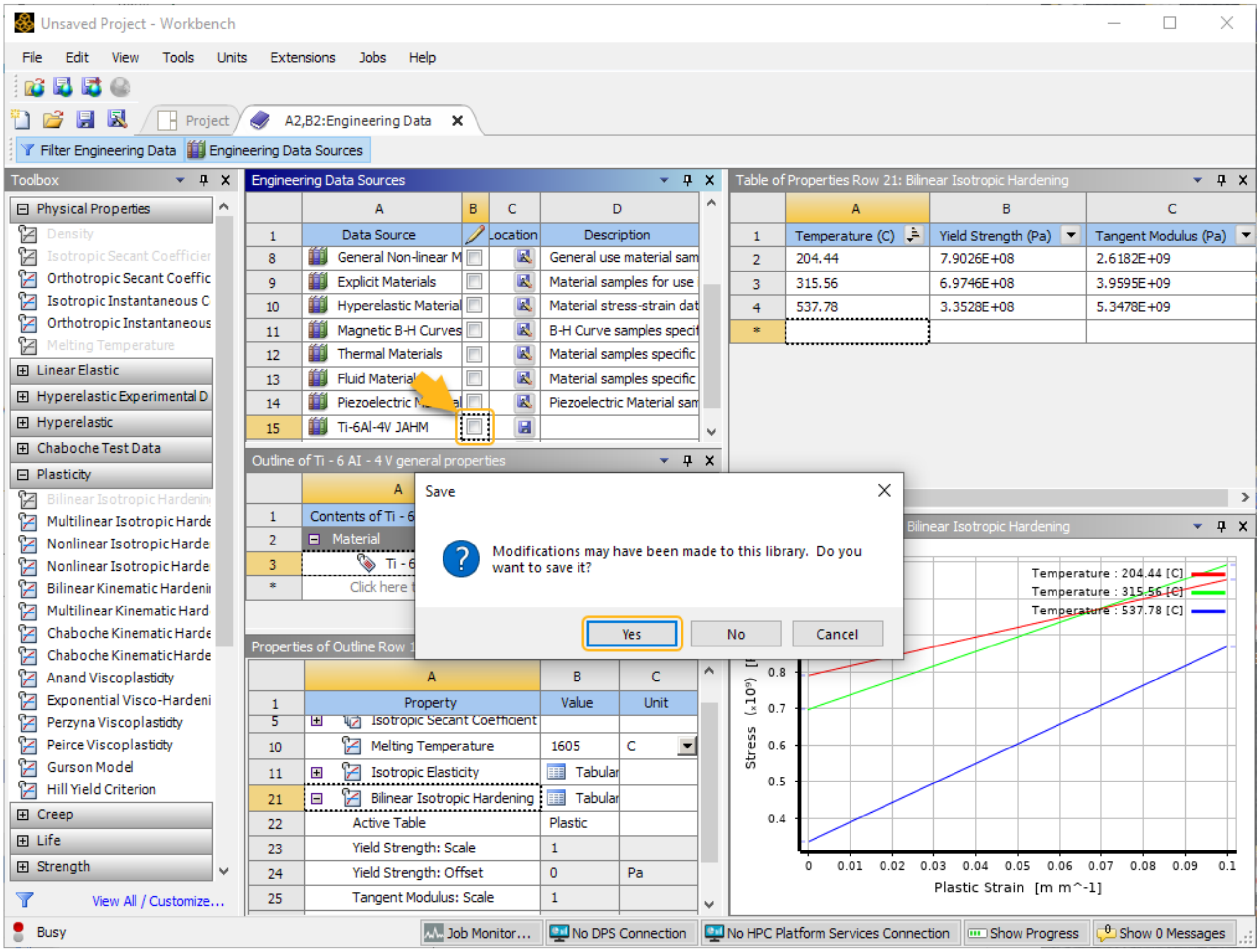


Rename and save the new material in Engineering Data

Now that all the missing data has been entered, the new material is complete and must be saved in Engineering Data for future use. First rename the material. In the Engineering Data Sources panel, **click the material name field** and enter a new name. We used Ti-6Al-4V JAHM.

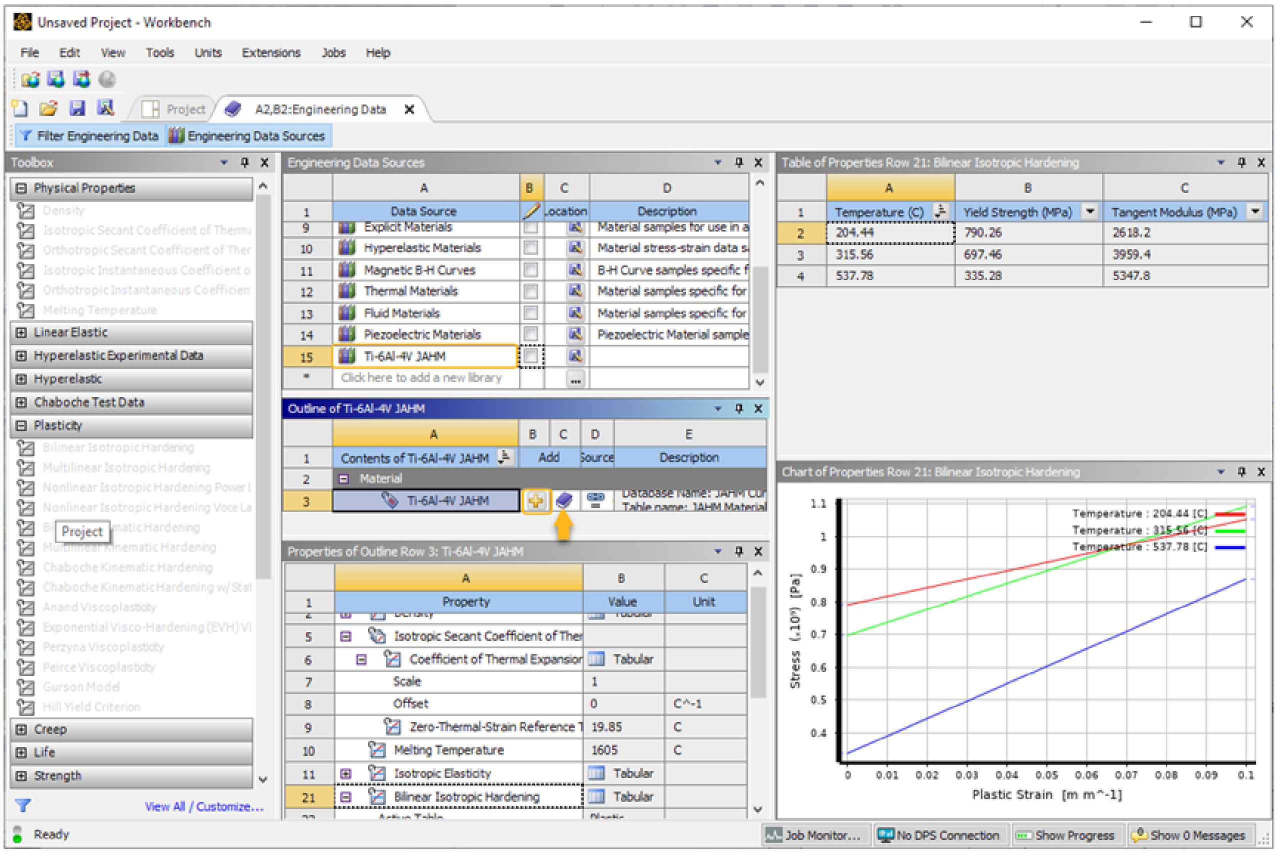


Clear the check box that allows editing of data and you will be prompted to save the material. Click **Yes**. The new material is saved in Engineering Data and to your original xml file.



6.3.2.3. Add JAHM Material to a Project and Assign it to a Geometry in a Simulation

Any time you want to use the material in a simulation, you must first add it to the project in Workbench's Engineering Data. Navigate to the new JAHM material in Engineering Data Sources and click the plus icon . The book icon indicates it has been added to the current project.

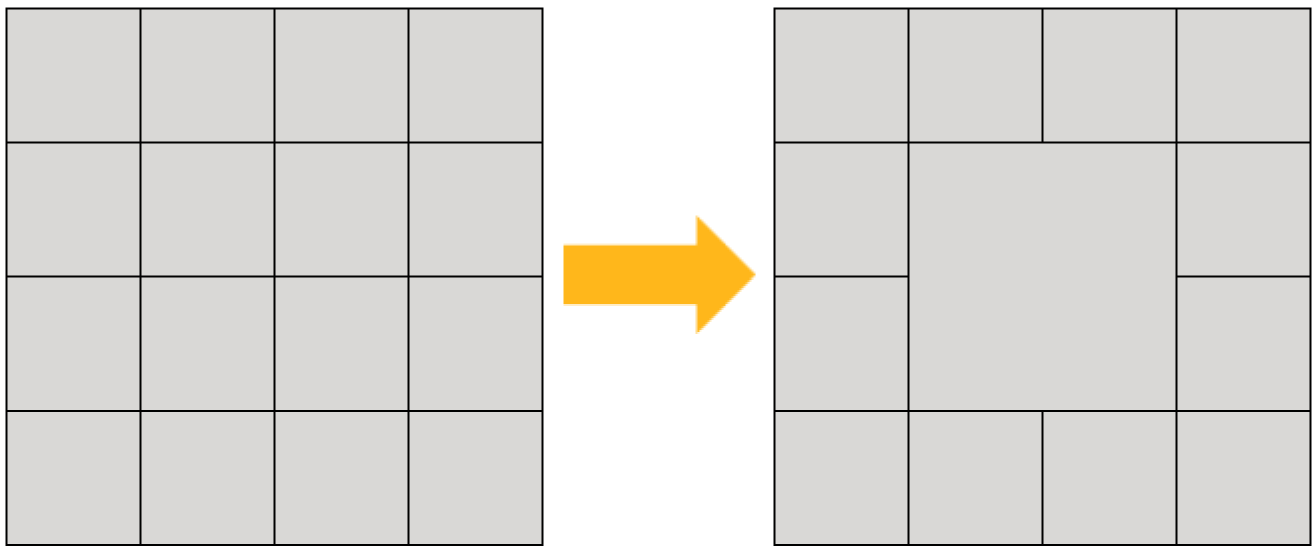


Work through the simulation as usual and then assign the new material for the appropriate geometry bodies.

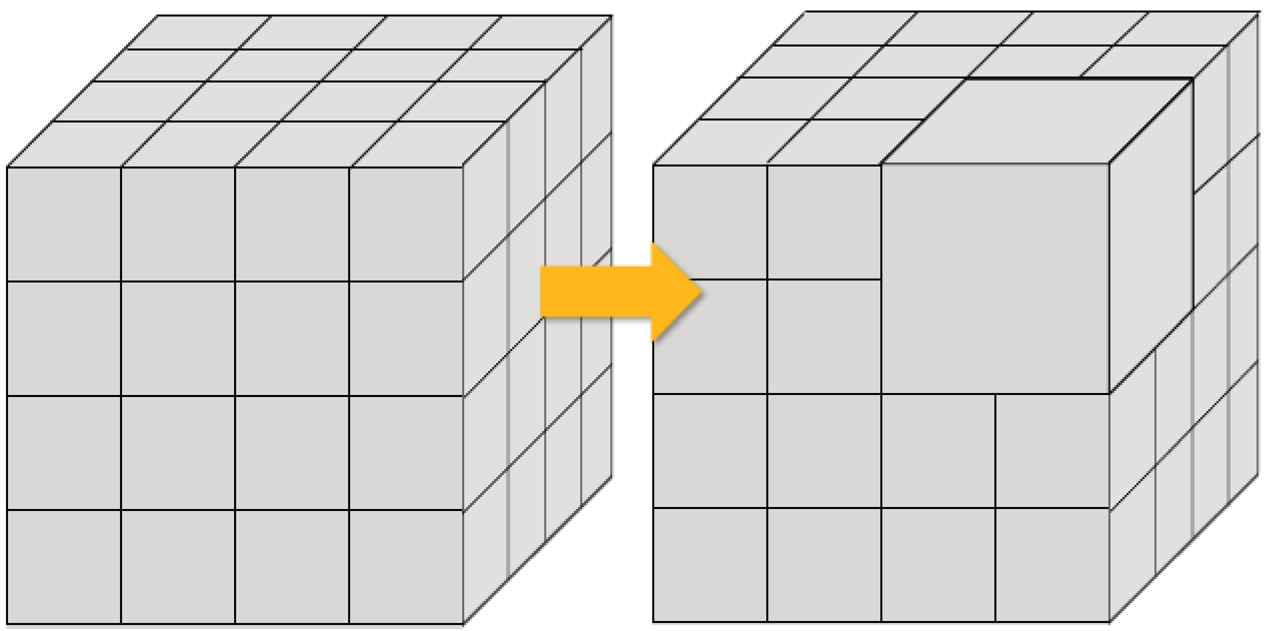
6.4. Using AM Octree Adaptive Meshing

For additive manufacturing Inherent Strain simulations, a special case of nonlinear mesh adaptivity is available in which the total number of elements is reduced by *coarsening* elements in previously deposited layers. This may help in reducing simulation time in specific cases.

A simple 2D representation of mesh coarsening is shown below, in which four square elements are combined into one larger square element.

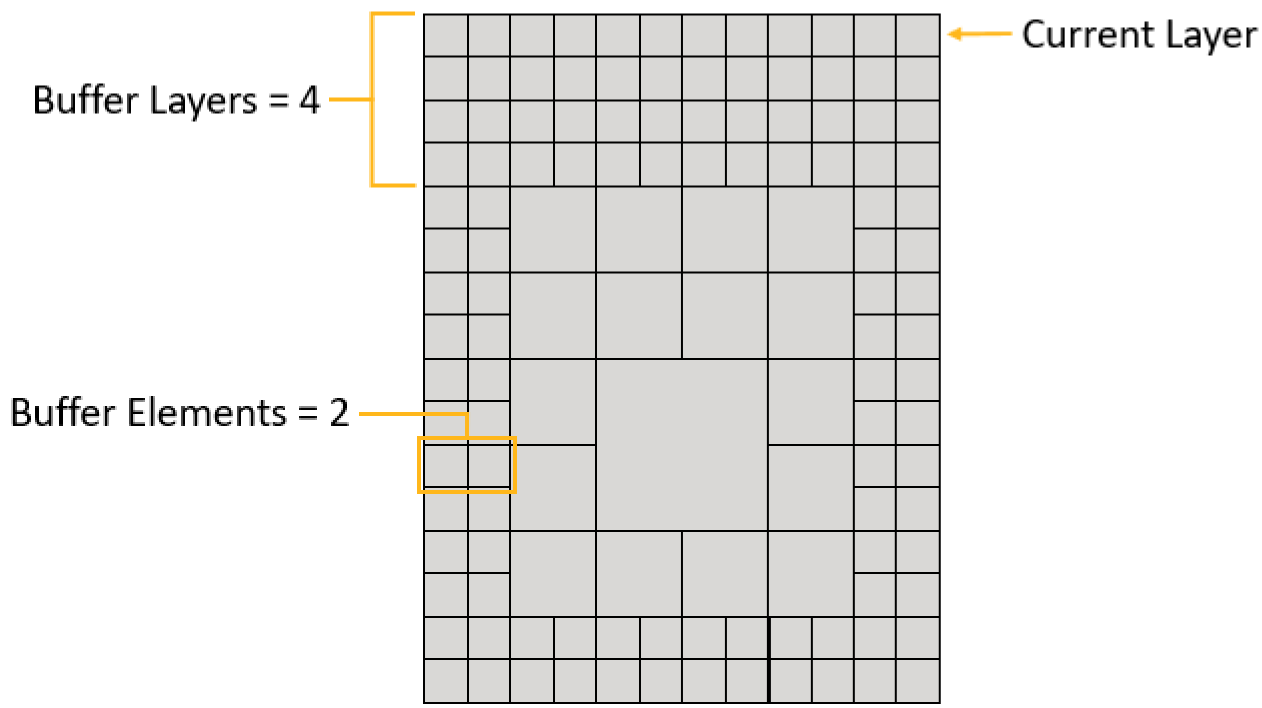


The AM octree method was created specifically for use with laser powder bed fusion simulations, which commonly use cubic (voxelized) elements. The octree method combines eight cubic elements into a new, single cubic element, as shown in this 3D representation.



Nonlinear mesh adaptivity modifies the mesh automatically during the solution based on specified criteria. Mesh modifications occur by general remeshing with the AM octree technique. Loads, boundary conditions, contact conditions, solutions variables, etc., are seamlessly transferred to the new mesh as the solution progresses. Aside from setting the initial criteria, no user action is required.

AM octree adaptive meshing criteria include the specification of layers designating when remeshing will start and end, buffer regions, both vertical layers and edge elements, to control where a fine mesh is maintained, and layer frequency to modify how often the program remeshes elements. The figure below shows four buffer layers and two buffer elements.



The goal of AM octree adaptive meshing is to reduce simulation time. However, keep in mind that remeshing takes time, so remeshing too frequently may not provide a simulation time benefit.

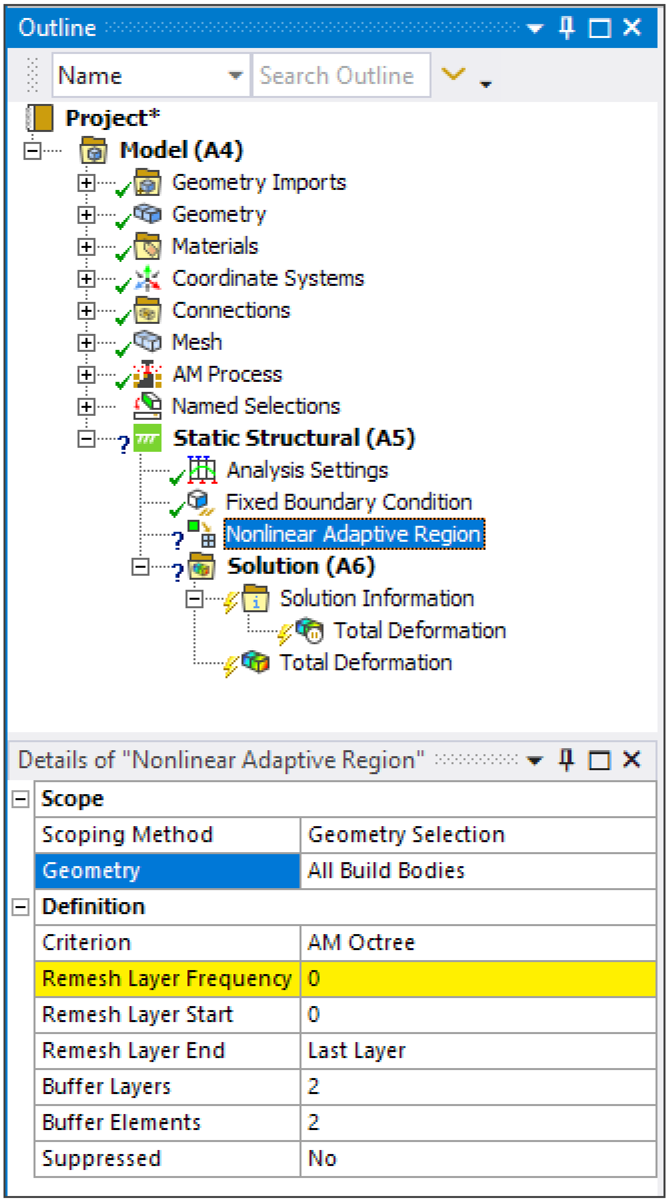
Procedural Steps

The AM octree method is a special case of nonlinear mesh adaptivity designed for mesh coarsening. To use AM octree adaptive meshing:

1. Set up an AM LPBF Inherent Strain (p. 138) simulation in the usual way and be sure to use voxilized meshing (p. 78) (Mesh Method with Method = “Cartesian” and Mesh Using Voxelization = “Yes”).

2. Apply a Nonlinear Adaptive Region. The Nonlinear Adaptive Region condition enables you to change the mesh during the solution phase. It acts as a remesh controller based on certain criteria. The criteria determine whether or not the mesh will be modified and, if so, when it will be modified and which locations will be modified.

   On the **Environment** context tab: click **Conditions** > **Nonlinear Adaptive Region**. (Or, right-click the Static Structural tree object, or anywhere in the geometry window, and select Insert > Nonlinear Adaptive Region.)



3. Set remesh criteria:

   - Scoping will always read All Build Bodies if AM Process is in the project tree, however supports will not be considered for mesh coarsening. Similarly, the base plate is not taken into account.

   - **Criterion**: This is set to AM Octree automatically if AM Process is in the project tree.

   - **Remesh Layer Frequency**: Frequency with which the program will perform remeshing. A value of 4 will remesh every 4 layers from Remesh Layer Start to Remesh Layer End.

   - **Remesh Layer Start**: The layer at which remeshing will start. Value must be greater than or equal to 0.

   - **Remesh Layer End**: The layer at which remeshing will end. Value must be greater than or equal to 0. A value of 0 (default) will display “Last Layer” and will continue to remesh through the end of the build.

- **Buffer Layers**: Number of layers to keep at fine mesh resolution between the current, top layer and the layers to be remeshed. Defaults to 2.

- **Buffer Elements**: Number of buffer elements to keep at fine mesh resolution between the part edges and the remeshed elements. Defaults to 2.

For both buffer inputs, higher values may ensure better accuracy, but reduces the number of elements that can be coarsened.

4. Solve the simulation.

5. View the coarsened mesh by selecting a result item and then using a section plane to see inside the part. Hint: The best way to see the effect of remeshing is to make a clean, vertical slice through the model.

6. View the time steps at which remeshing took place by looking for Changed Mesh = Yes in tabular data when a result item or Solution is selected.

Notes on Usage and Known Limitations

Following are considerations and limitations when using AM octree adaptive meshing:

- AM octree adaptive meshing is available only for LPBF Inherent Strain simulations. (Because the octree technique is used in the structural solver, it is not available for the thermal portion of a thermal-structural simulation or even the structural step of a thermal-structural simulation where temperature mapping is based on a consistent mesh between the thermal and structural analyses.)

- AM octree adaptive meshing is available only with a voxelized mesh (Cartesian mesh with voxelization option (p. 82)).

- AM octree adaptive meshing supports models with multiple build bodies as long as the voxel grid between bodies is consistent.

- Cubic elements that comprise the part are considered eligible for remeshing. The support, the base plate, and any other non-cubic elements are not taken into account.

- Cubic elements with a knockdown factor less than 1, such as those found around edges or in narrow areas that are not completely dense with material, will also not be considered for remeshing.

- A simulation can contain multiple nonlinear adaptive regions. This gives you more control and customization over when to remesh, which improves remeshing utilization and could result in significant time savings if used effectively. Consider using a nonlinear adaptive region whenever the build reaches a point where, during a remesh, many elements can be coarsened to maximize time savings. Multiple remeshes can take a significant amount of time, so limiting the number of remeshes to occur in areas where more elements can be coarsened will bring the greatest time savings.

- Because AM octree adaptive meshing occurs during solution, the coarsened mesh is visible via section plane only on *results displays*.

- The LPBF High Strain result is not available when using AM octree adaptive meshing.

6.5. Using Variable Layer Height

The variable layer height feature for additive manufacturing analyses allows you to vary the simulation's layer height to build with smaller layers in areas of interest, and larger layers in other areas of the part. You can capture fine details in a simulation without greatly increasing the overall simulation time by reducing the number of super-layer load steps that need to be solved. (See Methodology and Abstractions (p. 7)). You control this by setting the layer size (**AMBUILD**,LAYERT) and number of simulation layers to solve (**AMSTEP**,BUILD) iteratively until the part is complete.

Procedural Steps

This feature can be used by iteratively specifying the simulation layer size with the **AMBUILD** command, and the number of layers to build with the **AMSTEP** command. In Mechanical, these commands can be added to a Commands object under the analysis type. If variable layer height is used for a thermal-structural analysis, both the thermal analysis and the structural analysis must have the same sequence of layer sizes and layers built.

Create commands to use variable layer height:

1. Click the **Static Structural** object and select **Commands**  from the context menu. Or, right-click and select **Insert** > **Commands**. In the Commands window that is displayed, type something like the following, modifying the specific numbers to fit your part:

```
! Build 10 layers at 1 mm
AMBUILD,LAYERT,0.05,1
AMSTEP,BUILD,,10

! Build 6 layers at 0.25 mm
AMBUILD,LAYERT,0.05,0.25
AMSTEP,BUILD,,6

! Build the remaining layers at 1mm
AMBUILD,LAYERT,0.05,1
```

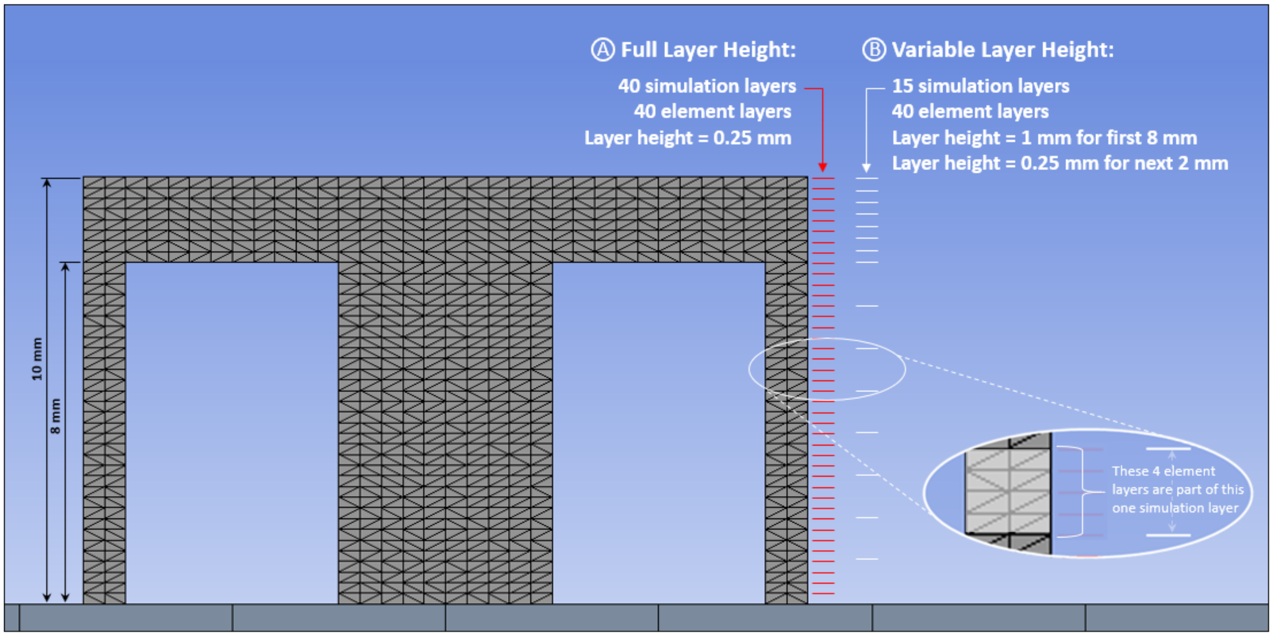
   Note:

   By default, the Mechanical application will send the AMSTEP,BUILD command after reading the Commands objects. If variable layer height is used in an independent script, the AMSTEP,BUILD command will need to be added to the end of the example script above.

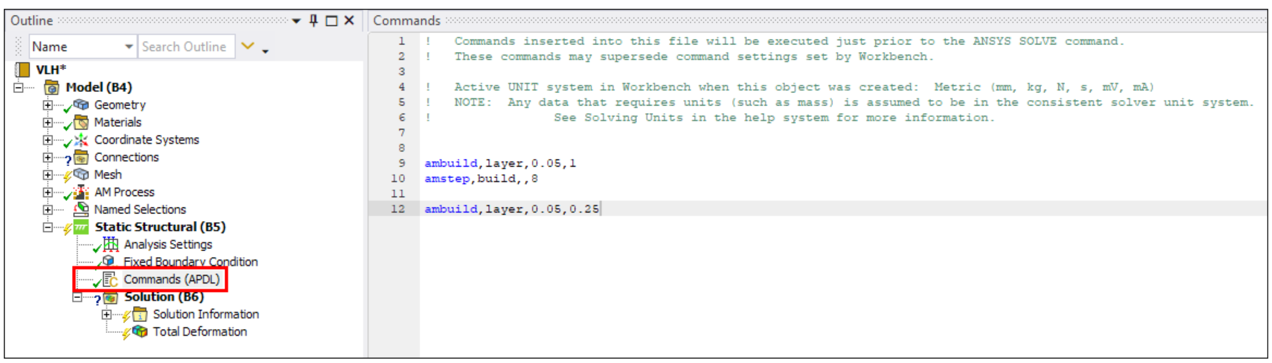
2. Repeat the above step for the **Transient Thermal** system if you have a thermal-structural analysis. (This step is not applicable for an AM simulation using Inherent Strain.)

Example

Following is an example to illustrate the variable layer height feature. The simple part has three columns that come together to form a beam at a height of 8 mm. We want to capture greater details at the location where the columns join together because we suspect that is where the greatest distortion will be.

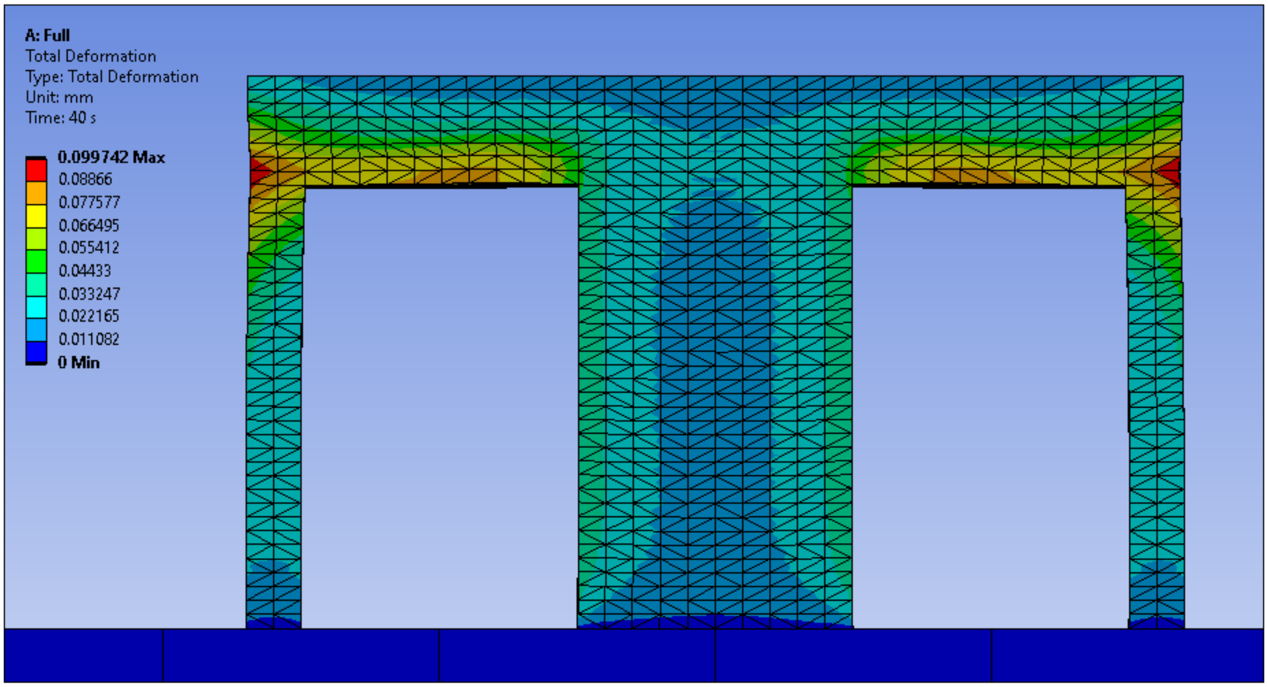


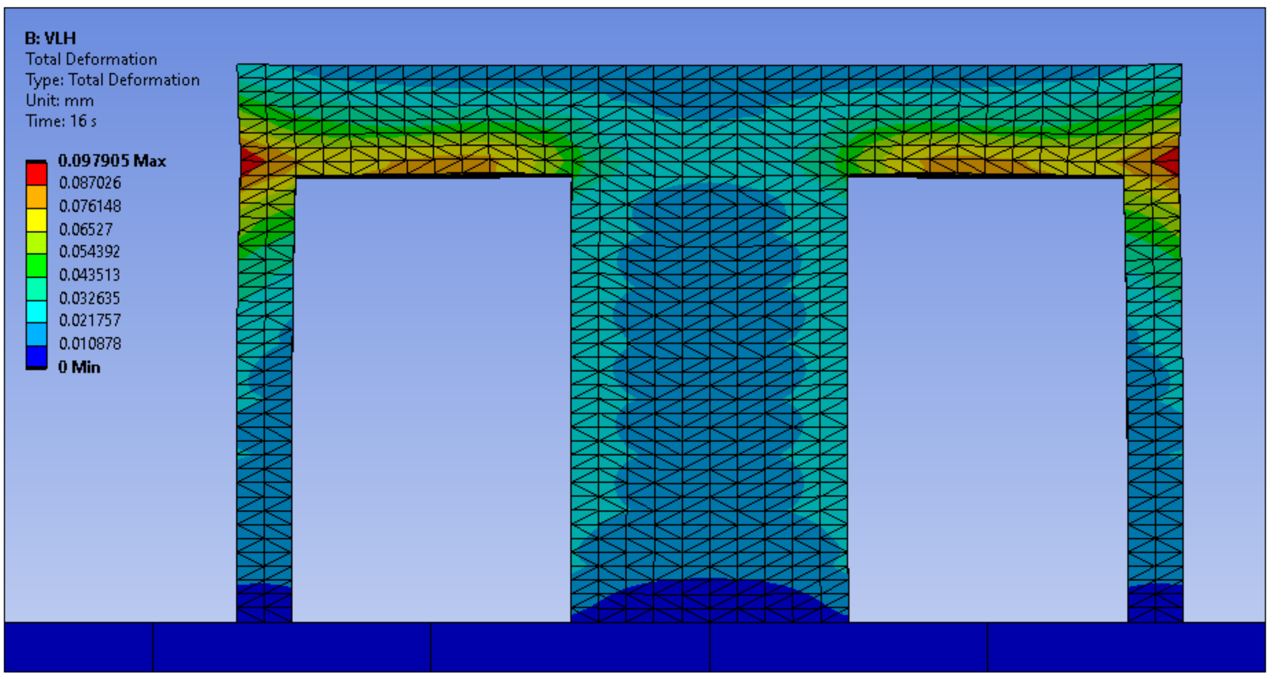
Using a Commands object inserted under the static structural system, we set up simulation layers with a height of 1 mm for the columns and a height of 0.25 mm after that for the beam. This simulation uses Inherent Strain so there is no transient thermal system.



Our results show comparable distortion at the location of interest and the variable layer height feature resulted in a 53.8% simulation time reduction for this model and variable layer height setup.

A. Full layer height	**B. Variable layer height**
Mesh layer size = 0.25 mm	Mesh layer size = 0.25 mm
Number of element layers = 40	Number of element layers = 40
Number of simulation layers = 40	**Number of simulation layers = 15**
Node count = 78,115	Node count = 78,115
Element count = 52,376	Element count = 52,376
Solution time = 6 min 30 sec (10 cores)	**Solution time = 3 min 30 sec** (10 cores)





Notes on Usage and Known Limitations

When using variable layer height:

- Using large layer thicknesses may lead to artificially high deformations. It's always important to carefully review your results.
- In some cases, it may be necessary to suppress an error check for elements spanning multiple layers. Suppression for this check can be controlled in the **AMBUILD**,LAYERT command. A value of 1 can be

sent for the “Layer-checking suppression” field shown in the command input below. Be careful when using this option as it may lead to improper boundary conditions. This is a beta option at this release.

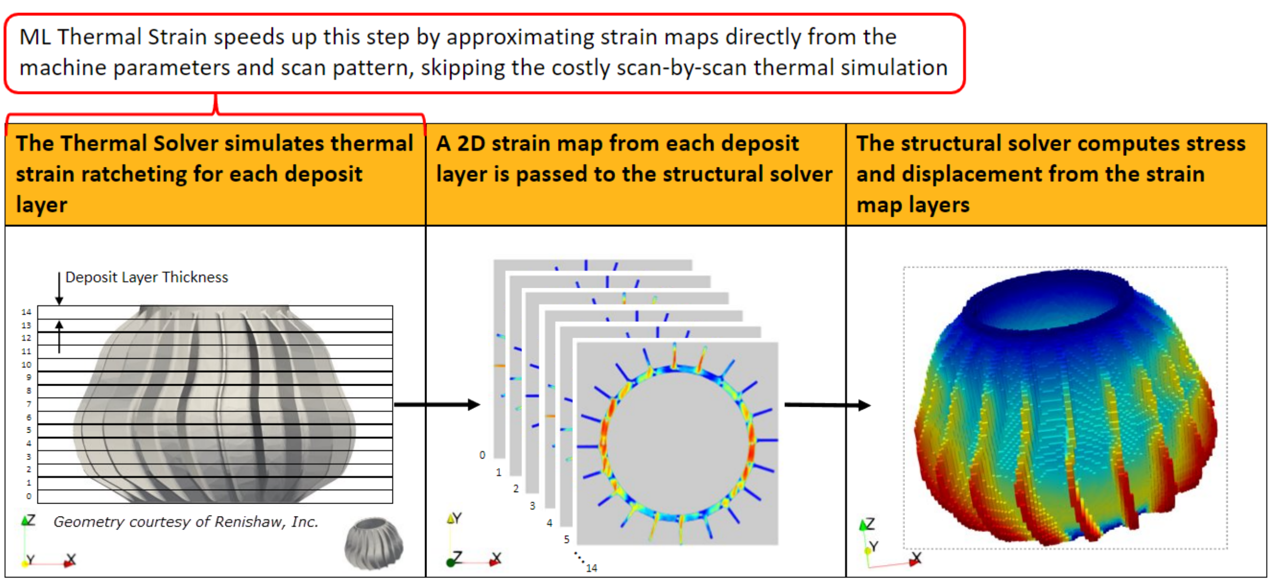
```
AMBUILD,LAYERT,Deposition thickness,Super-layer thickness,Layer-checking
suppression
```

- The progress bar may not accurately track the progress of the simulation.
- At this release, the estimate of the real machine build time in the output file will probably be incorrect.

6.6. Understanding Machine Learning Thermal Strain

Thermal Strain simulations, specifically those run using Additive Print's Thermal Strain simulation type, provide the highest level of fidelity at the cost of a much longer computational time. That is because Thermal Strain simulations predict how thermal cycling affects strain accumulation at each location within a part. The simulation follows the laser moving along each scan track on each deposit layer and is based on the machine process parameters (power, scan speed, beam diameter, etc.). See Thermal Strain - Anisotropic and Overview of the Thermal Solver in the *Additive Print and Science User's Guide*.

It can take millions of laser scan tracks to build a typical part with the laser powder bed fusion process, and performing a thermal simulation that follows the laser along each scan track can easily incur a computational cost that is prohibitive. Modern machine learning (ML) techniques provide a way to map from the machine parameters and scan pattern directly to the final outcome, skipping the costly scan-by-scan thermal simulation while providing a high-fidelity result that is computationally practical. Our ML Thermal Strain model, based upon the Thermal Strain simulation type in Additive Print and implemented in Mechanical, predicts the Thermal Strain result much faster than simulation. It can be one to three orders of magnitude faster than Thermal Strain simulation in calculating the strain that is passed to the structural solver. Speedup increases with part size, scan area, and melt pool size.



The ML model is based on a deep convolutional neural network that has been trained on an extensive database of Thermal Strain simulation results. No further data generation or training is required to use ML Thermal Strain. The database used to pre-train the ML model consisted of nearly one million layers from a variety of complex geometries. Machine parameters covered the entire range available when defining build settings, and were restricted to combinations with predicted single bead melt pool dimensions that meet the following constraints:

- Single bead melt pool depth must be greater than or equal to deposition thickness
- Single bead melt pool width must be greater than or equal to hatch spacing
- Single bead melt pool width divided by single bead melt pool depth must be greater than or equal to 1.25

When using ML Thermal Strain, violations of predicted single bead melt pool dimension constraints will trigger messages that recommend how to adjust machine parameters to resolve the violations. Note that the predicted single bead melt pool dimensions are an approximation and may not match results from a Single Bead simulation in Additive Science.

Porosity is neglected. All locations within the part are assigned at least the base strain. For this reason, and because the ML model is subject to error in approximating Thermal Strain results, ML Thermal Strain results may not be an exact match with Thermal Strain results.

We recommend you perform a Strain Scaling Factor calibration specifically for ML Thermal Strain. Do not use Strain Scaling Factor calibration that was completed for Thermal Strain.

Important:

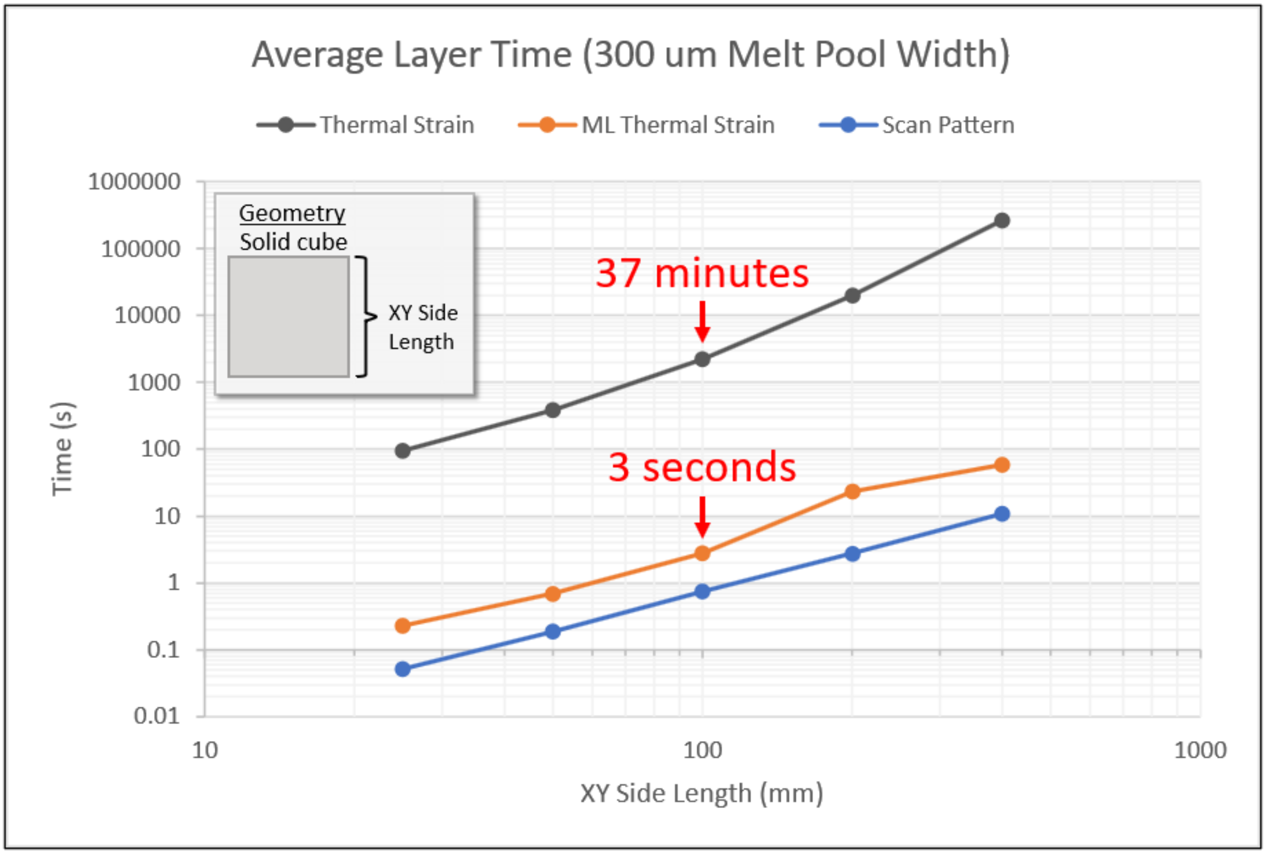
The Machine Learning Prediction feature requires a Structures AI+ license.

Frequently Asked Questions

- *How much of a speedup over Thermal Strain simulation can I expect when I choose ML Thermal Strain?*

  The following plot shows performance data for generating the *thermal strain for one deposit layer* in a sample part run in various strain modes in the Additive application. For an average layer time of 1 second, it would take ~1000 seconds for a 1000-layer part to complete the thermal strain mapping for the part. Overall, the speedup of strain generation with ML Thermal-Strain can be up to three orders of magnitude faster than Thermal Strain simulation and is comparable in speed to the Scan Pattern simulation type. Speedup increases with part size, scan area, and melt pool size. However, it is important to remember that the speedup from ML Thermal Strain is applicable to the thermal portion only of the overall simulation, and that the thermal strains are then passed to the structural solver to determine stress and deformation. To determine the overall simulation time, you need to add the time taken to solve the structural portion.

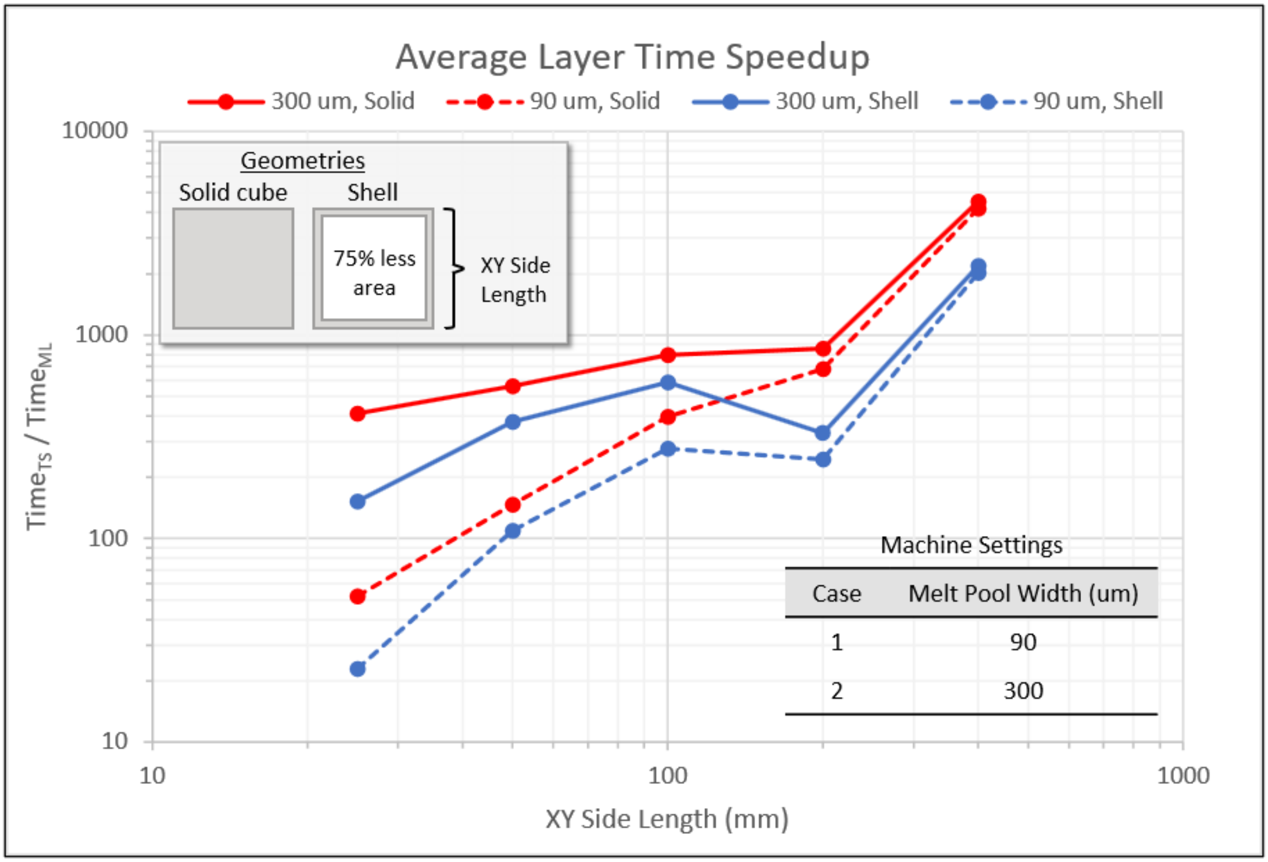
Time performance for thermal strain generation* per layer for various strain definitions (8 CPU threads)



* Thermal strains are then passed to structural solver to compute stress and distortion. Speedup shown represents thermal strain generation only.

The following plot demonstrates that the speedup of ML Thermal Strain over Thermal Strain simulation improves with increasing part size, scan area, and melt pool size. Melt pool width generally increases as energy is input to the system, such as by increasing power and/or beam diameter, and/or decreasing scan speed. As in the above plot, the data is from the generation of *thermal strain for one deposit layer*, and this data is from two sample parts with two sets of process parameters run in the Additive application. You can see a 20X to 4500X speedup in these scenarios.

Time performance for various setup scenarios (8 CPU threads)



- *Is the ML Thermal Strain capable of simulating my unique geometry without additional training?*

  Yes. Additional training is not needed. The ML model was trained with a collection of over 300 parts representing a wide range of geometries including organic shapes, bulky and thin features, state transitions, spiral and helical geometries, and many more.

- *Is there a limited range of machine input parameters I should stay within?*

  Yes. The table below shows the required ranges and recommended ranges, if applicable, for the machine parameters required for the ML model.

Parameter	**Required Range**	**Recommended Range**
Beam Power	50 to 700 Watts	50 to 500 Watts
Scan Speed	350 to 2500 mm/sec	500 to 2500 mm/sec
Beam Diameter	20 to 140 µm	80 to 120 µm
Deposition Thickness	10 to 100 µm	
Hatch Spacing	60[a] to 1000 µm	
Scan Stripe Width	1 to 100 mm	
Start Layer Angle	0 to 180°	
Layer Rotation Angle	0 to 180°	

Parameter	Required Range	Recommended Range
Preheat Temperature	20 to 500 °C	20 to 200 °C

[a] Different than Thermal Strain in Ansys Additive

- *What if I want to customize my material properties?*

  Since there are no material property inputs to the ML Model, we can only provide thermal strain results based on the material properties used to train the models. These are the materials validated for the Additive application. Choose the thermal Strain Material Model that most closely matches your material and then be sure to perform a calibration for ML Thermal Strain.

- *Why do I need to calibrate specifically for ML Thermal Strain? Are there any special considerations for ML when performing a calibration?*

  Each simulation type represents a specific set of underlying assumptions. For example, an Inherent Strain simulation with an Isotropic strain definition (referred to as the Assumed Strain simulation type in Additive Print) is the most simplifying assumption about strain behavior within a part during a build. More inputs are needed as the underlying assumptions become more complex for various simulation types. A calibration process determines the factor(s) that work with any given set of assumptions to get the best fit to your machine/material/part scenario. Therefore we recommend you perform a calibration for each simulation type you will be performing. See Additive Calibration - Full Procedure in the *Additive Print and Science Calibration Guide*. While the ML model was based on the Thermal Strain simulation type in Additive Print, we do not guarantee that the ML model, as trained, will always remain in sync with the Thermal Strain simulation type, as occasional tunings of the Thermal Solver are performed.

  There are no special considerations for ML when building a calibration part and taking measurements, but be sure to choose ML Thermal Strain in Mechanical when running the simulation portion of the calibration procedure.

  Just as we state for all simulation types, it is not necessary to perform a calibration for ML Thermal Strain if your goal is simply to examine *trends*, that is, the effects of variable changes on stress or distortion *relative to each other*.

6.7. Performing a Directed Energy Deposition (DED) Process Simulation (Simplified Approach)

Directed Energy Deposition (DED) is an additive manufacturing process where metal wire or powder is combined with an energy source to deposit material onto a part directly. Rather than spreading a layer of metal powder across a surface and scanning the part's profile with a laser to build up a part surrounded by unmelted powder, only the desired part is built in a DED process, with no surrounding leftover powder. The DED process is much less expensive than LPBF but less precise. It is widely used to repair, or add extra material to, existing parts.

Important:

Ansys provides two methods of DED process simulation. The first is a simplified approach in which super layers representing several actual welded layers are lumped together using the birth-and-death technique just as in an LPBF simulation. Most of the steps are the same, with special considerations as described in this section.

The second approach is a more detailed and accurate representation in which the path of each deposition weld track is "followed" using element *clusters* that are made alive using the birth-and-death technique. This approach uses the DED Process Add-on. Refer to the dedicated user's guide for details.

Simulation of a DED process using the simplified approach is almost the same as for an LPBF process, except that the convection of heat from the part as it is being built must be accounted for differently. Specifically, convection to the surrounding gas in the chamber is applied to all sides of the build as it is building. (This differs from an LPBF simulation, in which the convection is applied only to the top of a newly deposited layer.) A DED simulation uses only the gas convection properties of the build settings and not the powder convection properties. All the required adjustments are handled automatically by the program once you specify a DED simulation using the **AMTYPE** command.

At any point prior to solution, specify a DED simulation with a Commands object in *both the transient thermal and static structural analyses*:

1. Click the **Transient Thermal** object and select **Commands** from the context menu. Or, right-click and select **Insert** > **Commands**. In the Commands window that is displayed, copy and paste, or type, the following:

   ```
   AMTYPE,DED
   ```

2. Click the **Static Structural** object and select **Commands** from the context menu. Or, right-click and select **Insert** > **Commands**. In the Commands window that is displayed, copy and paste, or type, the following:

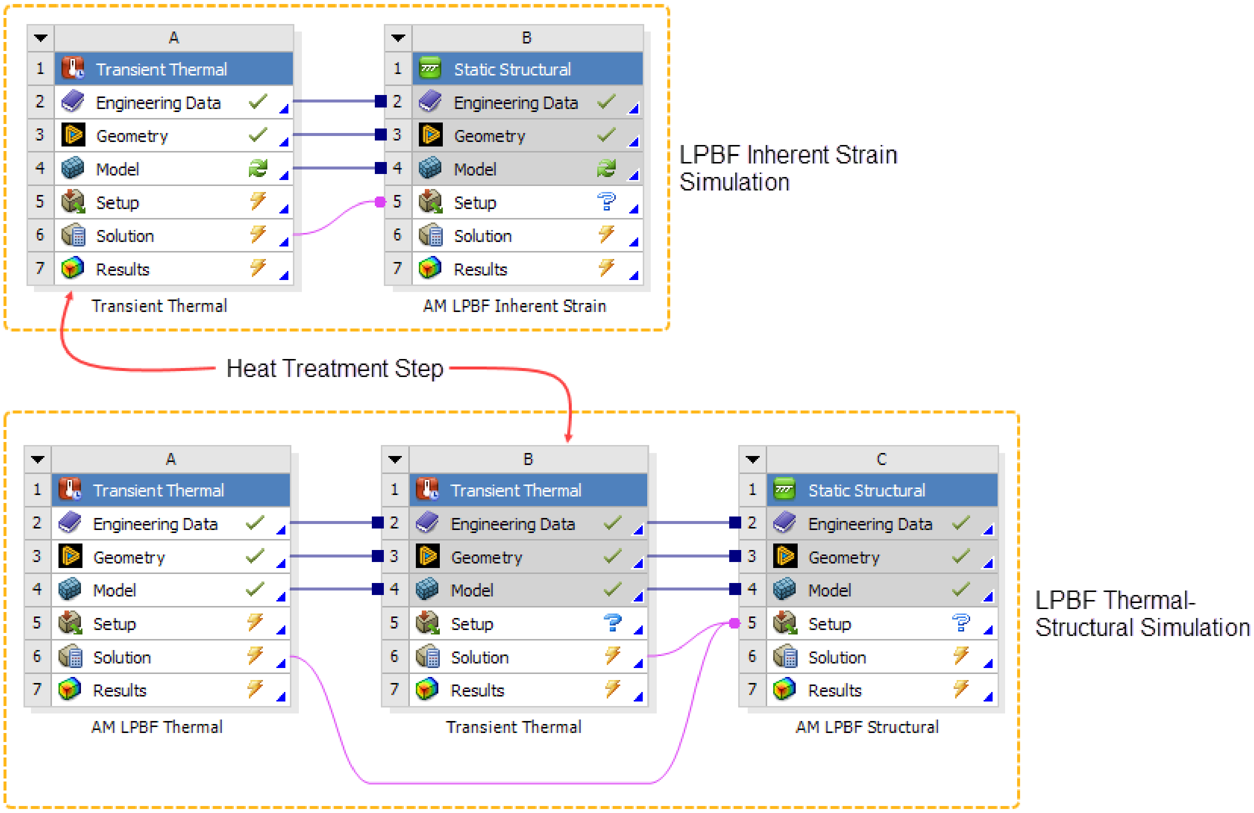
   ```
   AMTYPE,DED
   ```

All other procedures for a DED simulation are the same as in an LPBF simulation.

6.8. Simulating Heat Treatment after the Build

After a part is built, it is common to heat treat it to relieve residual stresses. Heat treating is a process using the controlled application of heat to alter the physical and chemical properties of a material. In an annealing process, metal is heated in a furnace to a particular high temperature, held there for a long time (hours or days) and then allowed to cool slowly. This may be done either before or after removing the part from the base plate. These scenarios can be simulated in Mechanical.

To model the heat treatment process, an additional Transient Thermal analysis is required. The data flow at the system level in Workbench is depicted in the figure below for both Inherent Strain and Thermal-Structural simulations. The workflow systems depict *how data flows* from one type of analysis to another. For example, temperatures from the two (uncoupled) thermal analyses are required as inputs for the structural analysis in the Thermal-Structural simulation. The proper time sequence—*or when that data is used*—is defined in the AM Process Sequence worksheet (p. 100).



You can account for the stress relief mechanism in a heat treatment analysis in one of two ways:

- By using a creep model in Workbench's Engineering Data. This approach results in a more realistic stress relaxation curve as shown in an equivalent stress plot.

Note:

The creep models for Ansys predefined AM materials are not applicable for DED or sintering simulations.

- By specifying a Relaxation Temperature in the Static Structural analysis settings. This is a simplified approach resulting in an abrupt stress relaxation.

The following topics related to heat treatment are available:

6.8.1. Overview of Heat Treatment Workflow

The LPBF Setup Wizard includes heat treatment modeling as an option. We strongly advise you to use the wizard to set up your simulation because it automates much of the tedious set-up, including enabling the creep model, applying boundary conditions, and adjusting the unit system during solution when you choose to simulate with creep effects. *If you do not use the wizard*, follow these steps to simulate heat treatment after the build:

Heat Treatment Workflow at a Glance (Non-Wizard Workflow)	
Simulation Step	**Considerations for Simulating Heat Treatment**
1. Create analysis system	Set up an LPBF simulation, either Inherent Strain or Thermal-Structural.
2. Define Engineering Data (p. 179)	If simulating heat treatment using a creep model as the mechanism for stress relief, identify it here in Engineering Data. For Ansys predefined AM materials, open Engineering Data, select the desired material, and unsuppress the creep model (either Norton or Generalized Garofalo). Take note of the unit system used to define the creep model.
3. Attach geometry and launch Mechanical	No special considerations.
4. Add Transient Thermal system for heat treatment step (p. 181)	In Mechanical, add a Transient Thermal system to the project for the heat treatment step. Be sure AM Process Simulation is set to No for the new system. Insert an Imported Load into the Static Structural system.
5. Identify geometry	No special considerations.
6. Assign materials	No special considerations.
7. Apply mesh controls and generate mesh	No special considerations.
8. Identify and/or generate supports	No special considerations.
9. Define connections	No special considerations.
10. Define AM process steps (p. 182)	Using the Sequencer, add a base unbolting step first, followed by a heat treatment step.
11. Define build settings	No special considerations.
12. Establish analysis settings (p. 183)	If simulating heat treatment using a relaxation temperature as the mechanism for stress relief, set it here in Static Structural > Analysis Settings.
13. Apply boundary conditions (p. 183)	In addition to the usual boundary conditions for an LPBF simulation, apply convection to the surfaces of the part, supports, and base plate for the heat treatment step. For the Imported Load, specify the Transfer Step to be Heat Treatment Step.
14. Solve the simulation (p. 184)	If simulating with creep properties, switch to the unit system used in the creep model before solving.
15. Review results	Add an Equivalent Stress result scoped to the build to see the stress relief. Add an Equivalent Creep Strain result to see the creep accumulation.

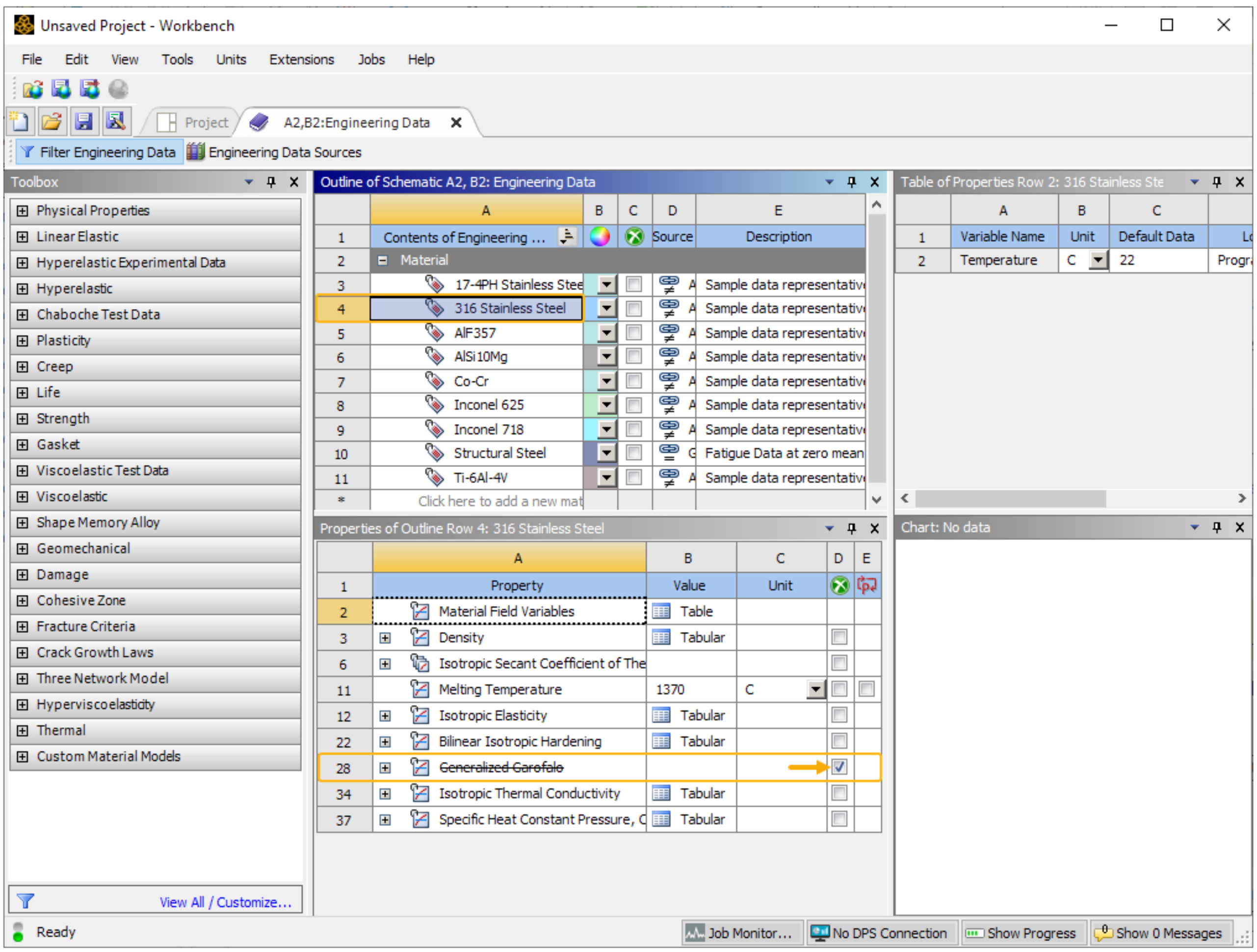
6.8.2. Define Engineering Data - Unsuppress Creep Properties

When simulating with heat treatment, use either creep properties or a relaxation temperature (p. 183) as the mechanism for stress relief, but not both.

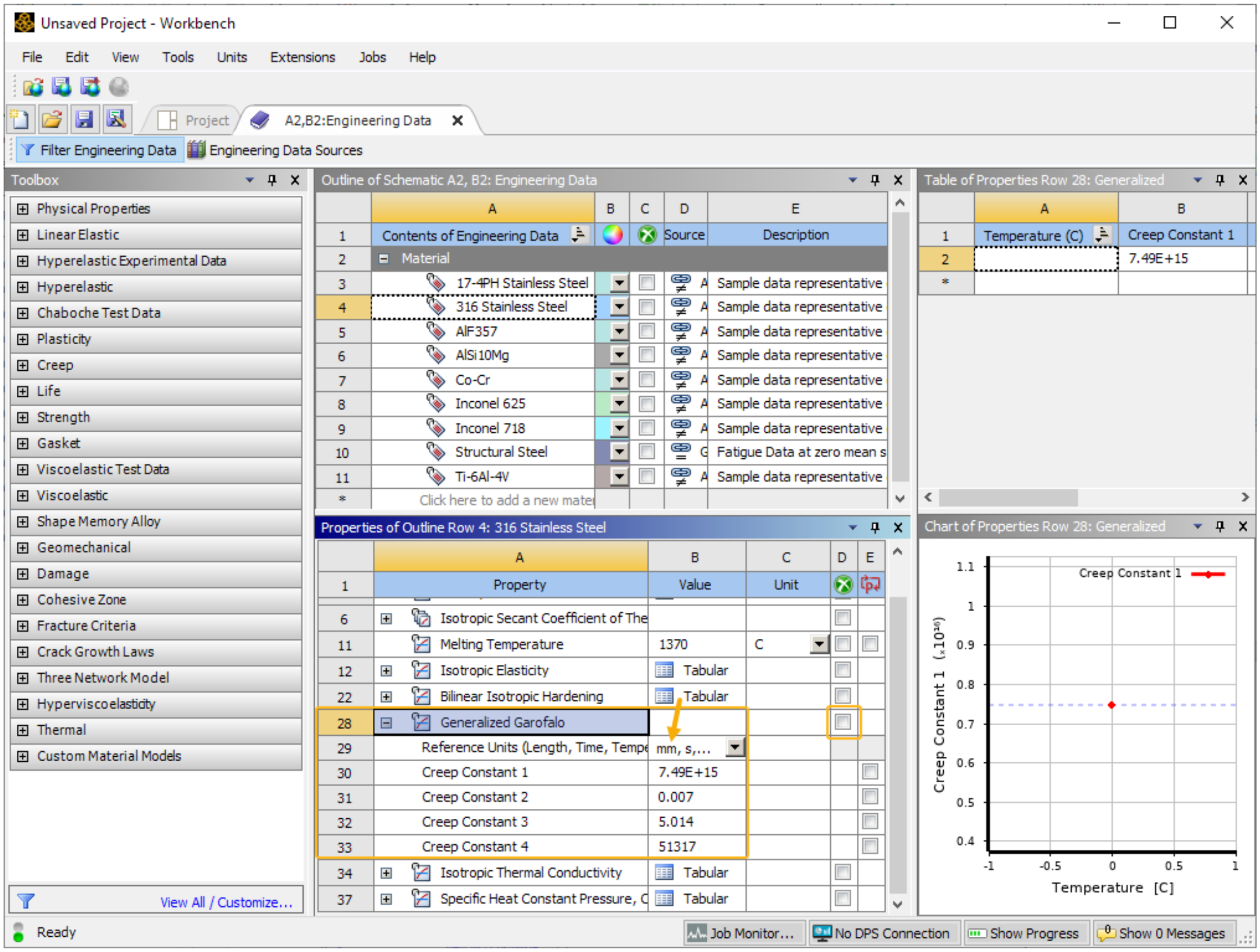
To use creep properties as the mechanism for stress relief, unsuppress the creep model in Engineering Data. (By default, creep properties are suppressed, meaning creep effects are ignored.) Unsuppress the creep model as follows:

1. In Workbench, double-click the **Engineering Data** cell in the first (left-most) system. This opens the Engineering Data tab and workspace

2. Click your choice of materials from the list of available AM materials shown. In this example, we've chosen 316 Stainless Steel. Notice the Source column (column D) shows Additive Manufacturing Materials.



3. **Clear the check box** to the right of the creep model so that the creep properties are unsuppressed. In this example, the model is Generalized Garofalo. The model may also be Norton, depending on your chosen material. **Expand the model cell** to view the creep constants and reference units. In this example, the reference unit system is in millimeters.



Important:

Take note of the reference unit system used to define the creep model. It may be defined in meters or millimeters. *The unit system used during solution must be the same unit system used to define the creep model.* You may set up your simulation and review results in any unit system but be sure to switch to the creep model system just before you initiate the solve. Should you forget to do this, you will see the following error message and the simulation will not solve. *"An error occurred inside the SOLVER module: The "Reference Units" defined for the Creep material property does not match the solver unit system."* If you see this error, clear generated data, change to the creep unit system, and solve again.

4. Click the **Project** tab at the top of the Workbench window to go back to the system view.

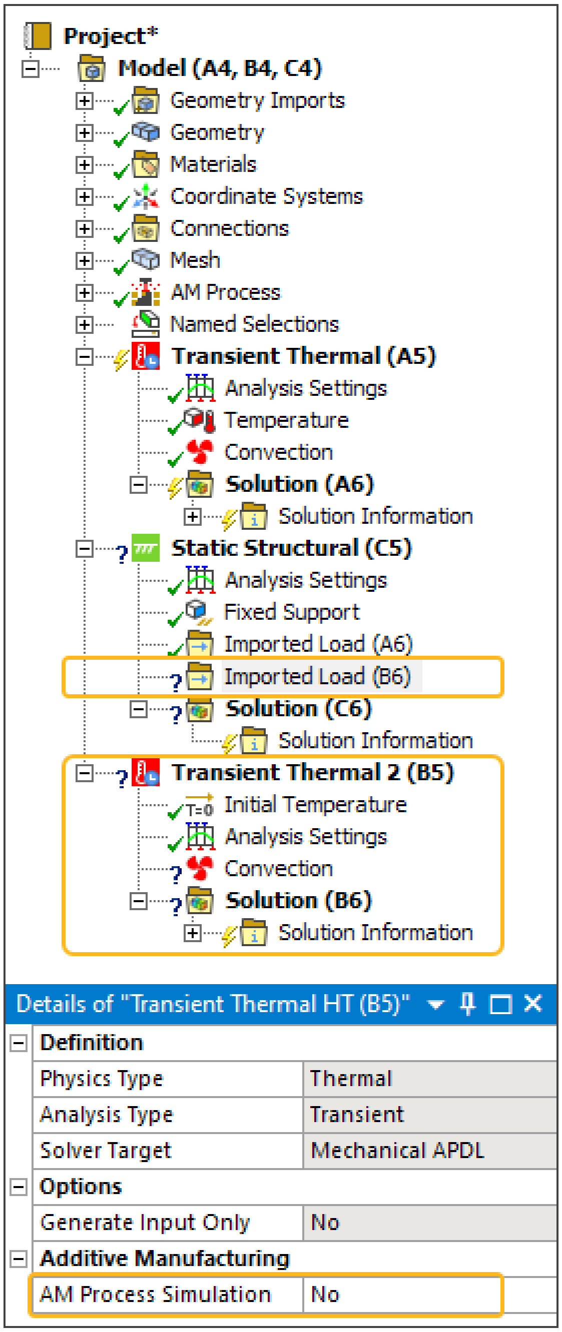
What if I forget to unsuppress the creep model?

If you forgot to unsuppress the creep model in Engineering Data and you have already assigned material properties to the bodies in Mechanical, you must update the project after unsuppressing the creep model. Unsuppress the creep model as described above, and then in Workbench system view, right-click the Model cell in the first Transient Thermal analysis and select **Update** before returning to Mechanical.

6.8.3. Add Transient Thermal System to the Project

Add the Transient Thermal system to account for heat treatment as follows:

1. In Mechanical, from the Home tab, click the **Analysis** button to reveal a drop-down list of analysis types. Select the **Transient Thermal** system.
2. Click the new **Transient Thermal** object in the project tree and in the Details view, under Additive Manufacturing, change the **AM Process Simulation** setting to **No**. (This option may already be set to No, depending on the LPBF system you set up.)
3. Right-click the **Static Structural** object and select **Import Load** > **Transient Thermal**. This adds a new Imported Load object under the structural system. You will need to go back to this object later after the AM process steps are defined.



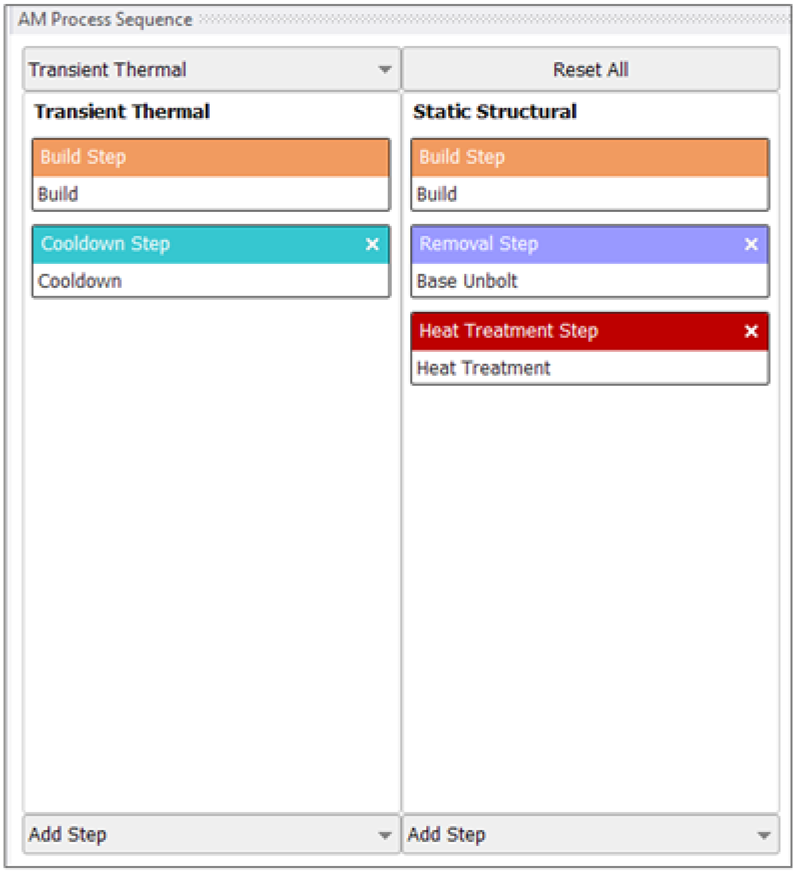
6.8.4. Define AM Process Steps - Add Base Unbolting and Heat Treatment Steps to the Sequencer

When modeling heat treatment, it is best to also include a base unbolting step, in which the constraints for UX, UY, and UZ are removed from the base plate nodes. This eliminates any artificial tension in the base caused by fixed boundary conditions and thermal expansion during heating, which would otherwise occur if base unbolting is not used.

Add base unbolting and heat treatment steps to the AM Process Sequence worksheet as follows:

1. Select the **AM Process** object and then click the **Sequence** button (AM Process group) on the context toolbar. The sequencer worksheet comes up.

2. Click **Add Step** on the Static Structural side of the sequencer worksheet and select **Base Unbolt** to add the unbolting step first. Click **Add Step** again and select **Heat Treatment Step**.



3. Select the **Static Structural** object in the project tree and then select the **Imported Load** object underneath it that belongs to the new heat treatment Transient Thermal system. In the Details view, under Additive Manufacturing, change the **Transfer Step** option to the new **Heat Treatment Step**. This ensures the thermal data from the heat treatment step is transferred at the correct time (load step).

6.8.5. Establish Analysis Settings - Define Creep Relaxation Temperature

When simulating with heat treatment, use either creep properties (p. 179) or a relaxation temperature as the mechanism for stress relief, but not both.

To use a relaxation temperature as the mechanism for stress relief, click **Analysis Settings** under the Static Structural object. In the Details view, under Additive Manufacturing Controls, change the step to heat treatment step, then change **Relaxation Temperature** to **User Specified** and then change the **Value** to your desired stress relaxation temperature for the material.

6.8.6. Apply Boundary Conditions - Add Convection for Heat Treatment Step

In addition to the typical boundary conditions for an LPBF simulation, apply convection to the surfaces of the part, supports, and base plate for the heat treatment step, as follows:

1. Right-click the **Transient Thermal** object corresponding to the heat treatment step and then select **Insert** > **Convection**.

2. Select all the bodies including the base plate (geometry selection) and hit **Apply** in the Details view. *Are STL Supports automatically selected and included in body count?*
3. In Details under Definition, **Film Coefficient**, enter a value for the convection coefficient of the gas in the furnace, or select **Tabular Data**.
4. Under **Ambient Temperature**, select **Tabular Data**.
5. In the **Tabular Data** area of the interface, specify the appropriate convection coefficient and temperature at different time points.
6. In **Analysis Settings**, change the **Step End Time** to match the final time in the convection tabular data. More details on how to control the time stepping settings are available here.

6.8.7. Solve the Simulation - Check Units First!

If simulating with creep properties (p. 179), switch to the unit system used in the creep model before solving. Should you forget to do this, you will see the following error message and the simulation will not solve. *"An error occurred inside the SOLVER module: The "Reference Units" defined for the Creep material property does not match the solver unit system."* If you see this error, clear generated data, change to the creep unit system, and solve again.

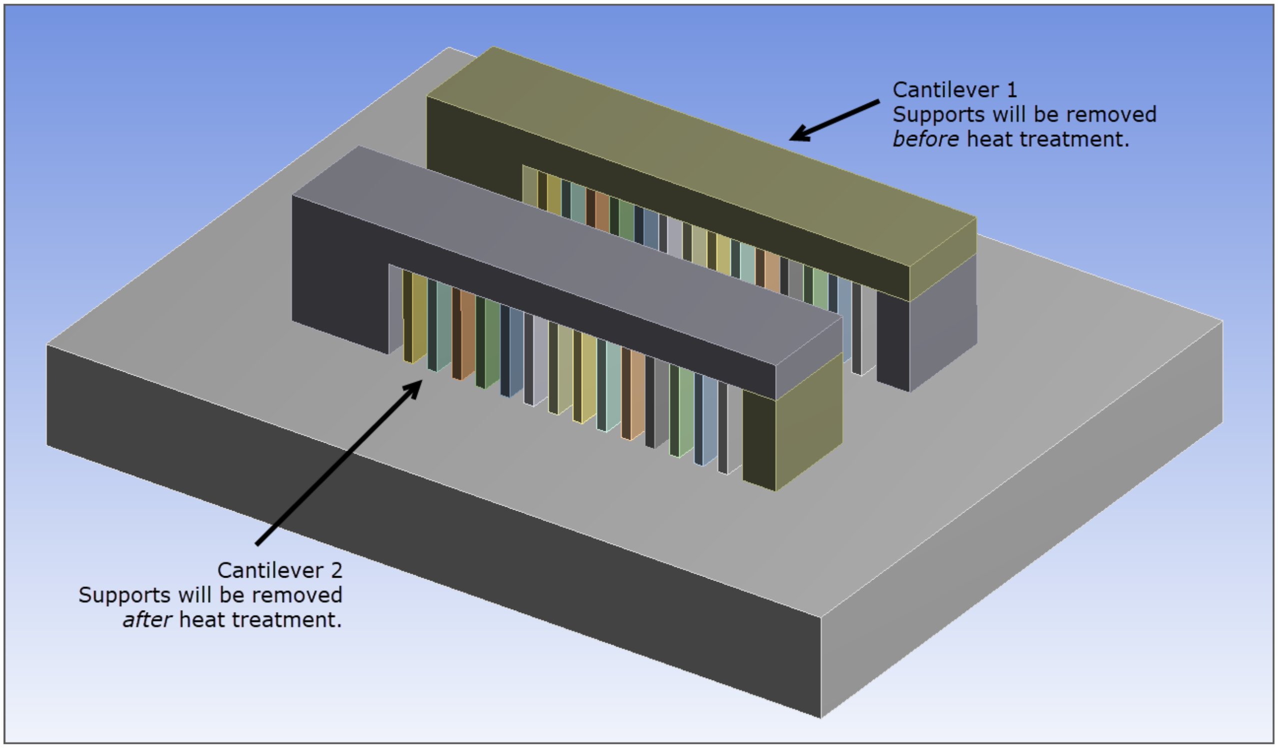
Note that you can switch back to your preferred unit system for reviewing results after the solution is finished.

6.8.8. Heat Treatment Examples

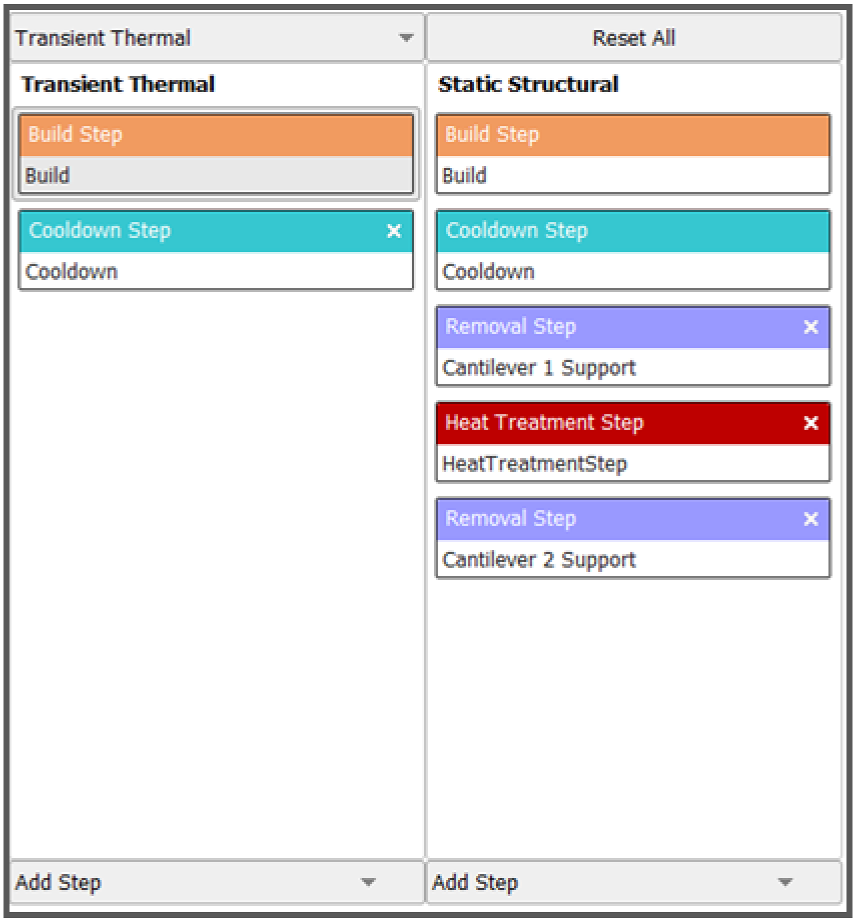
The following examples are available:

6.8.8.1. Example LPBF process simulation with and without heat treatment

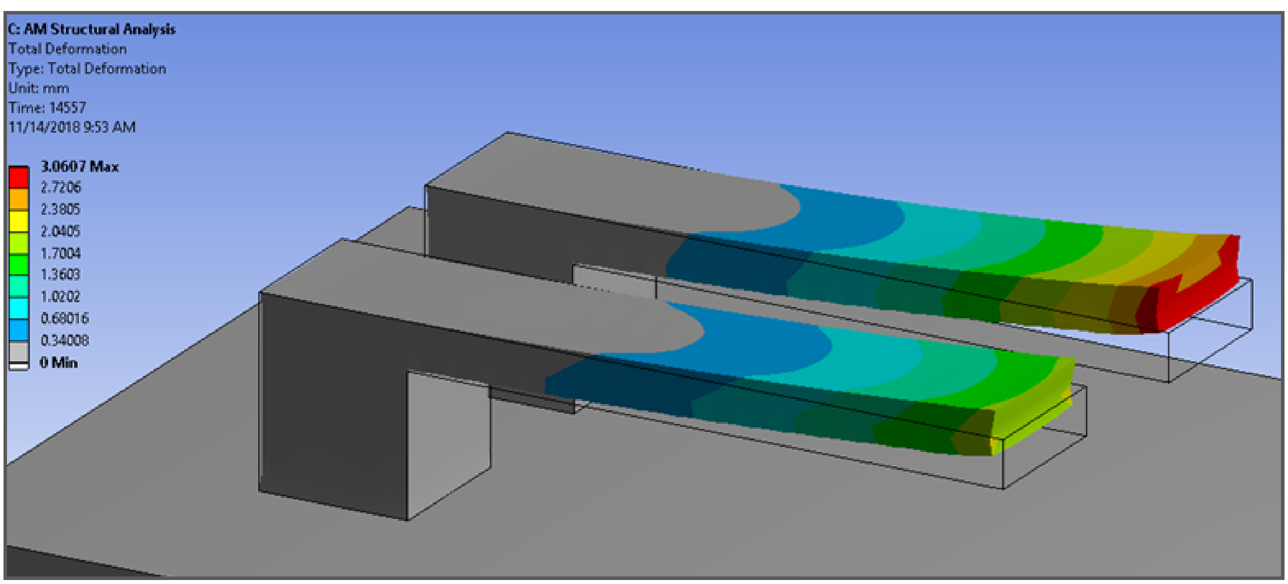
The following heat treatment project was set up to show the effects that heat treatment can have on stress and distortion in an additively manufactured part. Two cantilevers with supports are set up in the same additive manufacturing simulation with cutoff from the supports occurring at different time points for each cantilever. The first cantilever is removed from supports before heat treatment, and the second cantilever is removed after heat treatment.



To model this, the AM sequencer was set up as shown in the following figure.



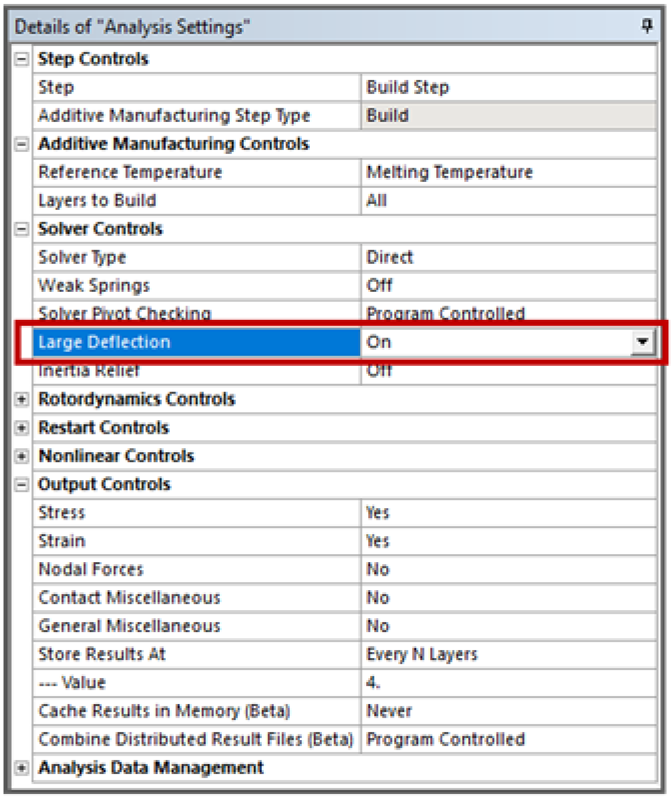
After simulating the additive manufacturing process and steps defined in the sequencer, the two cantilevers are left with different distortion magnitudes. The heat treatment step relieved residual stresses and led to decreased distortion in the second cantilever (on the left) after the cutoff step.



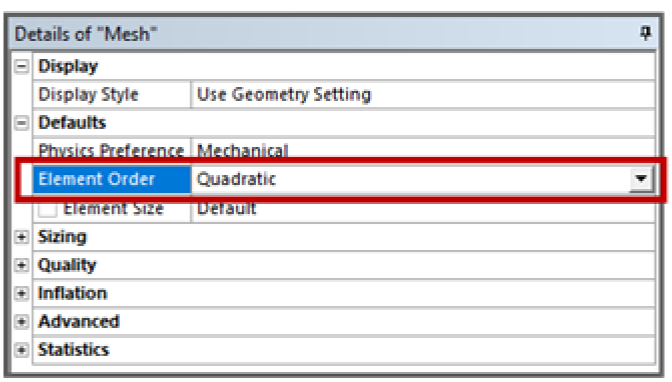
6.9. Capturing a Buckled Shape with Large Deflection

If your part has particularly thin walls and you are worried that it may deform significantly during the additive printing process, you can set up a simulation to capture a buckled shape deformation. The following two settings should be set to capture this phenomena in the AM simulation:

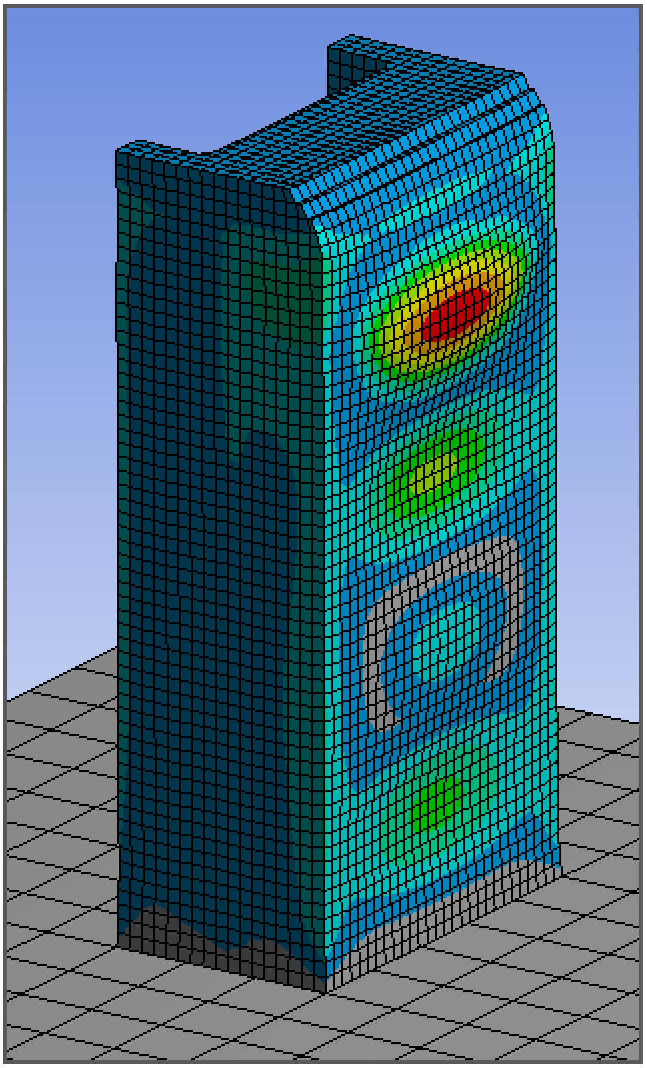
- Turn on Large Deflection: Under Static Structural, click **Analysis Settings**. In the Details view, under Solver Controls, change **Large Deflection** to **On**. This setting is off by default for AM simulations.

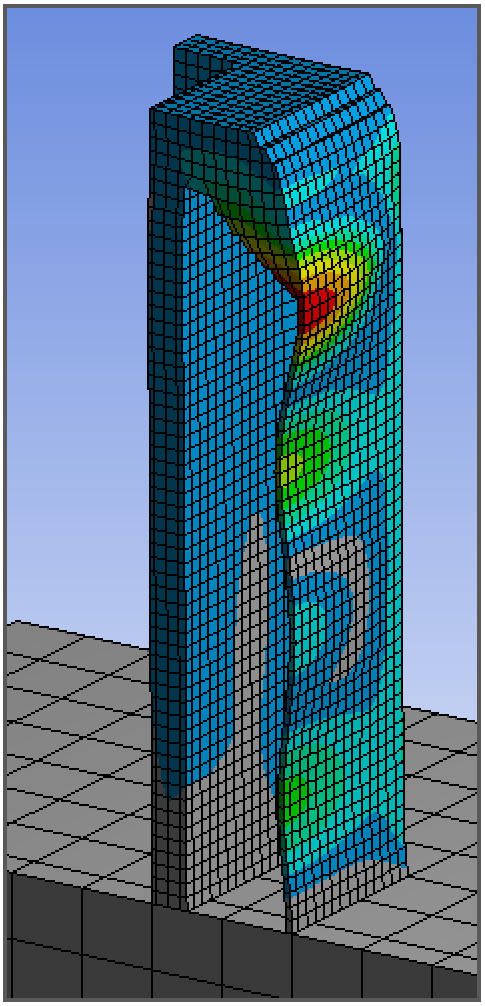


- Use Quadradic elements: Click **Mesh**, and in Details, under Defaults, change **Element Order** to **Quadradic**. Higher-order elements may help convergence in the simulation.



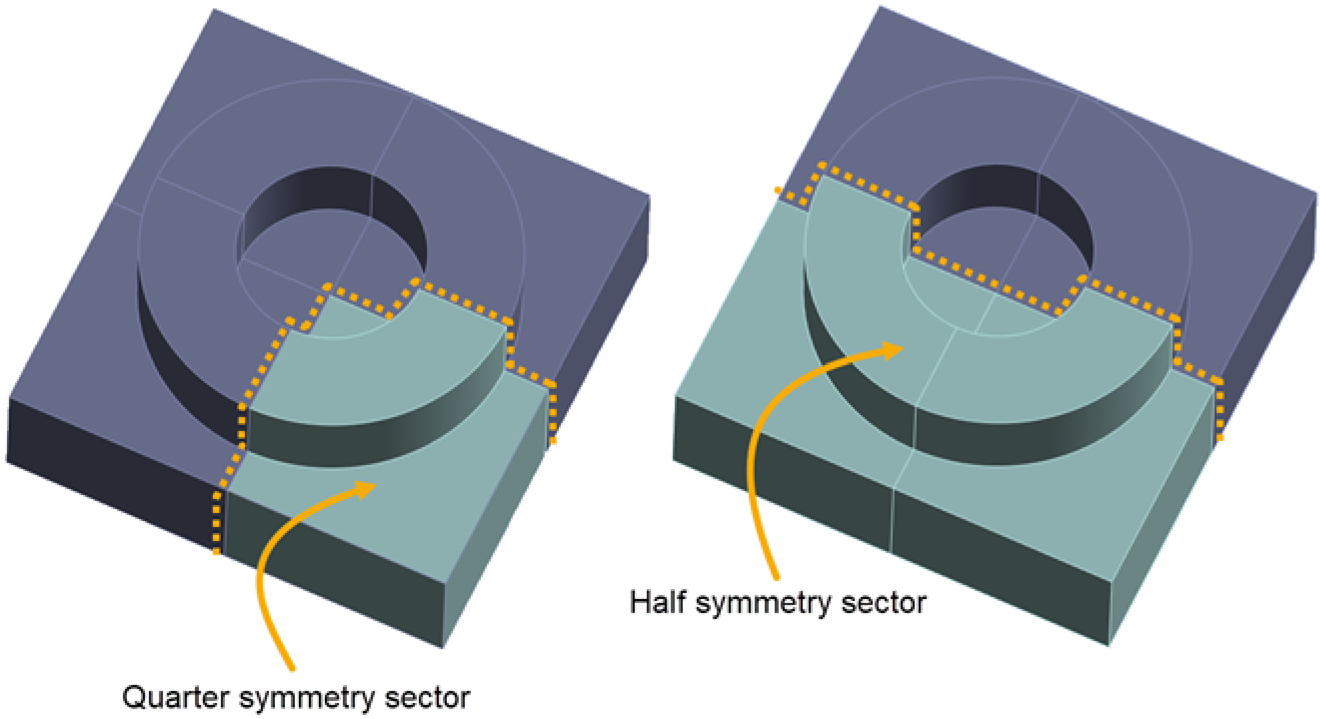
The following figures show distortion results from an AM simulation of a model with a thin outer wall. The full model is shown as well as a cross-section view. The upper portion shows significant distortion into a buckled shape. (Note: this model has a coarser mesh than recommended, it is for demonstration purposes only.)





6.10. Modeling a Symmetrical Part

If your model has symmetrical geometry—specifically, if it is symmetric with respect to the part, supports, and build plate, *and* it has symmetric loading and boundary conditions—you can simulate a sector of a large model to achieve a faster simulation time with fewer resources. Both the base plate and the build (part and supports) should be included in the symmetry sector.



The procedure is the same as described in Symmetry, with a few modifications for the AM process simulation. Specifically, you will need to do the following:

A. Prepare your part, supports, and baseplate geometries to form a symmetric sector.

B. After meshing, create a SYMM_NODES named selection consisting of nodes along the symmetry plane. This named selection is used internally and allows the program to handle the AM boundary conditions properly.

C. As a structural boundary condition, apply a constraint at the symmetry plane of the build and the base plate. Depending on the mesh type used, this is done in one of two ways:

   - For a Cartesian mesh, where the mesh is associated back to the geometry entities, insert a symmetry region on the geometry faces defining the symmetry planes.

   - For a layered tetrahedral or voxel mesh, where the mesh is not associated back to the geometry faces, define Mechanical APDL commands that constrain nodes from moving normal to the symmetry planes.

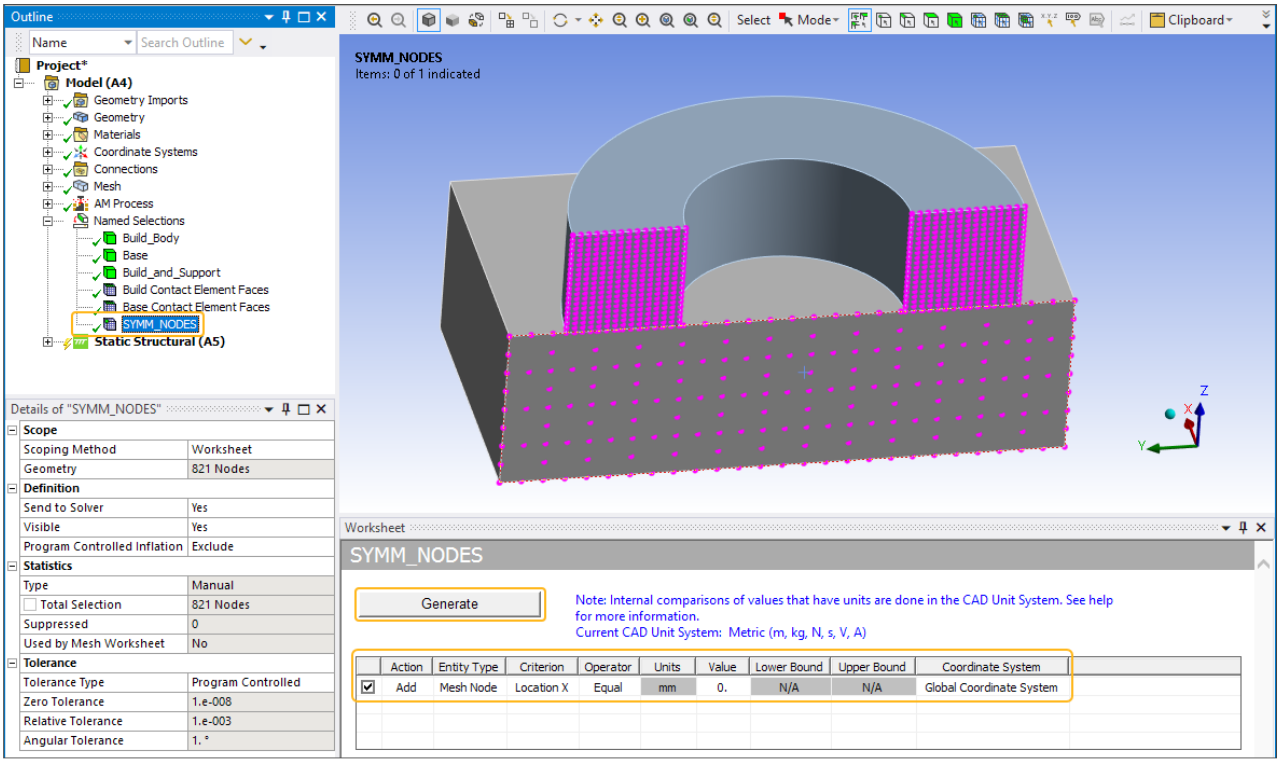
D. Finally, if your simulation includes a thermal solution, input a **Dwell Time Multiple** in Build Settings to account for the reduced simulation time that comes with a symmetry model.

For step A, prepare your model in a CAD package, as normal. Procedures for steps B through D are described next.

Procedural Steps

B. Create a SYMM_NODES named selection consisting of nodes along the symmetry plane

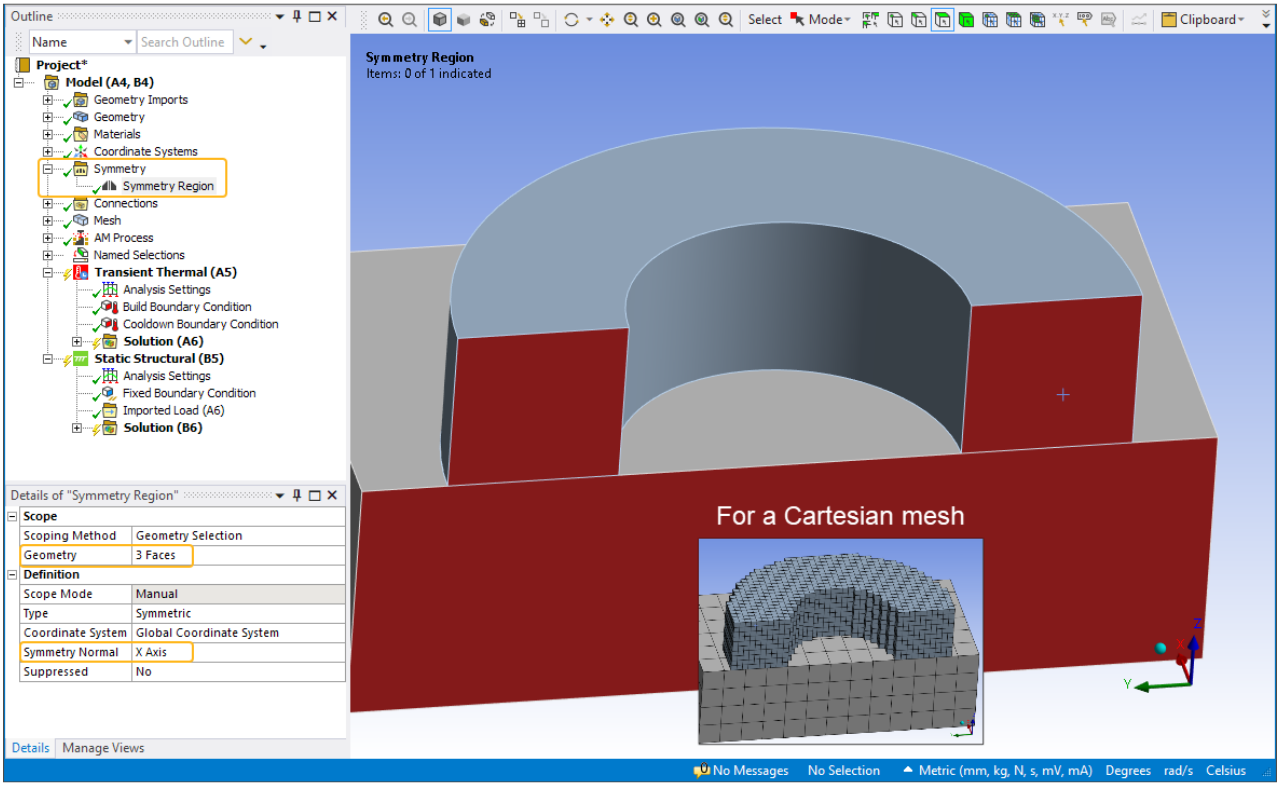
Right-click **Model** in the project tree, and select **Insert** > **Named Selection**. Right-click the newly created **Selection** object (under Named Selections) and **Rename** it to SYMM_NODES. In the Details view change **Scoping Method** to **Worksheet**. In the Worksheet, set up filtering criteria to identify the nodes of interest. In the half symmetry model below, we selected all the nodes along a plane of X = 0. Click **Generate** to create the named selection of nodes.



C. Apply a constraint at the symmetry plane

- For a Cartesian mesh, insert a symmetry region on the faces defining the symmetry plane of the build and the base plate. This will apply a frictionless support boundary condition on those faces. With this boundary condition, no portion of the body can move, rotate, or deform normal to the faces. (The use of a frictionless support is not unique to additive manufacturing simulations.)

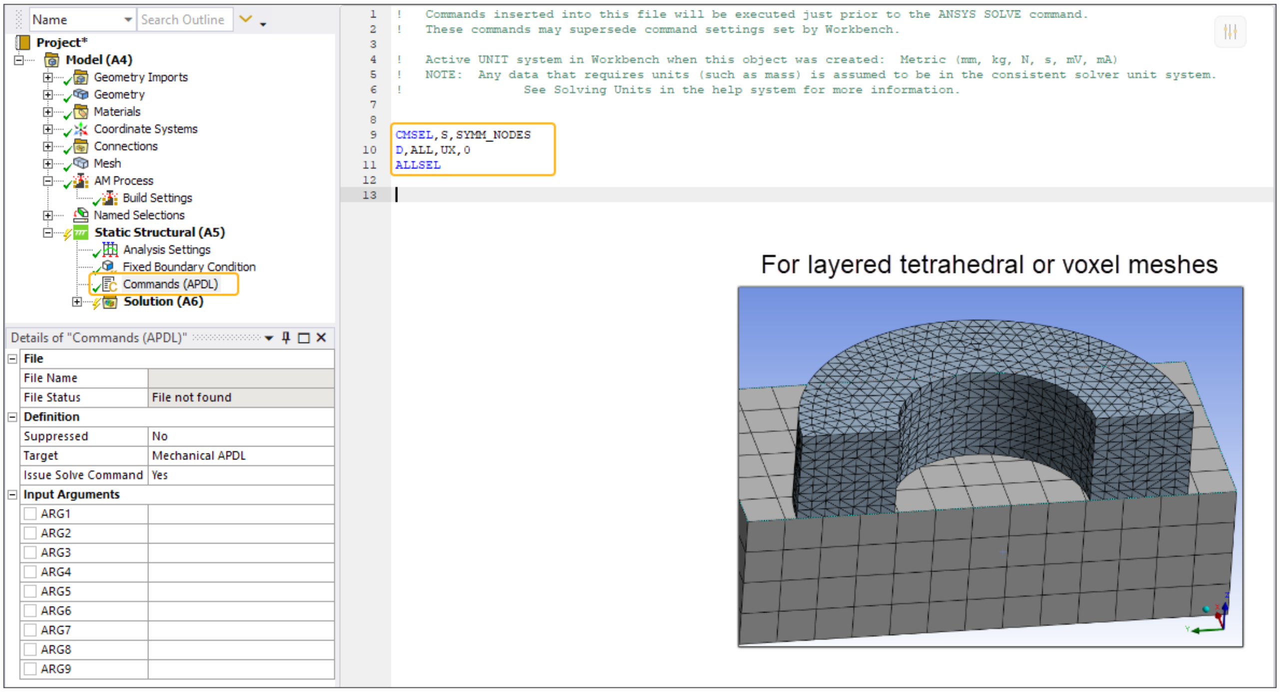
  Switch to face mode of selection and use Ctrl-left-click to select the multiple geometry faces along the symmetry plane. Highlight **Model**, right-click and **Insert** > **Symmetry**. Right-click the **Symmetry** object and select **Insert** > **Symmetry Region**. In the Details view, be sure that the **Symmetry Normal** property is set to the axis of the Global Coordinate System that is *normal* to the symmetry plane, in our example, the X-Axis.



- For a layered tetrahedral or voxel mesh, since the mesh is not associated back to geometry faces, planes of symmetry selected in the UI are not recognized. For these types of meshes, you must define Mechanical APDL commands that constrain nodes in the symmetry planes. (Note that a voxel mesh associates back to *body* entities only, not faces, edges, or vertexes.)

  Right-click **Static Structural** and select **Insert** > **Commands**. On the Commands object that appears in the tree, enter the appropriate commands that select the SYMM_NODES nodes, applies a constraint (**D** command) to prevent motion normal to the symmetry plane, and selects all nodes again. In our half symmetry example:

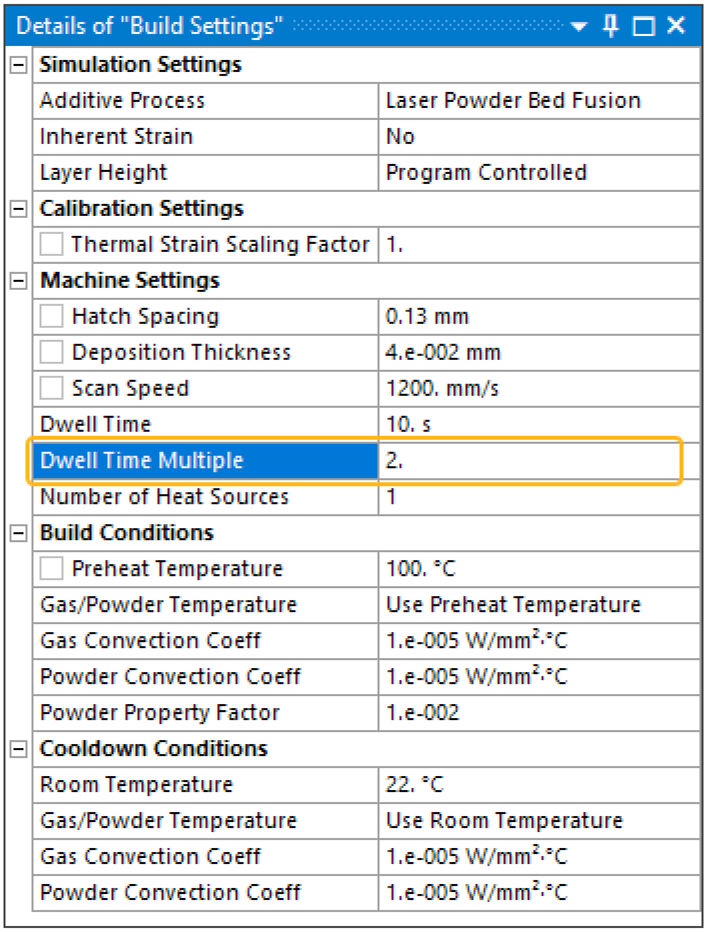
```
CMSEL,S,SYMM_NODES
D,ALL,UX,0
ALLSEL
```



D. For thermal simulations, input a Dwell Time Multiple equal to the number of symmetry sector repeats

For analyses with thermal solutions, such as an AM LPBF Thermal-Structural simulation, this step is required to reconcile the reduced time to simulate the build. For example, the build time for a half symmetry model is ½ of the build time of the full model, so the Dwell Time Multiple should be 2. Similarly, use a Dwell Time Multiple of 4 for a quarter symmetry model, and so on.

In the project tree, click **AM Process** > **Build Settings** and in the Details view under Machine Settings, change **Dwell Time Multiple** to 2, or 4, or the appropriate number of repeats. In the half symmetry model in our example, we changed Dwell Time Multiple to 2.



The remaining steps are the same as usual for additive simulations.

6.11. Modeling Powder with Elements

During a laser powder bed fusion (LPBF) process, almost all the heat dissipation is through the part and supports back to the build plate rather than out through the unmelted powder surrounding the part. Many users ignore the small effect of heat loss through powder but you may choose to model it in one of two ways:

- As equivalent heat convection
- By including powder elements as a named selection called POWDER_ELEMENTS. (For Mechanical APDL users, use an element component CM,*POWDER_ELEMENTS*,ELEM).

Modeling the powder as elements may be useful if you will be simulating multiple parts close together on the build plate or if the part has features close together, where accounting for the heat transfer occurring between the parts or features is important. The following is an overview of the procedure:

1. In your CAD program, create a separate body to represent the powder in-between the parts or features. If you will be using a Cartesian mesh, share topology between the bodies. If you will be using a layered Tetrahedrons mesh, do not share topology.

2. In the Mechanical application, identify the build and base bodies on the AM Process object as usual. Do *not* select the powder body when identifying the build and base bodies.

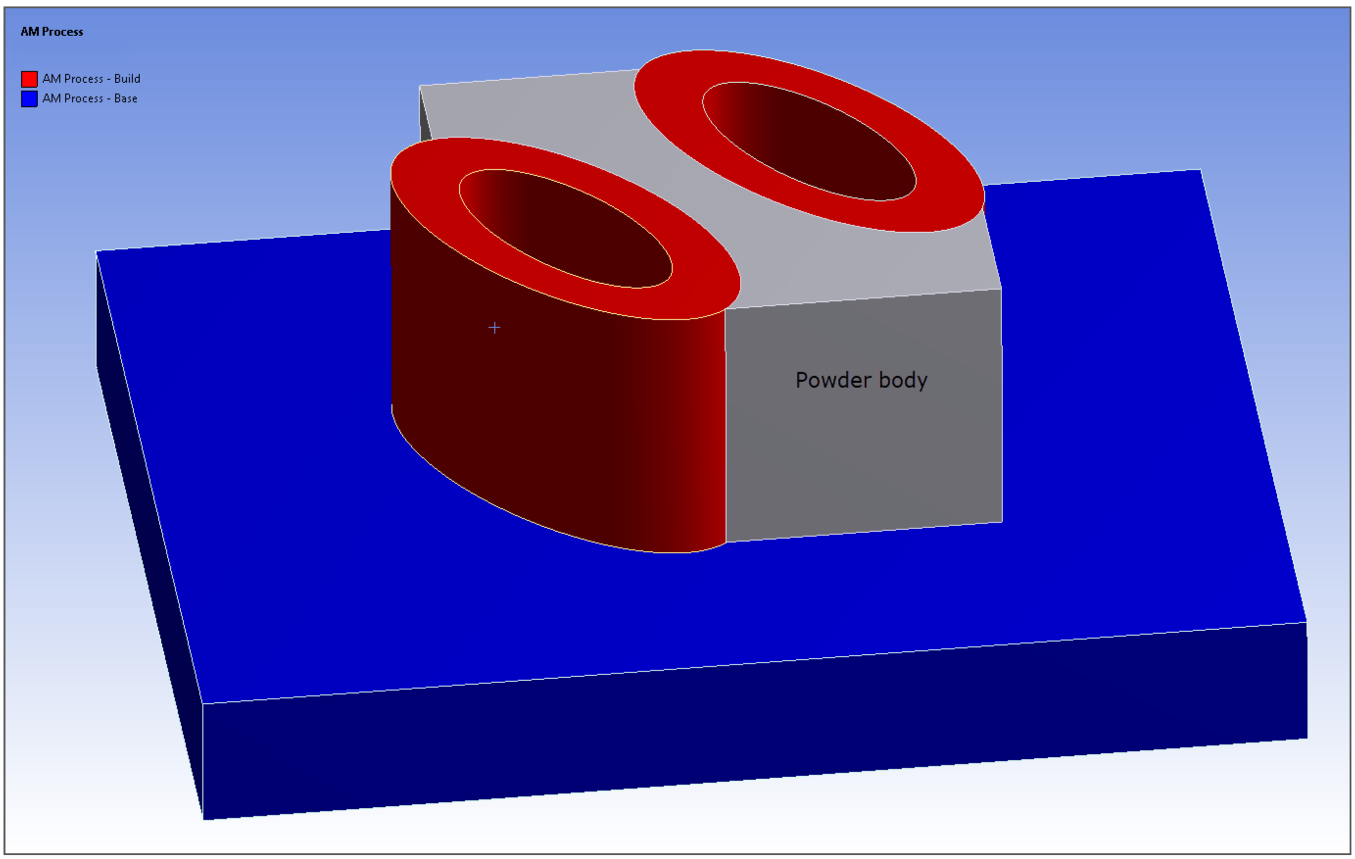
3. Assign the same material property that you use for the build to the powder body.

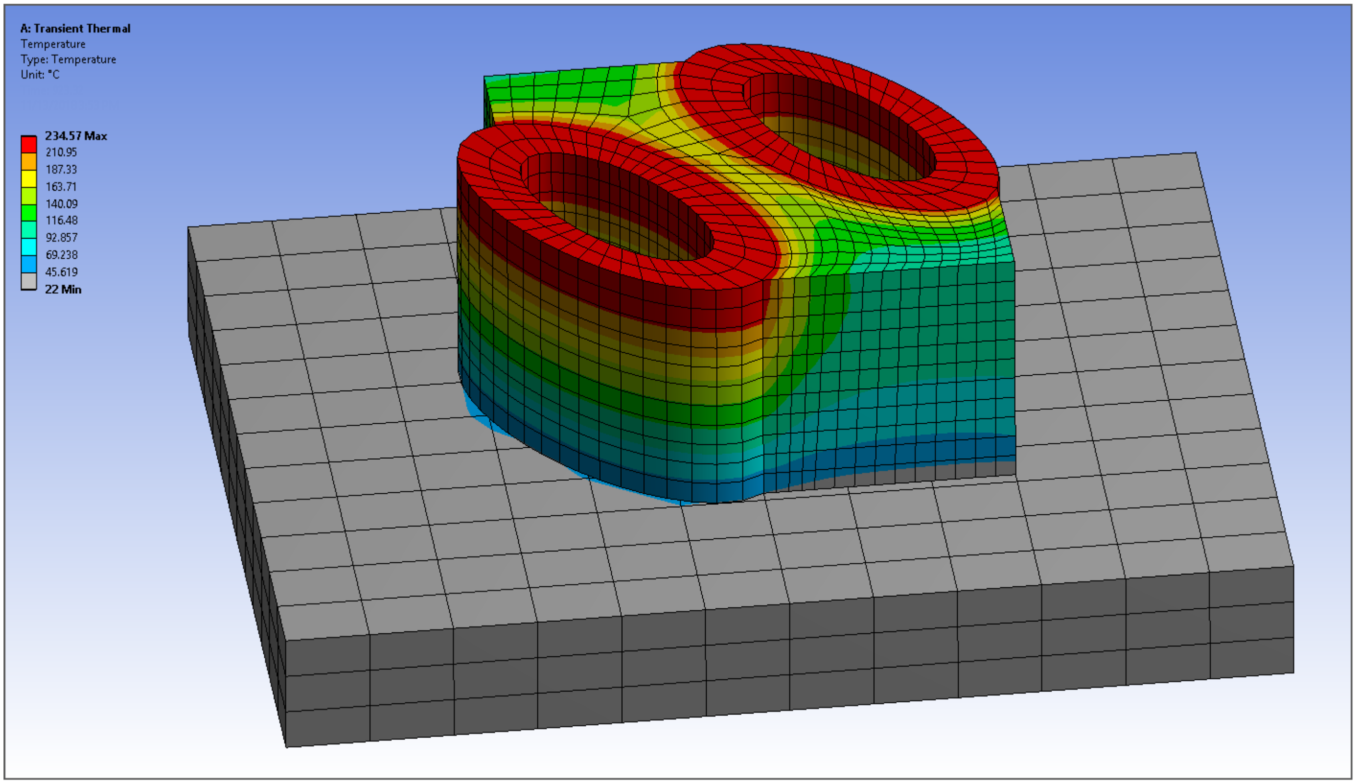
4. Depending on your chosen mesh type, account for the connections between bodies and mesh the model.

5. Create a named selection, called POWDER_ELEMENTS, of elements associated with the powder body. This Ansys-defined named selection will use the knockdown factor, Powder Property Factor, identified in the Build Conditions to estimate the powder properties.

6. Proceed with the rest of the simulation as normal.

In the following example, the powder was modeled between two cylindrical parts close together on the build plate. Temperature contours are shown in the second figure. Notice the heat transfer in the powder area between the parts.





6.12. Modeling Clamps, Measuring Devices and Other Non-Build Components

At times you may want to simulate geometry items that are present on the build plate but that are not being 3D-printed. These items may include clamps, bolts, measuring devices, instrumentation, etc. They may influence the heat dissipation and/or distortion of the part being built so they need to be included in the simulation.

Use the Ansys-defined named selection NONBUILD_ELEMENTS to identify this geometry. (For Mechanical APDL users, use an element component CM,*NONBUILD_ELEMENTS*,ELEM). This Ansys-defined component name signals to the application that those elements should not be part of the build process. That is, they will be present the entire simulation and will not be "birthed" at melting temperature in a layer-by-layer fashion.

6.13. Troubleshooting

If you are experiencing convergence issues during an AM Process Simulation, try one or more of these suggestions, as appropriate:

- If base unbolting and/or removal steps (p. 97) are included in your simulation, the fact that nonlinear stabilization (p. 102) is on by default should be enough to loosely hold parts in place to prevent rigid body motion when the base is removed. However, if you think you need additional constraints, consider using weak springs (first choice) or directly applying fixed boundary conditions on three nodes of the part (second choice) to prevent rigid body motion. These steps are described in Additional Constraint Options for Removal Steps (p. 120).

- If Large Deflection is on, turn it off, under Analysis Settings for the static structural analysis. Large deflection is off by default for AM Process Simulations but it is useful in some cases and users often turn it on.

  For thin-walled parts, the walls can tend to take on a buckled or dimpled shape as the build progresses. In these cases, large deflection is required and convergence difficulties may be seen. If that is the case, you will need an adequate mesh to capture this deformation. Consider using a finer mesh or switch to a quadratic (midside-noded) element (Element Order in Mesh Details). See Capturing a Buckled Shape with Large Deflection (p. 187) for details.

- Switch to the Direct Solver for Solver Type under Analysis Settings for the static structural analysis.

- If you are using customized materials, be sure you have defined a nominal strength at the melt temperature, as a near zero modulus and/or yield strength could lead to convergence problems. (Ansys predefined materials take care of this internally.)

- If you are using a Layered Tetrahedrons mesh with a geometry imported from an `.stl` file, be sure the faceted geometry is cleaned up as much as possible. (See Cleaning Up Facets (p. 13)). Depending on the quality of the `.stl` file, you may see convergence issues because of poor tet elements created with a "dirty" `.stl` file.

- If you are using creep properties in a heat treatment analysis and you experience convergence issues in the transient thermal solution, increase the sub-steps in the Transient Thermal module in order to avoid an abrupt change in temperature.

If you are using an AM Bond connection and see unrealistic spikes in deformation at the interface where the AM Bond is applied, adding a second AM Bond connection with flipped scoping may improve the connection and prevent the unrealistic deformations.

Chapter 7: Performing a Calibration

Deformation values within additively manufactured parts vary across different machine and material combinations in real-world fabrication scenarios, especially when considering the numerous laser powder bed fusion (LPBF) machine manufacturers and powder material suppliers available. Before simulating your production part, you should calibrate your simulation software to help reduce variability. Calibration accounts for the difference between measured and simulated deformation.

The overall calibration process for Additive in Mechanical consists of obtaining measured distortion values from physical experiments and then performing a calibration simulation. The first part of the process, obtaining measured distortion from a calibration build, is exactly the same process as described in the Additive Print and Science Calibration Guide. Refer to the following topics from that document for details:

- Determine Your Calibration Part
- Appendix B: Download Calibration Files Here
- Build the Calibration Parts
- Take Distortion Measurements

The objective of this calibration procedure is to determine a set of calibration coefficients to use when performing additive simulations in Ansys Mechanical. The coefficients to be determined depend on the simulation type and strain definition selected. This is summarized in the following table. SSF is Strain Scaling Factor and ASC is Anisotropic Scaling Coefficient. Throughout this document, we will distinguish between an SSF/TSSF-only calibration and an SSF combined with ASCs (or SSF + ASCs) calibration.

Calibration Type	Simulation type / Strain definition	Calibration Coefficients			
SSF only	AM LPBF Inherent Strain / Isotropic	SSF	-	-	-
	AM LPBF Thermal-Structural	Thermal SSF (TSSF)	-	-	-
SSF + ASCs	AM LPBF Inherent Strain / Scan Pattern	SSF	Parallel ASC	Perpendicular ASC	Vertical ASC = 1
	AM LPBF Inherent Strain / Thermal Strain	SSF	Parallel ASC	Perpendicular ASC	Vertical ASC = 1

The values of the calibration coefficient(s) compensate for the difference between an experimentally measured target deformation and a simulated deformation value obtained at the same location on a chosen calibration geometry. Using the calibrated coefficient(s) will greatly improve the simulation prediction accuracy of your production part when using the same simulation setting combinations, therefore increasing the chance of successful builds as well as reducing the cost of trial-and-error experiments.

For more information about calibration coefficients, see the beginning of this section (p. 138).

7.1. When to Calibrate

The values for calibration coefficients depend upon many variables from both fabrication and simulation setup:

- Material
- LPBF machine
- Process parameters (laser power, scan speed, deposition thickness, base plate preheat temperature, hatch spacing, etc.)
- Simulation type and strain definition
- Material properties (linear or nonlinear)
- Mesh type (Cartesian, layered tetrahedrons, or voxel)
- Other simulation specific configurations

Calibrate only for the type of simulation you will be performing on your production part. For example, if you care about the as-built distortion of a part for a given material, machine, and process parameter combination using an Inherent Strain isotropic simulation, run the calibration just for that combination of variables. When the combination changes, re-calibrate the calibration coefficients. Even changing the material supplier for a material you have already calibrated for may require a new calibration.

It is important to note that you can expect different calibration coefficient values from different simulation types, material properties, and/or mesh types even when using the same experimental distortion measurement value(s).

7.2. Calibration Simulation Workflow

The steps in a calibration simulation are summarized here and described in more detail following the table.

Calibration Simulation Workflow at a Glance	
Simulation Step	**Considerations for Calibration**
1. Set up AM simulation	Set up either an LPBF Thermal-Structural or LPBF Inherent Strain simulation. Import your chosen calibration geometry. Use the LPBF Setup Wizard (p. 17) to set up your simulation the same as you would to simulate your production part. For the meshing step, use a 0.5 mm voxel mesh size if you are using either the Cantilever or Double arches calibration geometry from Ansys.

Calibration Simulation Workflow at a Glance	
Simulation Step	**Considerations for Calibration**
	Use defaults for all calibration values. Perform the steps in the wizard until you reach the last step, Set Up Calibration.
2. Set up calibration The last step of the LPBF Setup Wizard is the same as the single-step LPBF Calibration Wizard.	Set **Set up Calibration** to Yes. Identify **Node Measurement Location**, such as a line of nodes along a surface, corresponding to the experimental measurement location. Choose **Deformation Direction**, either X, Y, or Z in which deformation is measured. Usually this is normal to the surface of the measurement location. Set **Calibration Deformation Results**, either Max, Min, or Avg **Calibration Type** (either SSF or SSF + ASCs) is set automatically based on your settings of simulation type and Strain Definition. Set up **Start Layer Angle** and **Layer Rotate Angle** if SSF + ASC mode is chosen. Set **Target EXP Deformation**, the target deformation value. Set **Calibration Tolerance**, an acceptable level of difference between experimental and simulated deformation.
3. Solve the calibration simulation	In Workbench, right-click **Optimization** > **Update** in the Direct Optimization system to initiate the calibration. Calibration iterations are fully automatic. Requires an Additive Suite license
4. View and save optimized calibration coefficients	Record the last set of parameters and then Save Build Settings.

Details of Workflow

Let's look at the workflow to run a calibration simulation in Mechanical in a little more detail.

1. Set up either an LPBF Thermal-Structural or LPBF Inherent Strain simulation. See Create the Analysis System (p. 67).

2. Import your chosen calibration geometry.

3. Use the LPBF Setup Wizard (p. 17) to set up your simulation the same as you would to simulate your production part.

4. There are a few preferred mesh types for LPBF simulations. See Apply Mesh Controls and Generate Mesh (p. 78). Mesh type and size affect results convergence. For best results, we recom-

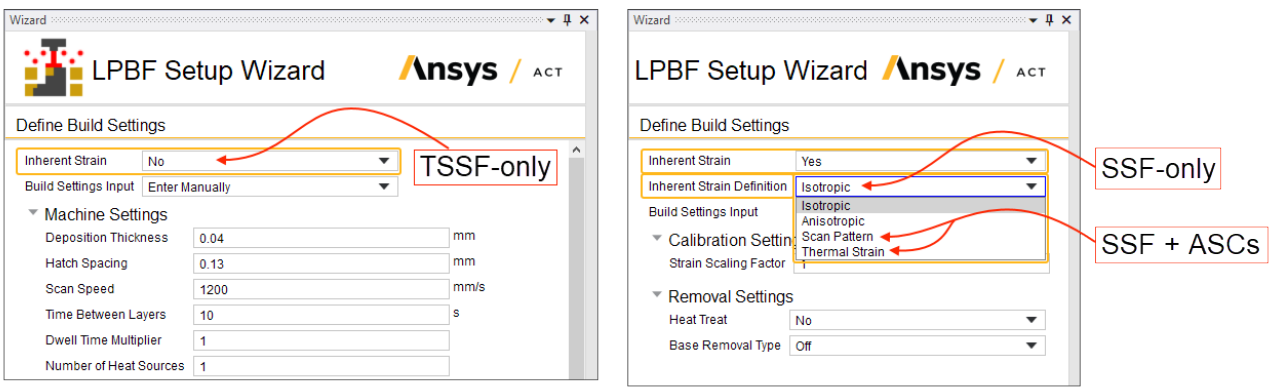
mend you use the same mesh *type* for the calibration simulation that you will use for the simulation of your production part.

These are our mesh *size* recommendations for the Ansys-supplied Cantilever and Double arches calibration geometries:

Geometry	Mesh type	Mesh size (mm)	Layer height (mm)	Projection Factor
Double arches	Voxel	0.5	-	-
	Layered tetrahedrons	0.5	0.5	-
	Cartesian	0.5	-	0
Cantilever	Voxel	0.5	-	-
	Layered tetrahedrons	0.75	0.75	-
	Cartesian	0.5		0

5. The selections you make in the Build Settings step determine whether you will need a SSF/TSSF-only calibration or an SSF + ASCs calibration.

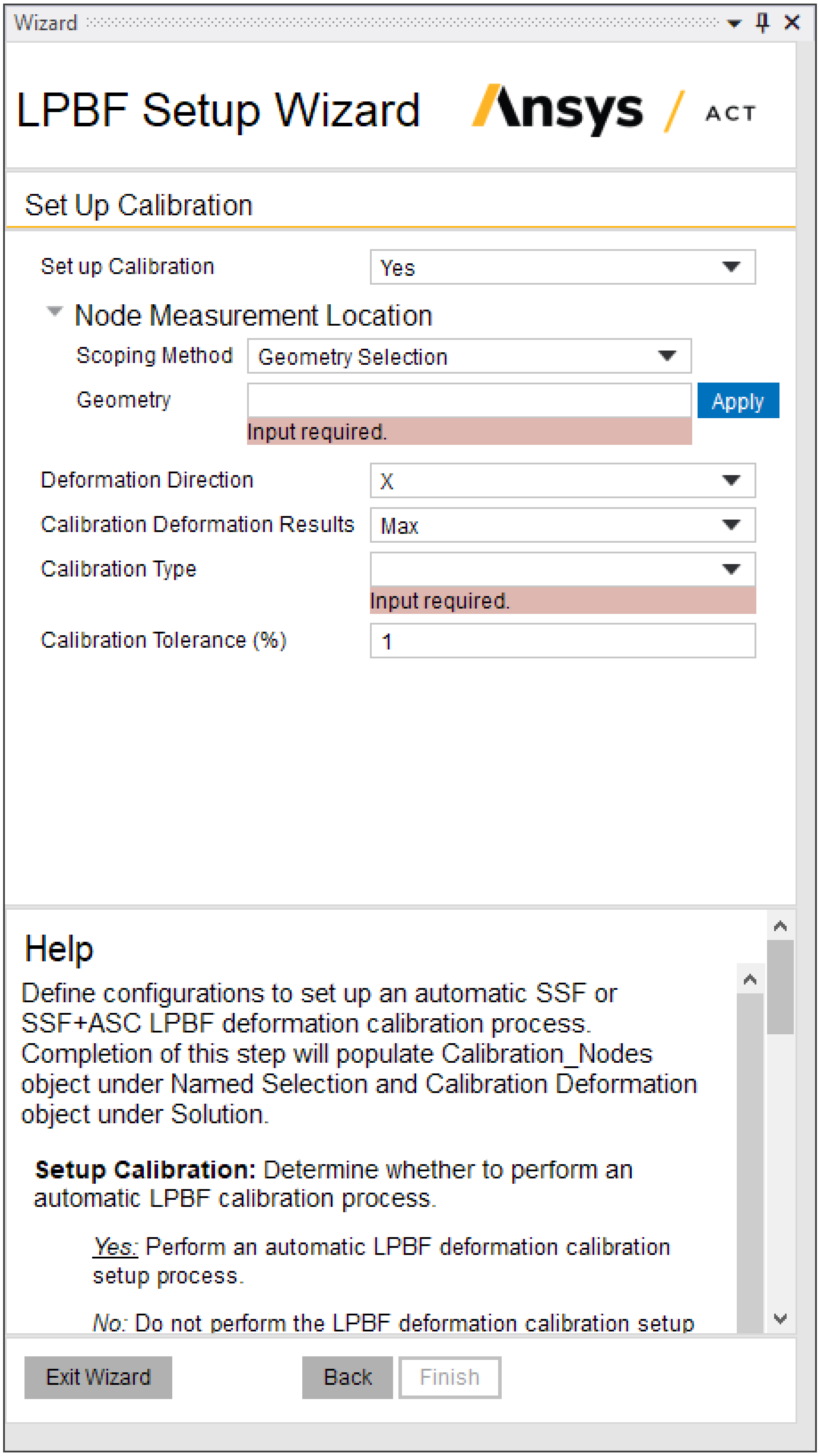
LPBF Thermal-Structural Simulation LPBF Inherent Strain Simulation



Note that the Inherent Strain Definition of Anisotropic is not available/supported for automatic calibration.

6. Use default values for calibration coefficients.

7. The last step of the LPBF Setup Wizard is the calibration set-up. Follow the instructions in the Help section to complete the settings.



Set up Calibration = Yes

Node Measurement Location: Use geometry selection or named selection to identify the nodes corresponding to the measurement location. Directional deformation data will be extracted here for comparison to the target EXP deformation values. It may be easiest to define a named selection based on geometric location such as a line of nodes along an edge located at X=0 and Y=5 mm, for example. Refer to the tutorial for an example.

Deformation Direction: The direction, either X, Y, or Z, in which deformation is measured. Usually this is normal to the surface of the measurement location.

Calibration Tolerance: An acceptable level of difference between experimental and simulated deformation. Defaults to 1%.

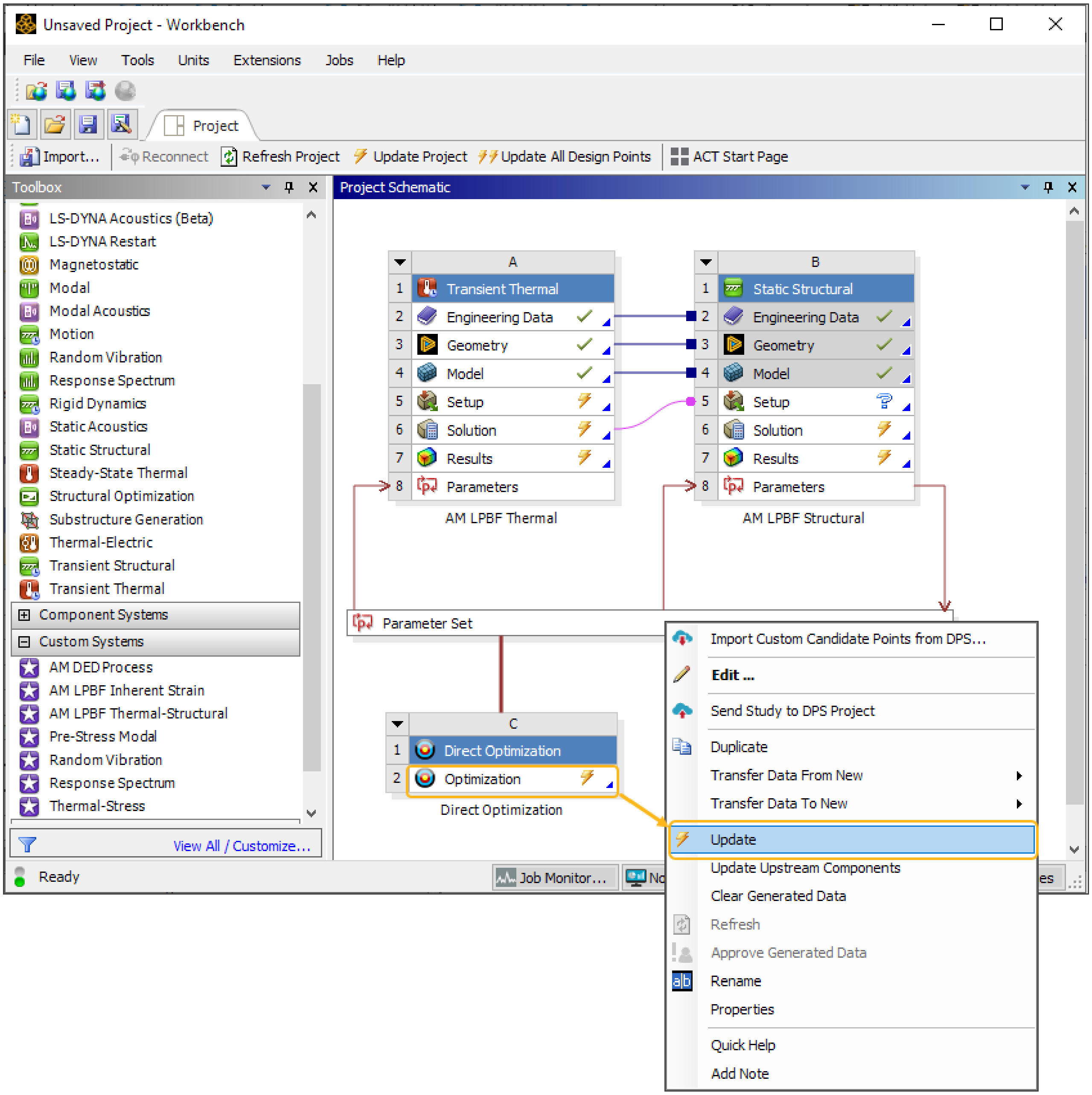
Calibration Deformation Results: Either Max, Min, or Avg. Choose this input to match the data you measured.

Calibration Type: Either SSF or SSF + ASCs. This is set automatically based on your settings of simulation type and Strain Definition.

Target EXP Deformation: The target deformation value.

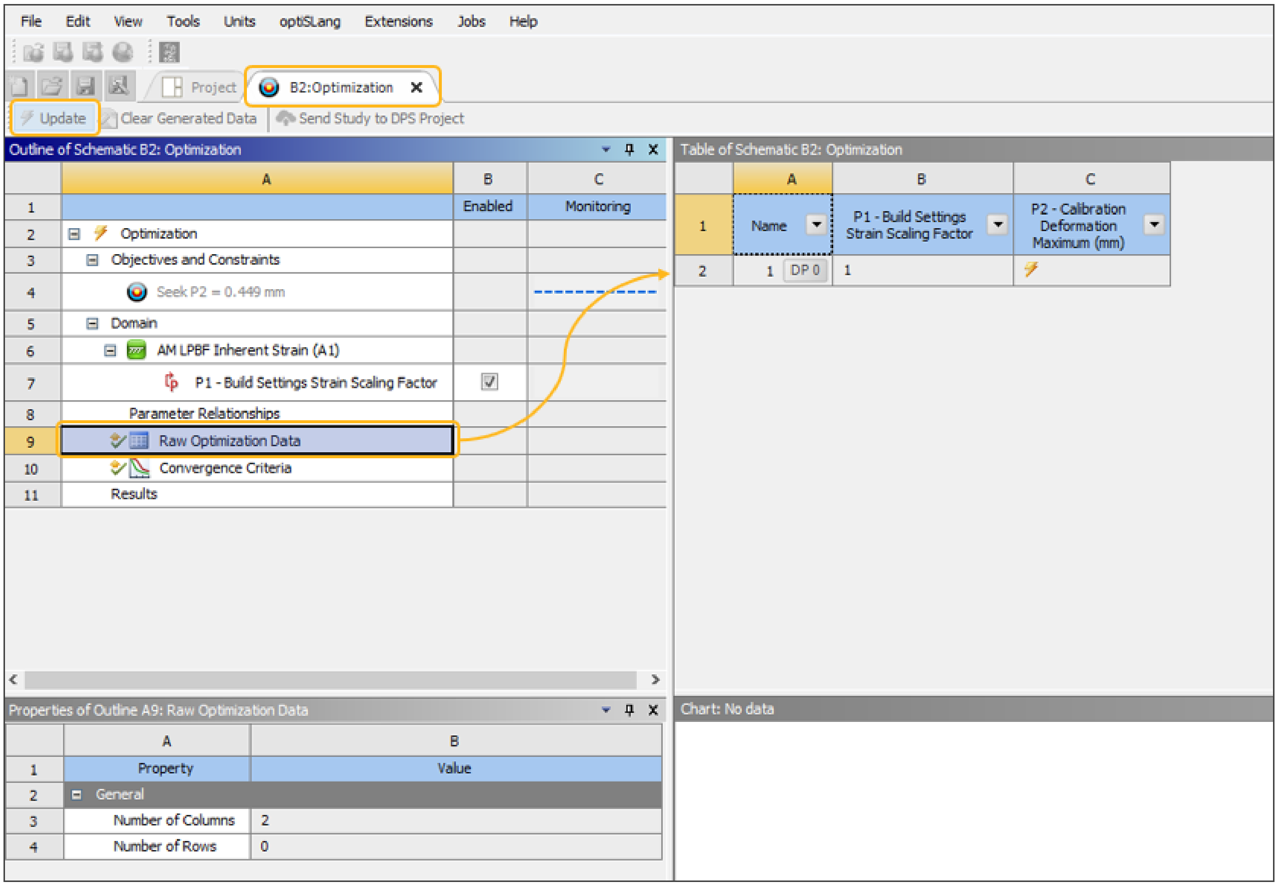
8. Once all the required settings are configured, click **Finish** and close the wizard. Proceed back to Workbench UI. For the calibration iterations, a pre-configured Direct Optimization system is added to the project and linked to the AM simulation system.

   In Workbench, right-click **Optimization** > **Update** in the Direct Optimization system. The optimization will start to run calibration iterations automatically. Depending on the calibration mode you are using, either a one step SSF/TSSF-only calibration or a two-step SSF + ASCs calibration is performed.

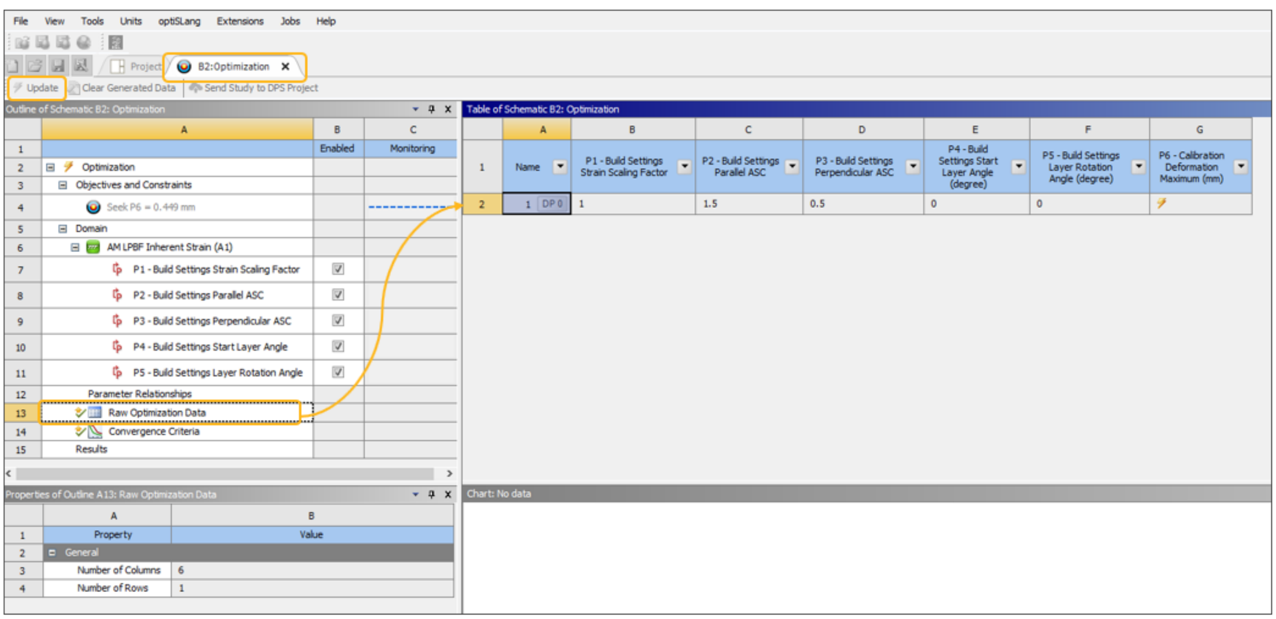


9. To check the progress during calibration iterations, double-click **Optimization** (in the Direct Optimization system) and click **Raw Optimization Data**.

Data for SSF/TSSF-only calibration

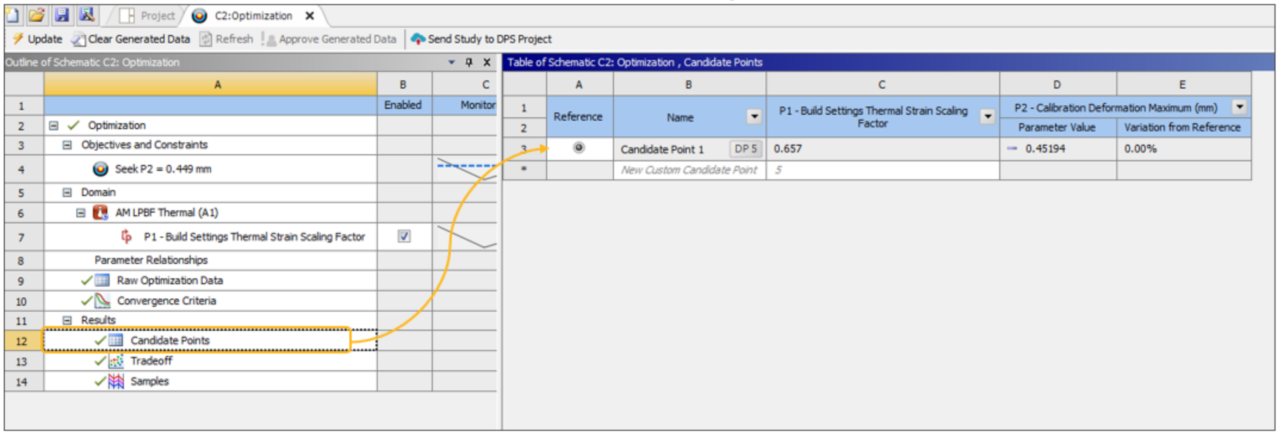


Data for SSF + ASCs calibration

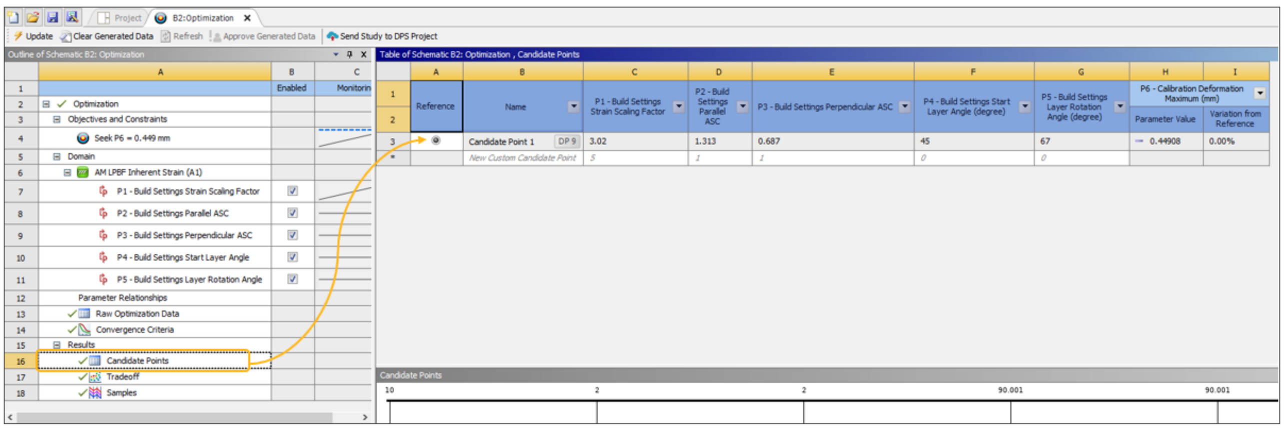


10. Once the optimization is completed, you will see the yellow lightening bolt become a green check mark in the Direct Optimization system. In the Optimization tab, click **Candidate Points**. The optimized values for calibration coefficients with their corresponding simulation deformation values will be shown on the right.

Results for SSF/TSSF-only calibration



Results for SSF + ASCs calibration



11. We recommend that you save the optimized SSF and ASC values for use in future simulations of production parts. Use them whenever you simulate with the same set of variables for which you calibrated. See When to Calibrate (p. 200).

    In Workbench, double-click **Parameter Set**. Click inside the last row, then right-click and choose **Set Update Order by Row**. Set to Current. Back in the Mechanical application, right-click **Build Settings** in the project tree and select **Save Build Settings**. This will save the entire set of build parameters including the optimized calibration coefficients.

7.3. Known Limitations

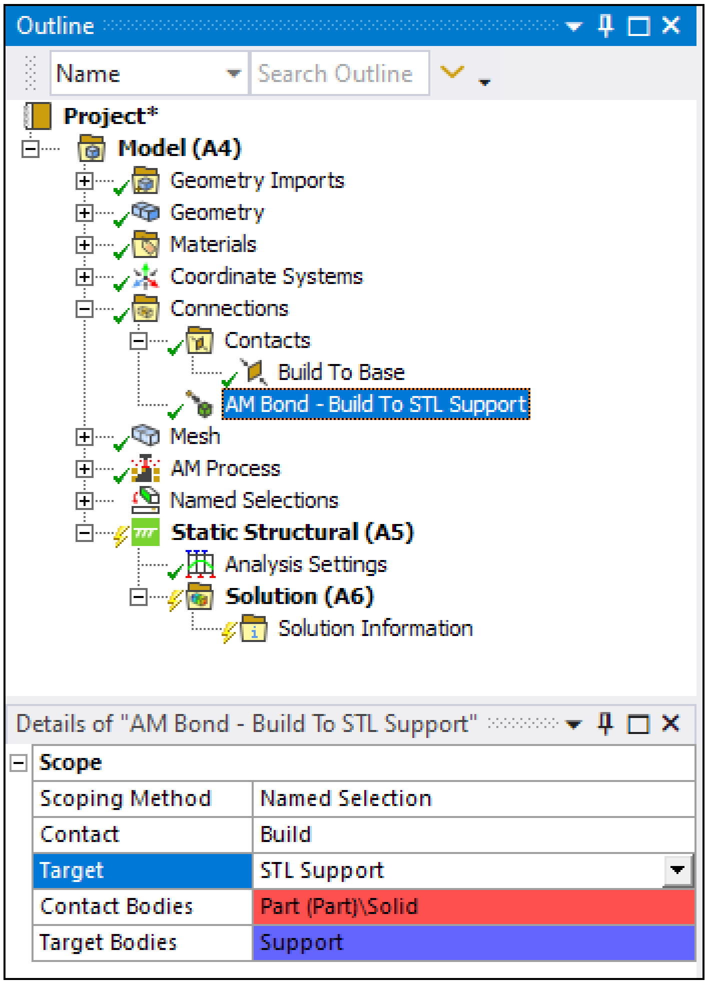
- The Inherent Strain Definition of Anisotropic is not available/supported in the LPBF Calibration Wizard.
- For calibrations that use cutoff results as target deformation values, turning on large deflection (Static Structural > Analysis Settings under Solver Controls) is a reasonable option to consider. However, convergence issues may arise from time to time during the calibration process.

Chapter 8: Object Reference

Specifications are provided for the following Mechanical application objects that are unique to LPBF process simulations:

8.1. AM Bond

Connects a meshed part to a meshed support when the mesh is non-contiguous between them in an AM Process Simulation. The internal means of connection is through constraint equations that connect the support nodes to the part elements.



Object Properties

The Details Pane for this object includes the following properties.

Category	Properties/Options/Descriptions
Scope	**Scoping Method**: Options include **Geometry Selection** and **Named Selection** (default). **Contact**: In most cases, choose the part as the contact.[a] When the **Scoping Method** is set to **Named Selection**, select a desired Named Selection from the drop-down menu. When the **Scoping Method** is set to **Geometry Selection**, use selection filters to pick the part geometry, click in the **Geometry** field, then click **Apply**. **Target**: In most cases, choose the support as the target.[a] When the **Scoping Method** is set to **Named Selection**, select a desired Named Selection from the drop-down menu. **Contact Bodies**: Read-only indication of scoped geometry or Named Selection. **Target Bodies**: Read-only indication of scoped geometry or Named Selection.

[a] Reversing the entities identified as contact and target will not only produce a warning message, but will typically produce unexpected and unrealistic deformation. However, if the support mesh is much coarser than the part mesh, say greater than 2:1 proportions, reverse the selection and make the support the contact and the part the target.

Tree Dependencies

- **Valid Parent Tree Object**: Connections.
- **Valid Child Tree Objects**: None.

Insertion Methods

Use any of the following methods after highlighting Connections object:

- Select **AM Bond** on the Connections Context Tab.
- Right-click the Connections object or in the **Geometry** window and select **Insert** > **AM Bond**.

Right-click Options

In addition to common right-click options, relevant right-click options for this object include:

- **Suppress**: Suppresses the contact connection from the simulation.
- Enable/Disable Transparency
- Hide All Other Bodies
- **Flip Contact/Target**: Reverses the identification of contact and target entities. For AM Bond connections, typically the part is the contact and the support is the target. However, if the support mesh is much coarser than the part mesh, say greater than 2:1 proportions, use this option to flip the selection and make the support the contact and the part the target.

API Reference

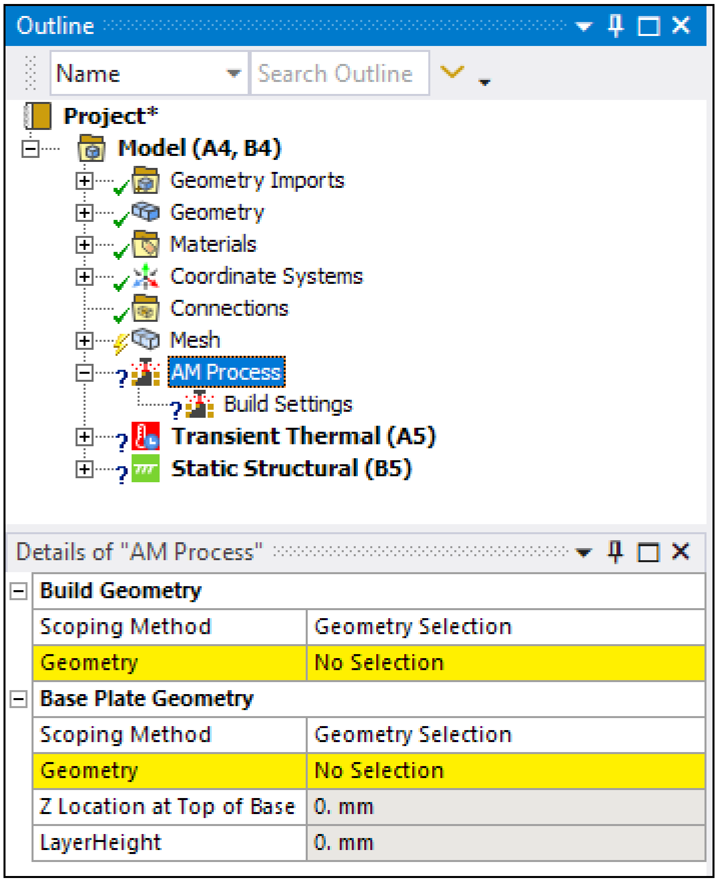
See the AM Bond Connection section of the ACT API Reference Guide for specific scripting information.

Additional Related Information

See Define Connections in the *LPBF Simulation Guide* (p. 93) section for more information.

8.2. AM Process

Identifies geometries and sets global options for all AM-related objects in an AM Process Simulation.



Object Properties

The Details Pane for this object includes the following properties.

Category	Properties/Options/Descriptions
Build Geometry	The part to be printed. **Scoping Method**: Options include **Geometry Selection** (default) and **Named Selection**. **Geometry**: Use selection filters to pick geometry, click in the **Geometry** field, then click **Apply**. **Named Selection**: Select a desired Named Selection from the drop-down menu.
Base Geometry	The base plate upon which the part rests. **Scoping Method**: Options include **Geometry Selection** (default) and **Named Selection**. **Geometry**: Use selection filters to pick geometry, click in the **Geometry** field, then click **Apply**. **Named Selection**: Select a desired Named Selection from the drop-down menu. **Z Location at the Top of Base**: Read-only indication of the Z location at the top of the base plate. This is often 0.

Category	Properties/Options/Descriptions
	Layer Height: Read-only indication of the layer height used in the mesh.

Tree Dependencies

- **Valid Parent Tree Object**: Model.
- **Valid Child Tree Object**: Build Settings and Support Group.

Insertion Methods

The AM Process object is inserted under the Model object in the project tree by default when using either of these two custom systems for additive manufacturing: AM LPBF Inherent Strain or AM LPBF Thermal-Structural. When not inserted automatically by a custom system, use any of the following methods after highlighting **Model** object:

- Select the **AM Process** option on the Model Context Tab.
- Right-click the **Model** object or in the geometry window and select **Insert** > **AM Process**.

Right-click Options

In addition to common right-click options, relevant right-click options for this object include:

- **Insert** >
  - Support Group
  - Predefined Support
  - Generated Support
  - STL Support
  - **Cartesian Mesh**
- **Create Build To Base Contact**: This option creates a **Contact Region** between the **Build Geometry** and the **Base Plate Geometry**. The **Contact Side** is defined as the element faces of the bottom of the Build or Support. The **Target Side** is defined as the element faces of the top of the Base Plate Geometry. Both contact and target scopes are defined by Named Selection. This option is visible once you have generated a mesh.

API Reference

See the AM Process section of the ACT API Reference Guide for specific scripting information.

Additional Related Information

See the following sections for more information:

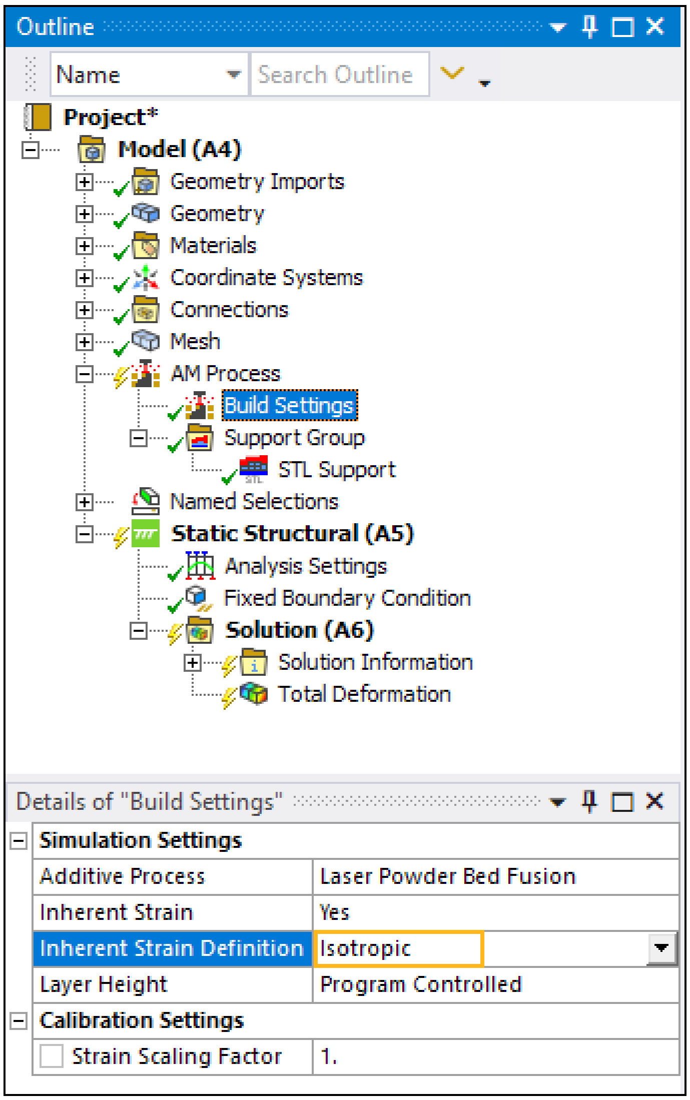
- LPBF Simulation Guide (p. 1)

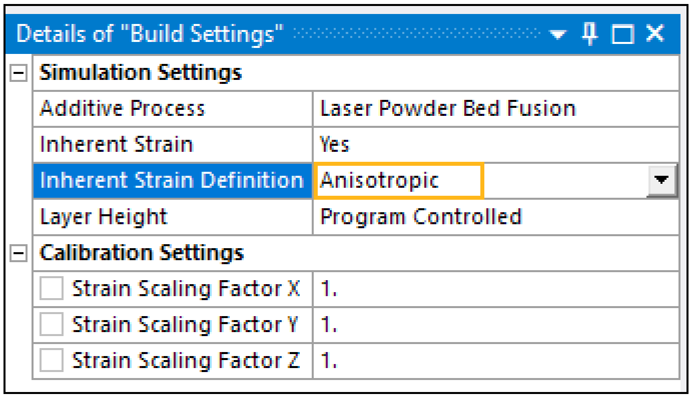
• Identify Geometries (AM Process Object) (p. 76)

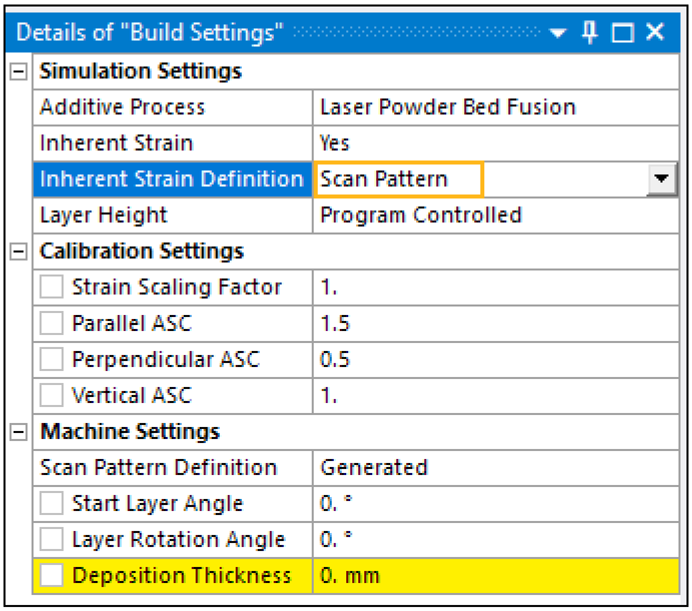
8.3. Build Settings

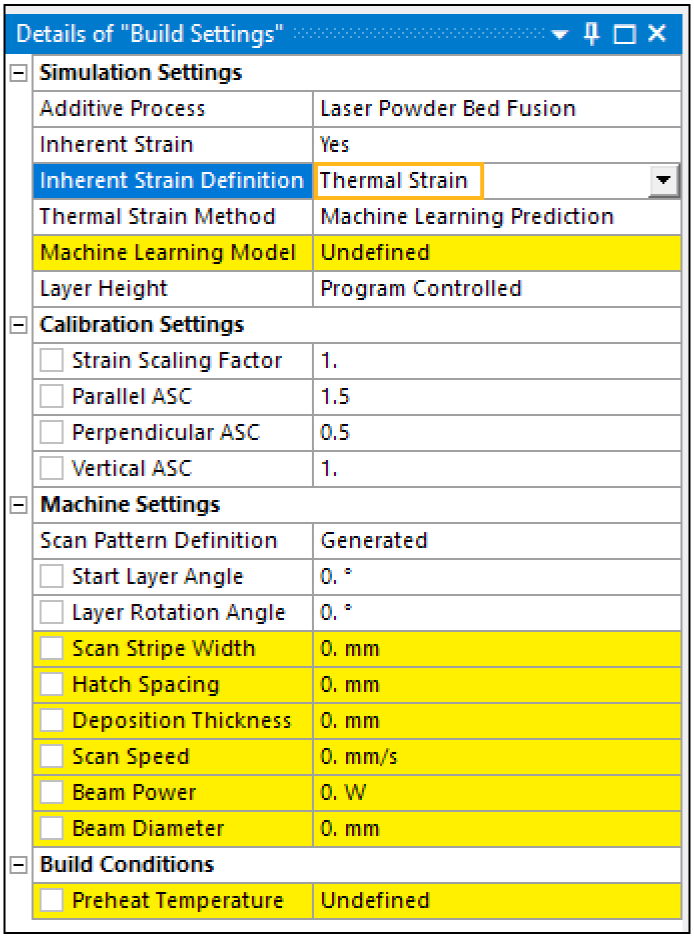
Defines simulation and strain assumptions, process parameters, and build conditions related to an additive manufacturing material, machine, and process.

Build Settings for an AM LPBF Inherent Strain system showing Details for the four strain definitions (Isotropic, Anisotropic, Scan Pattern, and Thermal Strain):

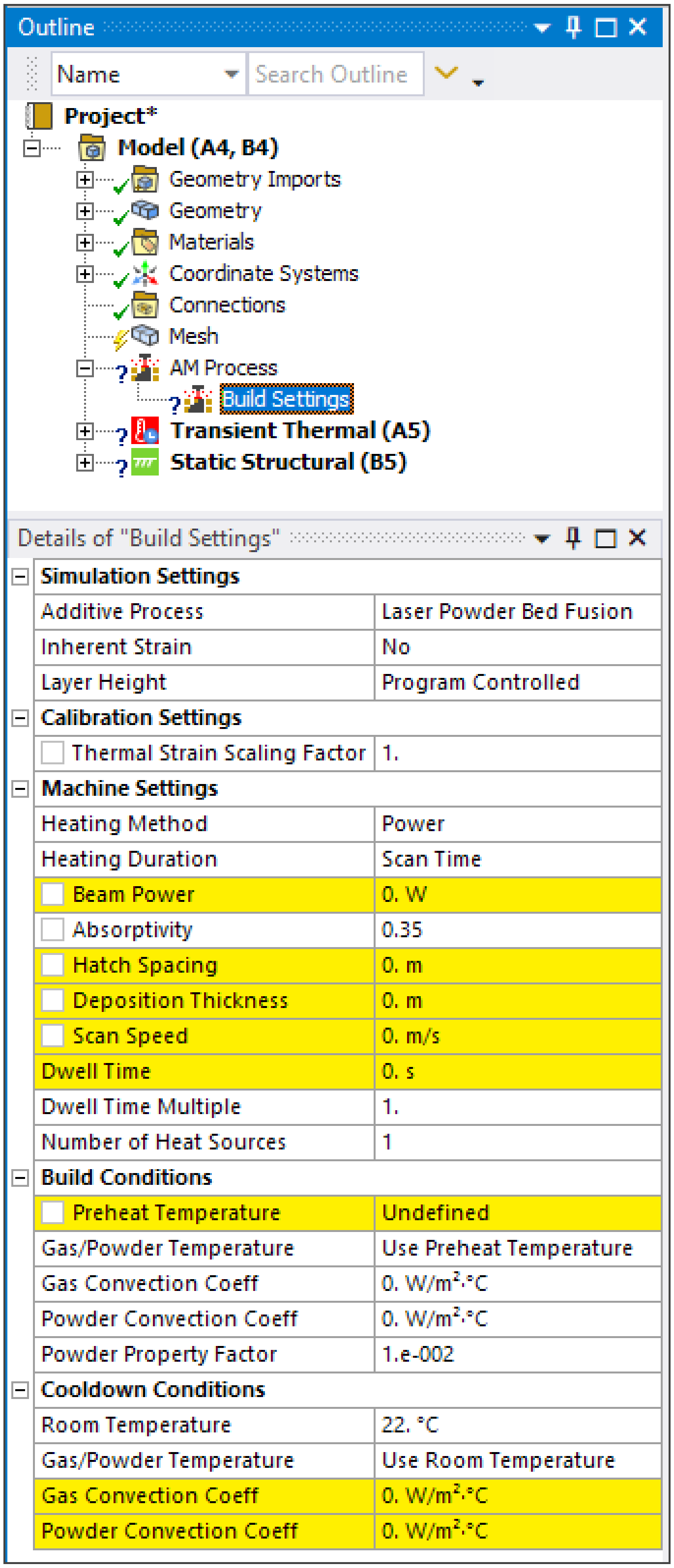



Details of "Build Settings"
Simulation Settings
Additive Process
Laser Powder Bed Fusion
Inherent Strain
Yes
Inherent Strain Definition
Anisotropic
Layer Height
Program Controlled
Calibration Settings
Strain Scaling Factor X
1.
Strain Scaling Factor Y
1.
Strain Scaling Factor Z
1.


Details of "Build Settings"
Simulation Settings
Additive Process
Laser Powder Bed Fusion
Inherent Strain
Yes
Inherent Strain Definition
Scan Pattern
Layer Height
Program Controlled
Calibration Settings
Strain Scaling Factor
1.
Parallel ASC
1.5
Perpendicular ASC
0.5
Vertical ASC
1.
Machine Settings
Scan Pattern Definition
Generated
Start Layer Angle
0. °
Layer Rotation Angle
0. °
Deposition Thickness
0. mm



Build Settings for an AM LPBF Thermal-Structural system shown with power heating method:



Object Properties

The Details Pane for this object includes the following properties.

Category	Properties/Options/Descriptions
Simulation Settings	**Additive Process**: The Additive Process Type - Laser Powder Bed Fusion. The LPBF process uses thermal energy from a laser or electron beam to selectively fuse powder in a powder bed. Inherent Strain (p. 138): Yes or No. If Yes, the AM simulation uses a static structural system and the loading strains are calculated based on an experimentally calibrated Strain Scaling Factor. If No, the AM simulation uses a linked thermal-structural system in which strains are calculated from temperature-dependent material properties (p. 69) and loads. The remaining options under Machine Settings differ depending on whether Inherent Strain = Yes or No. If **Inherent Strain** = **Yes**: • **Inherent Strain Definition**: Isotropic, Anisotropic, Scan Pattern, or Thermal Strain. – **Isotropic** assumes that a constant, uniform strain occurs at every location within a part as it is being built. – **Anisotropic** uses the same average strain magnitude as isotropic strain, but it subdivides that strain into anisotropic components in the X, Y, and Z directions based on the Global coordinate system. – **Scan Pattern** uses the same average strain magnitude as isotropic strain, but it subdivides that strain into anisotropic components based on the local orientation of scan vectors within the part. Scan vectors may be generated internally via a slicing function assuming a rotating stripe scan pattern or input via a build file. – **Thermal Strain** is a method that provides the highest level of fidelity and takes thermal cycling into account at each location within the part. • **Thermal Strain Method**: At this release, only the Machine Learning Prediction method is available. – **Machine Learning Prediction** uses a machine learning model prediction of the anisotropic Thermal Strain simulation result from the Ansys Additive application.—Thermal Strain simulations provide the highest level of fidelity by predicting how thermal cycling affects strain accumulation at each location within a part. The simulation follows the full laser path on every layer, and is based on the machine process parameters (power, scan speed, beam diameter, etc.)—The machine learning model has been trained to predict the Thermal Strain result much faster than simulation. It can be one to three orders of magnitude faster than Thermal Strain simulation in Additive Print in calculating the strain that is passed to the structural solver. Speedup increases with part size, scan area, and melt pool size. See Thermal Strain - Anisotropic in the *Additive Print and Science User's Guide* and Understanding Machine Learning Thermal Strain (p. 171).

<table>
<tr><th>Category</th><th>Properties/Options/Descriptions</th></tr>
<tr><td></td><td>

- **Machine Learning Model**: A list of materials that were used to train the ML prediction. Choose the material that most closely matches your material assignment in Engineering Data. ML models may be based on different material properties than those in Engineering Data. The ML models are used to generate loading strains. Materials in Engineering Data are used for the structural analysis.
- **Layer Height**: Sets the element layer height for the mesh, which must conform to uniform layer sizes in the global Z direction. Options include **Program Controlled** (default) and **Manual**. For Program Controlled, the application finds the first layered tetrahedrons mesh method that is scoped to the AM build body and sets the Layer Height to the value specified in the Details pane of the layered tetrahedrons mesh method. If there are no layered tetrahedrons meshes present/scoped to the build, then no Layer Height value is used. When set to Manual, the user specified Layer Height is used, regardless of whether a layered tetrahedrons mesh is present.

</td></tr>
<tr><td>**Calibration Settings**</td><td>

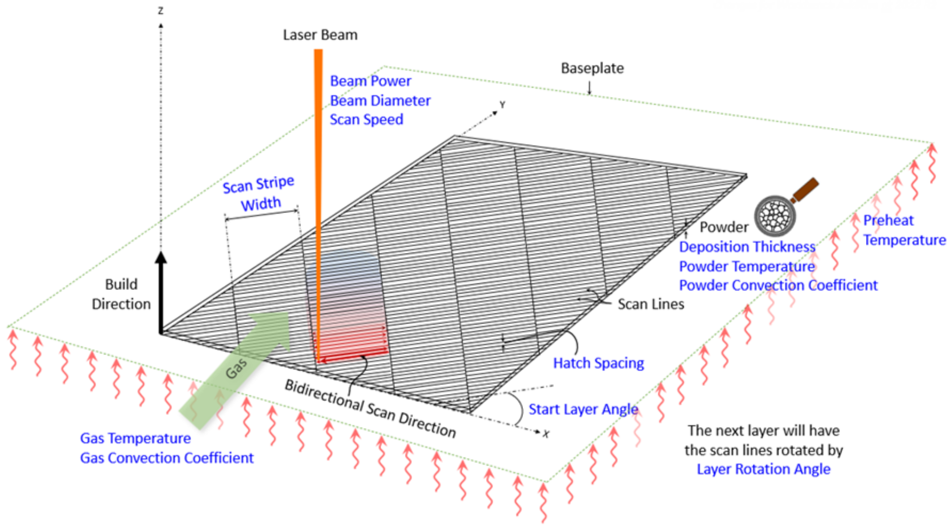
Strain Scaling Factor(s): A calibration factor, or factors, used to account for differences in additive machines and materials that you may use to improve the accuracy of your simulations. The SSF scales the inherent strains in the analysis by the given value.

- If **Inherent Strain** = **Yes**:
  - If **Inherent Strain Definition** = **Isotropic**, a constant Strain Scaling Factor may be entered to scale strain everywhere uniformly.
  - If **Inherent Strain Definition** = **Anisotropic**, individual Strain Scaling Factors may be entered for X, Y, and Z directions based on the Global coordinate system.
  - If **Inherent Strain Definition** = **Scan Pattern** or **Thermal Strain**, individual Strain Scaling Factors, called anisotropic scaling factors (ASC), may be entered based on the local orientation of scan vectors within the part, that is, parallel and perpendicular to scanning direction and in the build direction.
- If **Inherent Strain** = **No**:
  - **Thermal Strain Scaling Factor**: An optional input that scales the thermal strains in the structural portion of AM simulations by a given value.

</td></tr>
</table>

<table>
<tr><th>Category</th><th>Properties/Options/Descriptions</th></tr>
<tr><td>Machine Settings</td><td>

Properties in this category are illustrated in the following figure:



Machine parameters for a typical rotating stripe scan pattern

If **Inherent Strain** = **Yes**:

- **Scan Pattern Definition** (visible if Inherent Strain Definition = Scan Pattern or Thermal Strain): How the scan pattern is defined, either generated using a rotating stripe pattern (default) or input via a build file.
  - Generated: Start Layer Angle and Layer Rotation Angle as defined below. Scan Stripe Width is also visible if Inherent Strain Definition = Thermal Strain. These inputs define an internally generated scan pattern.
  - Build File: Machine Type and Build File Path inputs become available with this option. These inputs specify an external build file to be used.
    - → Machine Type: Specifies the machine or OEM associated with the build file specified. Options are Additive Industries, EOS, HB3D, Renishaw, Sisma, SLM, and Trumpf.
    - → Build File Path: Location of a .zip file containing the scan pattern file(s), and an stl of the part geometry.

      Build files for simulations that use the Machine Learning Prediction Thermal Strain Method only support *stripe* scan patterns. See Build File Requirements in the *LPBF Simulation Guide* (p. 109) for additional requirements.
- **Start Layer Angle**: The orientation of fill rasters on the first layer of the build. It is measured from the X axis, such that 0 degrees results in scan lines parallel to the X axis. The starting layer angle is commonly set to 57 degrees. Must be between 0 and 180°.

</td></tr>
</table>

Category	Properties/Options/Descriptions

	• **Layer Rotation Angle**: The angle at which the major scan vector orientation changes from layer to layer. It is commonly 67 degrees. Must be between 0 and 180°.
	
	• **Scan Stripe Width**: Width of the sections, called stripes, into which the geometry is sliced. The stripes are scanned sequentially to break up what would otherwise be very long continuous scan vectors. Scan Stripe Width is commonly set to 10 mm wide. Must be between 1 and 100 mm.
	
	• **Hatch Spacing**: The average distance between adjacent scan vectors when rastering back and forth with the laser. Hatch spacing should allow for a slight overlap of scan vector tracks such that some of the material re-melts to ensure full coverage of solid material. For Machine Learning strain definition, must be between 60 and 1000 microns.
	
	• **Deposition Thickness**: The thickness of added powder material in every pass of the recoater blade. Specifically, use the amount the base plate drops between layers. For Thermal Strain strain definition, must be between 10 and 100 microns.
	
	• **Scan Speed**: The average speed at which the laser spot moves across the powder bed along a scan vector to melt material, excluding jump speeds and ramp up and down speeds. For Thermal Strain strain definition, must be between 350 and 2500 mm/sec and the recommended range is between 500 and 2500 mm/sec.
	
	• **Beam Power**: The power setting for the laser in the machine. Must be between 50 and 700 Watts. The recommended range is between 50 and 500 Watts.
	
	• **Beam Diameter**: The width of the laser on the powder or substrate surface defined using the D4σ beam diameter definition. Usually, this value is provided by the machine manufacturer. Sometimes called laser spot diameter. Must be between 20 and 140 µm. The recommended range is between 80 and 120 µm.
	
	If **Inherent Strain** = **No**:
	
	• **Heating Method**: Controls how new layers are heated in the transient thermal analysis. Choose Melting Temperature (default) or Power.
	  – Melting Temperature sets the new layer to the melt temperature specified in Engineering Data.
	  – Power uses a heat generation load to heat new layers.
	
	• **Heating Duration**: Visible if Heating Method = Power. The amount of time the heat is applied, either Scan Time (default) or Flash.
	  – **Scan Time**: Heat is applied for the time it takes to scan the volume of material in each element layer. This setting may give better temperature results at the end of the layer but may not yield a temperature spike above melting. Note that this Scan Time option will have a different

Category	Properties/Options/Descriptions
	end time of the simulation because the layer thickness adjustment for cooling will not be used. – **Flash**: Heat is applied in a very short time increment resulting in a spike in temperature then cooling before the next element layer is added. • **Beam Power**: Visible if Heating Method = Power. The power of the laser. • **Absorptivity**: Visible if Heating Method = Power. The average fraction of energy that is absorbed by the deposited material and contributes to the heating process. Must be between 0 and 1. Defaults to 0.35. • **Hatch Spacing**: See above. • **Deposition Thickness**: See above. • **Scan Speed**: See above. • **Dwell Time**: The span of time from the end of the deposition of a layer to the start of the deposition of the next layer. It includes the time required for recoater-blade repositioning and powder-layer spreading. • **Dwell Time Multiple**: The dwell-time multiplier accounts for more than one part in the build. If they are the same part arranged in the same orientation on the build plate, the multiplier is the number of parts. • **Number of Heat Sources**: For multiple-beam printers, specifies the number of beams. This divides the time it takes to scan a layer by the number of heat sources specified.
Build Conditions	**Preheat Temperature**: The starting temperature of the build plate. Used when Inherent Strain = Yes and Inherent Strain Definition = Thermal Strain, and when Inherent Strain = No. For Thermal Strain strain definition, must be between 20 and 500 °C and the recommended range is between 20 and 200 °C. If **Inherent Strain** = **No**: • **Gas/Powder Temperature**: Options include **Use Preheat Temperature** (default) and **Specified**. • **Gas Convection Coefficient**: Convection coefficient from the build to the enclosure gas. The convection is applied only to the top of a newly laid layer. • **Gas Temperature**: Temperature of the gas in the build enclosure. • **Powder Convection Coefficient**: Effective convection coefficient from the sides of the build to the powder bed. To estimate, divide the conduction property of the powder by a characteristic conduction length into the powder (for example, a quarter of the distance from the build boundary to the build-chamber wall). • **Powder Temperature**: Temperature of the newly added powder.

Category	Properties/Options/Descriptions
	• **Powder Property Factor**: The application uses this factor to estimate the powder properties. The application applies the factor to the solid material properties to estimate the properties of the material in its powder state. The powder-state properties are used during the heating of the new layer (before its subsequent solidification and cooldown) prior to the next layer being applied.
Cool-down Con-di-tions (**In-her-ent Strain** = **No** option only)	**Room Temperature** **Gas/Powder Temperature**: Options include **Use Room Temperature** (default) and **Specified**. **Gas Convection Coefficient**: Convection coefficient from the build to the enclosure gas. The convection is applied only to the top of a newly laid layer. **Gas Temperature**: Temperature of the gas in the build enclosure. **Powder Convection Coefficient**: Effective convection coefficient from the build to the powder bed. To estimate, divide the conduction property of the powder by a characteristic conduction length into the powder (for example, a quarter of the distance from the build boundary to the build-chamber wall). **Powder Temperature**: Temperature of the newly added powder.

Tree Dependencies

- **Valid Parent Tree Object**: AM Process.
- **Valid Child Tree Objects**: None.

Insertion Methods

Inserted automatically by the AM Process object.

Right-click Options

In addition to common right-click options, relevant right-click options for this object include:

- **Save Build Settings**: Saves your build settings as an XML file. The property values are always saved in Consistent MKS Unit System.
- **Load Build Settings**: Imports a build setting XML file. Once loaded, the settings properly populate the properties of the **Details** pane. View and use example XML files at: [Ansys Installation Directory]\aisol\DesignSpace\DSPages\SampleData\AdditiveManufacturing.
- **Reset to Default**: Resets build settings to default values.

API Reference

See the AM Build Settings section of the ACT API Reference Guide for specific scripting information.

Additional Related Information

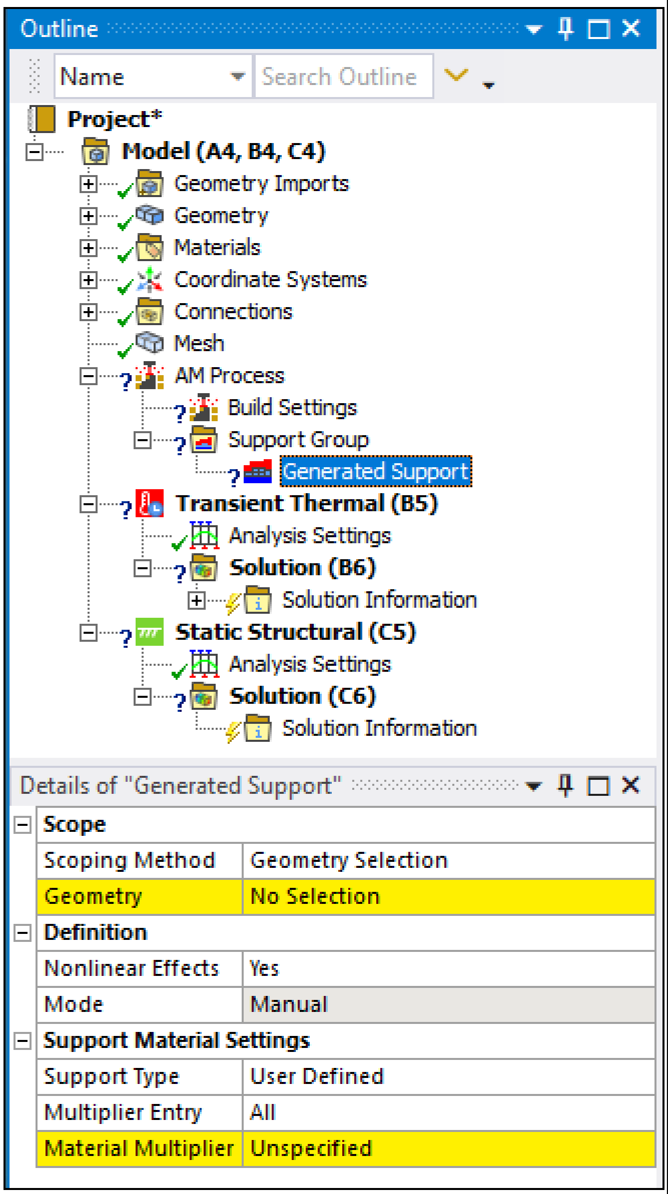
LPBF Simulation Guide (p. 1)

Define Build Settings (p. 104)

8.4. Generated Support

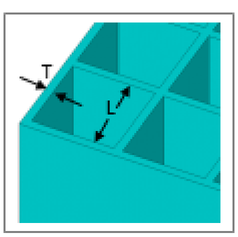
Generates a support structure consisting of finite elements for Additive Manufacturing simulations. Elements are generated between the part geometry and the base plate geometry based on an existing mesh and the criteria set in Support Group.

After inserting this object, select element faces of the part under which you want elements to be generated. Then right-click and select Generate Support Bodies to generate the elements. Elements are generated straight down from there to fill the gap between the part and the base plate.



Object Properties

The Details Pane properties for this object include the following.

Category	Properties/Options/Descriptions
Scope	**Scoping Method**: Options include **Geometry Selection** (default) and **Named Selections**. **Geometry**: Displays when the **Scoping Method** is set to **Geometry Selection**. Using the **Element Face** selection filter on the graphics toolbar, select the desired element faces and click **Apply**. **Named Selection**: Displays when the **Scoping Method** is set to **Named Selection**. It provides a drop-down menu. Only element face-based Named Selections are displayed in the drop-down menu.
Defin-i-tion	Nonlinear Effects: Options include **Yes** (default) and **No**. This property applies nonlinear material effects to the generated elements. **Mode**: Indicates whether you manually specified the **Geometry Selection** or whether it was detected using the options of the **Support Group** object. **Important:** Only automatically generated supports are affected by the **Generate on Remesh** property of the **Support Group** object **Visible**: Show or hide the generated elements whenever the mesh is displayed (using the **Show Mesh** option or when the mesh object is selected).
Sup-port Ma-ter-ial Set-tings	**Support Type**: Options include **User Defined** (default) and **Block**. When you select the **Block** option, the following properties become the only visible properties. • **Wall Thickness**: Enter a thickness value (illustrated by **T** length below). • **Wall Spacing**: Enter a spacing value (illustrated by **L** length below).  **Multiplier Entry**: Options include **All** (default) and **Manual**. **Material Multiplier** **All Setting** When the **Multiplier Entry** property is set to **All**, the **Material Multiplier** property displays. The **Material Multiplier** property applies the same value

Category	Properties/Options/Descriptions
	to all of the material multipliers listed below. The multiplication factors are homogenization factors and, in each direction, reflect the ratio of the support area projected onto the area of a fully solid support. **Manual Setting** When the **Multiplier Entry** property is set to **Manual**, the following multiplier properties display: • **Elastic Modulus Multiple in X/Y/Z** • **Shear Modulus Multiple in XY/YZ/XZ** • **Density Multiple** • **Thermal Conductivity Multiple in X/Y/Z**
Statistics	**Nodes**: Read-only property that displays the number of nodes in the generated elements. **Elements**: Read-only property that displays the number of generated elements. **Volume**: Read-only property that displays the volume of the generated elements.

Tree Dependencies

- **Valid Parent Tree Object**: Support Group, which is under AM Process object.
- **Valid Child Tree Objects**: None.

Insertion Methods

- Select the **AM Process** object and then select the **Generated Support** option from the AM Process Context Tab.
- Right-click the **AM Process** object and then select **Insert** > **Generated Support**.
- Select the **Support Group** object and then select the **Generated Support** option from the AM Process Context Tab.
- Right-click the **Support Group** object and then select **Insert** > **Generated Support**.

Right-click Options

In addition to common right-click options, relevant right-click options for this object include:

- **Insert**
  - Predefined Support
  - Generated Support

- STL Support
- Commands

- **Create Named Selection of Generated Elements**
- **Hide Support**
- **Hide All Other Bodies**
- **Generate Support Bodies**
- Clear Generated Data

API Reference

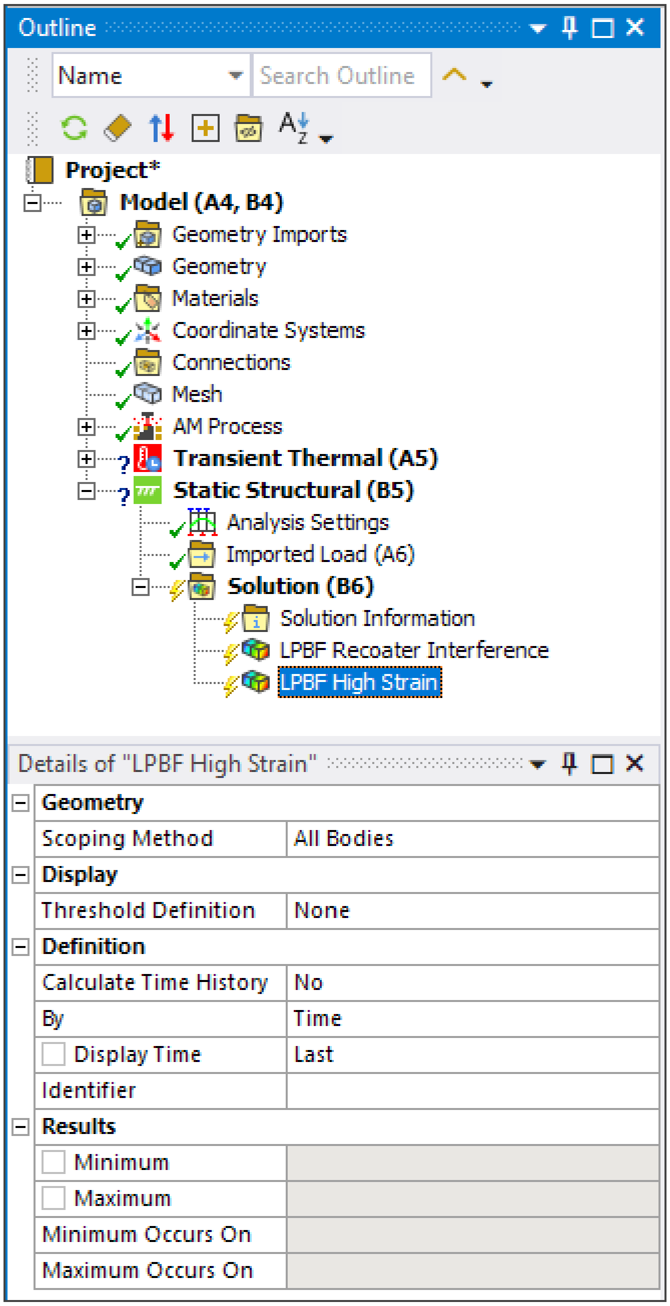
See the Generated AM Support section of the ACT API Reference Guide for specific scripting information.

Additional Related Information

- LPBF Simulation Guide (p. 1)
- Identify and/or Generate Supports (p. 86)

8.5. LPBF High Strain

Identifies areas of high strain that are prone to cracking in an AM Process simulation.



Object Properties

The Details Pane for this object includes the following properties.

Category	Properties/Options/Descriptions
Geometry	**Scoping Method**: Options include Geometry Selection, Named Selection, All Bodies (default), and Material IDs.

Category	Properties/Options/Descriptions
Display	**Threshold Definition**: • **None** (default): The result tool will show pure maximum equivalent strain in the build, where the value at each point corresponds to the strain when that material was a new layer. • **User Defined**: Specify your own warning and critical thresholds. – Warning Threshold: Strains above this are considered to be in a warning state for potential cracking. – Critical Threshold: Strains above this are considered to be in a critical state for potential cracking. If None is selected, the display will show the default color range. For User Defined threshold definition, the display colors are: • Blue = good • Green = warning • Red = critical
Definition	**Calculate Time History**: No (default) or Yes. • **No** (default): Results are not time dependent. Ensures a quick, dynamic calculation of results. • **Yes**: Setting Calculate Time History to Yes is unnecessary unless you want to animate the result. Set to Yes to see the time history results, such as for a layer-by-layer animated build. For large models, the calculation may take some time.
Results	Read-only properties displaying the following: **Minimum**: The minimum equivalent strain experienced during the build process. **Maximum**: The maximum equivalent strain experienced during the build process. **Minimum Occurs On**: The geometry body (part or support) on which the minimum equivalent strain occurs. **Maximum Occurs On**: The geometry body (part or support) on which the maximum equivalent strain occurs.

Tree Dependencies

- **Valid Parent Tree Object**: Solution, under Static Structural.
- **Valid Child Tree Objects**: None.

Insertion Methods

- Click the **LPBF High Strain** button in the LPBF Process ribbon.
- Under the Static Structural system, right-click **Solution** and then select **Insert** > **LPBF High Strain**.

Right-click Options

In addition to common right-click options, relevant right-click options for this object include:

- **Evaluate All Results**: Retrieves the equivalent strain data and displays it in graphical form.
- **Clear Generated Data**: Clears evaluated data for this result item.

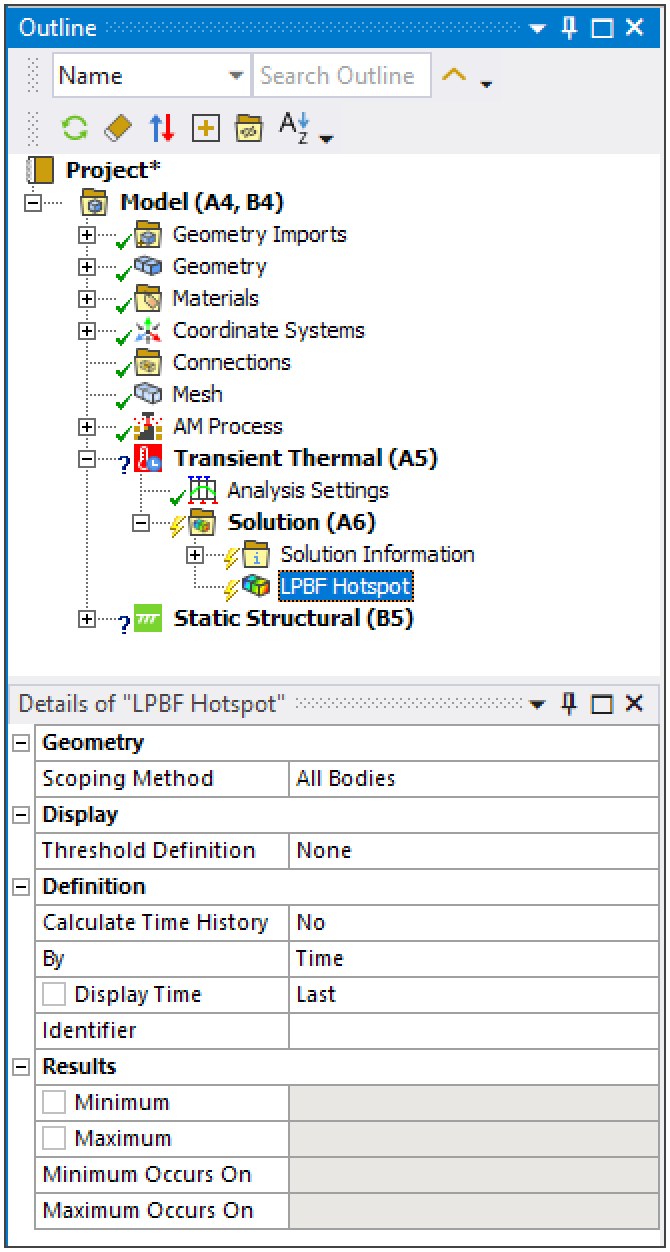
Additional Related Information

Establish Structural Analysis Settings (p. 117)

Review Results (p. 123)

8.6. LPBF Hotspot

Identifies areas of overheating that may result in problematic thermal conditions in an AM LPBF Thermal Structural simulation.



Object Properties

The Details Pane for this object includes the following properties.

Category	Properties/Options/Descriptions
Scope	**Scoping Method**: Options include Geometry Selection, Named Selection, All Bodies (default), and Material IDs.
Display	**Threshold Definition**: • **None** (default): The result tool will show temperature that the build cools down to for each layer right before the next layer is added. (It does not represent the "final state" of temperatures at the end of the entire process's

Category	Properties/Options/Descriptions
	cooldown step.) The worst hotspots are going to be the areas with the highest temperature for that layer. • **User Defined**: Specify your own warning and critical thresholds: – Warning Threshold: Temperatures above this are considered to be in a warning state for hotspots. – Critical Threshold: Temperatures above this are considered to be in a critical state for hotspots. If None is selected, the display will show the default color range. For User Defined threshold definition, the display colors are: • Blue = good • Green = warning • Red = critical
Definition	**Calculate Time History**: No (default) or Yes. • **No** (default): Results are not time dependent. Ensures a quick, dynamic calculation of results. • **Yes**: Setting Calculate Time History to Yes is unnecessary unless you want to animate the result. Set to Yes to see the time history results, such as for a layer-by-layer animated build. For large models, the calculation may take some time.
Results	Read-only properties displaying the following: **Minimum**: The minimum temperature experienced during the build process. **Maximum**: The maximum temperature experienced during the build process. **Minimum Occurs On**: The geometry body (part or support) on which the minimum temperature occurs. **Maximum Occurs On**: The geometry body (part or support) on which the maximum temperature occurs.

Tree Dependencies

- **Valid Parent Tree Object**: Solution, under Transient Thermal.
- **Valid Child Tree Objects**: None.

Insertion Methods

- Click the **LPBF Hotspot** button in the LPBF Process ribbon.

- Under the Transient Thermal system, right-click **Solution** and then select **Insert** > **LPBF Hotspot**.

Right-click Options

In addition to common right-click options, relevant right-click options for this object include:

- **Evaluate All Results**: Retrieves the temperature data and displays it in graphical form.
- **Clear Generated Data**: Clears evaluated data for this result item.

Additional Related Information

Establish Thermal Analysis Settings (Thermal-Structural System) (p. 113)

Review Results (p. 123)

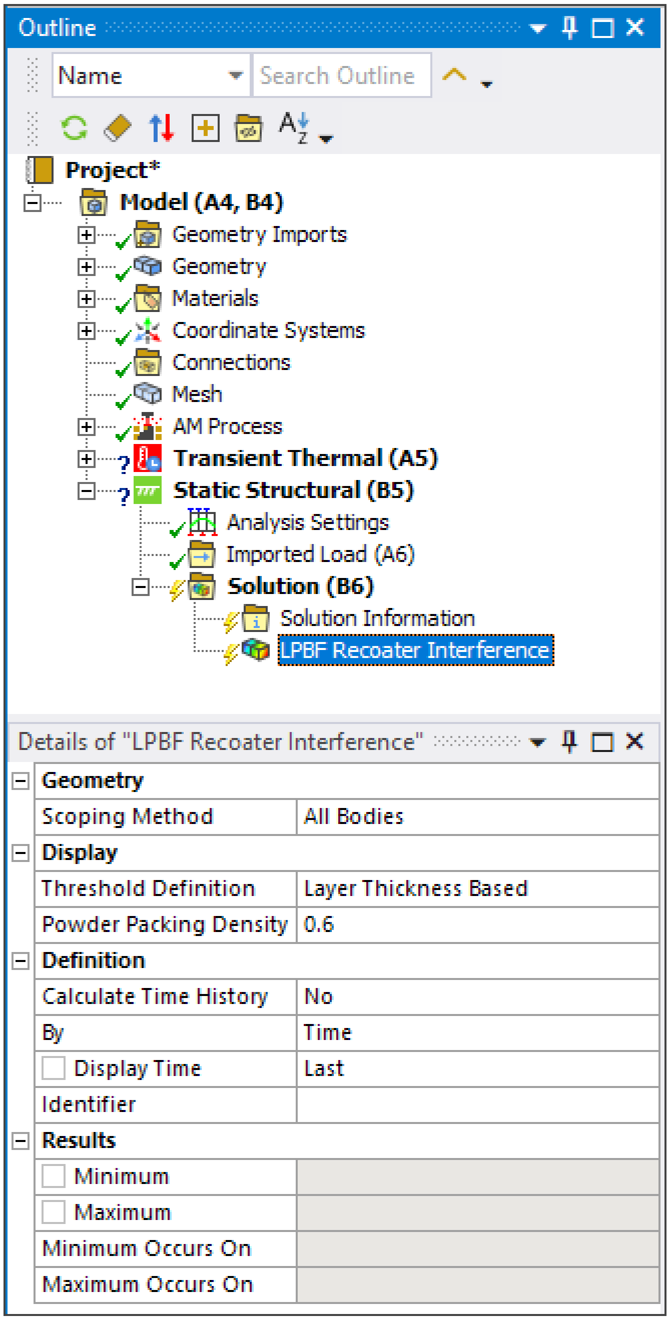
8.7. LPBF Hotspot Time Correction (Beta)

This result helps you define how much time should be spent on each layer to ensure that the build cools down below a target temperature to prevent overheating. Whereas the LPBF Hotspot result (p. 230) identifies problem areas, this result provides a method to resolve the problem by increasing time spent on specific layers.

The hotspot time correction result item is a Beta feature at this release. See Hotspot Time Correction Result (Beta) in the *Additive Manufacturing Beta Features* document.

8.8. LPBF Recoater Interference

Identifies areas that may result in recoater interference issues in an AM Process simulation.



Object Properties

The Details Pane for this object includes the following properties.

Category	Properties/Options/Descriptions
Geometry	**Scoping Method**: Options include Geometry Selection, Named Selection, All Bodies (default), and Material IDs.
Display	**Threshold Definition**: • Layer Thickness Based: The result tool uses the Deposition Thickness (set in the Build Settings (p. 214) object for some simulation types, otherwise

Category	Properties/Options/Descriptions
	set it here) and a new input field, Powder Packing Density, to determine the thresholds for warning and critical deformation. Specifically, if the Z-deformation for any layer is equal to or greater than the Deposition Thickness it is considered a warning. If the Z-deformation is equal to or exceeds the value of Deposition Thickness/Powder Packing Density, it is considered critical. Powder Packing Density may be dependent on the powder particle size, material, spreading mechanism, or other factors • None: The result tool will show pure Z-deformation in the build, where the value at each point corresponds to the Z-deformation when that material was a new layer. • User Defined: You define your own warning and critical thresholds. If None is selected, the display will show the default color range. For Layer Thickness Based and User Defined threshold definitions, the display colors are: • Blue = good • Green = warning • Red = critical **Deposition Thickness**: The thickness of added powder material in every pass of the recoater blade. Specifically, use the amount the base plate drops between layers. **Powder Packing Density**: May be dependent on the powder particle size, material, spreading mechanism, or other factors.
Definition	**Calculate Time History**: No (default) or Yes. • **No** (default): Results are not time dependent. Ensures a quick, dynamic calculation of results. • **Yes**: Setting Calculate Time History to Yes is unnecessary unless you want to animate the result. Set to Yes to see the time history results, such as for a layer-by-layer animated build. For large models, the calculation may take some time.
Results	Read-only properties displaying the following: **Minimum**: The minimum displacement experienced during the build process. **Maximum**: The maximum displacement experienced during the build process. **Minimum Occurs On**: The geometry body on which the minimum displacement occurs. **Maximum Occurs On**: The geometry body on which the maximum displacement occurs.

Tree Dependencies

- **Valid Parent Tree Object**: Solution, under Static Structural.
- **Valid Child Tree Objects**: None.

Insertion Methods

- Click the **LPBF Recoater Interference** button in the LPBF Process ribbon.
- Under the Static Structural system, right-click **Solution** and then select **Insert** > **LPBF Recoater Interference**.

Right-click Options

In addition to common right-click options, relevant right-click options for this object include:

- **Evaluate All Results**: Retrieves the displacement data and displays it in graphical form.
- **Clear Generated Data**: Clears evaluated data for this result item.

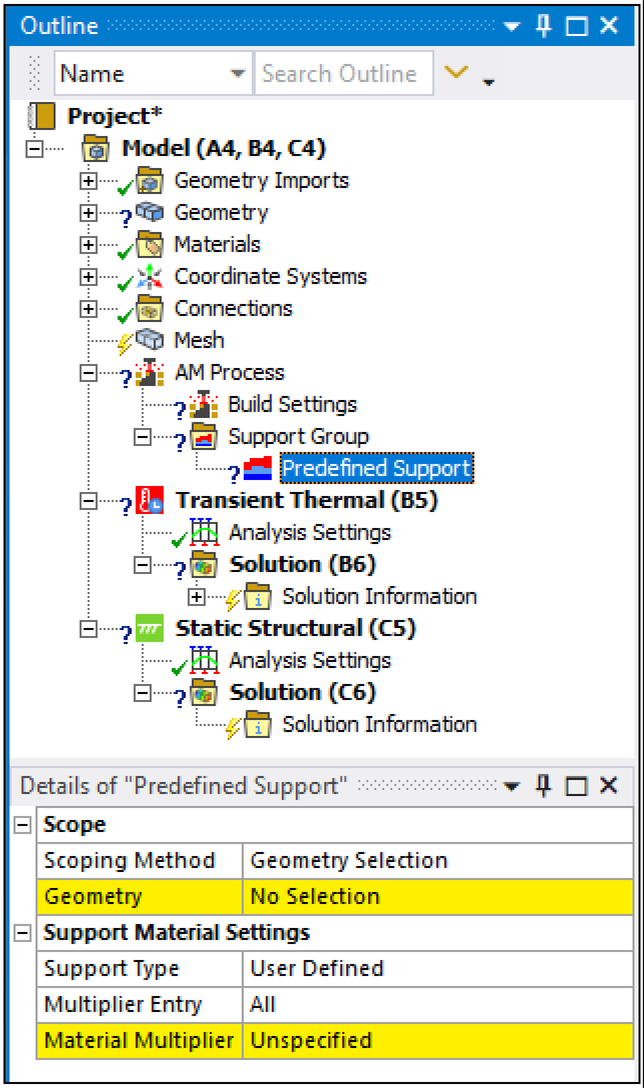
Additional Related Information

Establish Structural Analysis Settings (p. 117)

Review Results (p. 123)

8.9. Predefined Support

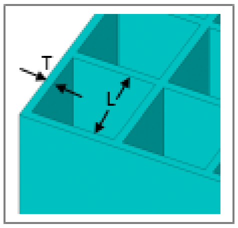
Identifies a support structure that was imported with your part geometry for Additive Manufacturing simulations. Supports are modelled as elements between the **Build Geometry** and the **Base Plate Geometry**.



Object Properties

The Details Pane properties for this object include the following.

Category	Properties/Options/Descriptions
Scope	**Scoping Method**: Options include **Geometry Selection** (default) and **Named Selections**. **Geometry**: Displays when the **Scoping Method** is set to **Geometry Selection**. **Named Selection**: Displays when the **Scoping Method** is set to **Named Selection**. Select a desired Named Selection from the drop-down menu.
Support Material	**Support Type**: Options include **User Defined** (default) and **Block**. When you select the **Block** option, the following properties become the only visible properties. • **Wall Thickness**: Enter a thickness value (illustrated by **T** length below). • **Wall Spacing**: Enter a spacing value (illustrated by **L** length below).

Category	Properties/Options/Descriptions
Set-tings	 **Multiplier Entry**: Options include **All** (default) and **Manual**. **Material Multiplier** **All Setting** When the **Multiplier Entry** property is set to **All**, the **Material Multiplier** property displays. The **Material Multiplier** property applies the same value to all of the material multipliers listed below. The multiplication factors are homogenization factors and, in each direction, reflect the ratio of the support area projected onto the area of a fully solid support. **Manual Setting** When the **Multiplier Entry** property is set to **Manual**, the following multiplier properties display: • **Elastic Modulus Multiple in X/Y/Z** • **Shear Modulus Multiple in XY/YZ/XZ** • **Density Multiple** • **Thermal Conductivity Multiple in X/Y/Z**
Stat-ist-ics	**Volume**: Read-only property that displays the volume of the added finite element body.

Tree Dependencies

- **Valid Parent Tree Object**: Support Group, which is under AM Process object.
- **Valid Child Tree Object**: This object does not support any child objects.

Insertion Methods

- Select the **Support Group** object and then select the **Predefined** option from the **Supports** group of the AM Process Context Tab.
- Right-click the **Support Group** object and then select the **Insert** > **Predefined Support**.

Right-click Options

In addition to common right-click options, relevant right-click options for this object include:

- **Insert**
  - Predefined Support
  - Generated Support
  - STL Support
  - Commands
- **Suppress/Unsuppress**

API Reference

See the Predefined AM Support section of the ACT API Reference Guide for specific scripting information.

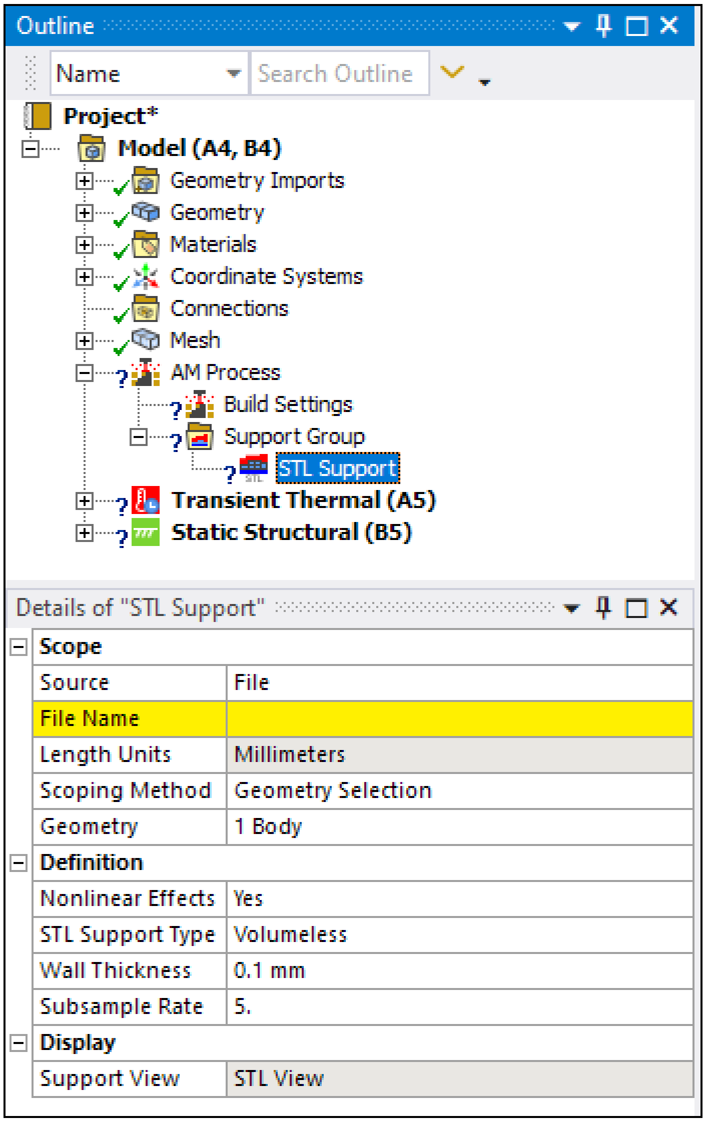
Additional Related Information

- LPBF Simulation Guide (p. 1)
- Identify and/or Generate Supports (p. 86)

8.10. STL Support

Imports and meshes a support structure that is an STL (Stereolithography) file, of either volumeless or solid type. This feature is designed primarily for inserting volumeless (that is, not watertight) supports, such as those created by Additive Prep, into an additive manufacturing model for simulation.

After inserting this object, right-click and select Generate Mesh to mesh the support. The mesh is created using elements that are internally divided into subdivisions to sample the presence of material and calculate the overall density of each element. Each element's density is used, in turn, as a material property knockdown factor.



Object Properties (p. 240)

Tree Dependencies (p. 241)

Insertion Methods (p. 242)

Right-click Options (p. 242)

API Reference (p. 242)

Additional Related Information (p. 242)

Object Properties

The Details Pane properties for this object include the following.

Category	Properties/Options/Descriptions
Scope	**Source**: File or STL. If you choose File, enter the File Name in the File Name property. If you choose STL, an STL dropdown menu appears where you select from previously imported Construction Geometry STLs. **File Name**: Navigate to the appropriate folder and select your STL support file. **Length Units**: A drop-down menu with units. Choose the units used in the support file. Millimeters is the default setting. If there are multiple bodies in the model, use the next two fields to designate which body is associated with the STL support.

Category	Properties/Options/Descriptions
	Scoping Method: Either Geometry Selection or Named Selection. **Geometry/Named Selection**: Select the body (or Named Selection) that is associated with the STL support.
Defin-ition	**Nonlinear Effects**: Yes (default) or No. This property applies nonlinear effects to the finite element body. **STL Support Type**: The type of support in the STL file. Options include: Volumeless: These are usually thin, single-bead width support walls that are not watertight, such as lattice or tree-type supports, or Block/Heartcell/Rod/Line supports from Additive Prep. Other names in the industry for this type include thin wall, vector, and facet. Solid: These supports are standard, watertight geometry bodies. Custom supports from Additive Prep are in this category. Other names in the industry for this type include thick wall, bulk, and volume. **Wall Thickness**: Wall Thickness value is the thickness of a single-bead width laser scan set by your machine. Available only if STL Support Type = Volumeless. **Subsample Rate**: Subsample Rate value. Each element is divided into sampling regions to determine an overall element density used as a material knockdown factor within that element. A Subsample Rate of 5 (default) = 5 x 5 x 5 = 125 subdivisions. Subsample Rate affects the accuracy of element density. **Element Size**: Read-only indication of the element size, which is taken from the part's mesh size criteria. Visible only after mesh generation.
Dis-play	**Support View**: A drop-down menu with the following options: **STL View**: Displays the STL support. **Mesh View**: Displays the elements associated with the STL support. Visible only after mesh generation. **Knockdown Factors**: Displays the element densities associated with the STL support. Visible only after mesh generation.
Stat-istics	Read-only indications only visible once you have generated the mesh for the support. **Nodes**: The number of nodes generated for the support. **Elements**: The number of elements generated for the support. **Volume**: The volume of all the elements generated for the support.

Tree Dependencies

- **Valid Parent Tree Object**: Support Group, which is under AM Process object.
- **Valid Child Tree Objects**: None.

Insertion Methods

- Select the **AM Process** object and then select the **STL Support** option from the AM Process Context Tab.
- Right-click the **AM Process** object and then select the **Insert** > **STL Support**.
- Select the **Support Group** object and then select the **STL Support** option from the AM Process Context Tab.
- Right-click the **Support Group** object and then select the **Insert** > **STL Support**.

Right-click Options

In addition to common right-click options, relevant right-click options for this object include:

- **Insert**
  - Predefined Support
  - Generated Support
  - STL Support
  - Commands
- **Generate Mesh**: Generates a mesh of elements for the support using Subsample Rate to determine knockdown factors.
- **Create Named Selection of Generated Elements**: Creates a Named Selection consisting of the generated elements for the support.
- **Create Named Selection of External Element Faces**: Creates a Named Selection consisting of the faces of the generated elements for the support.
- **Suppress**: Suppresses the STL support from the simulation.
- **Clear Generated Data**: Clears any generated elements on the supports.

API Reference

See the SLT AM Support section of the ACT API Reference Guide for specific scripting information.

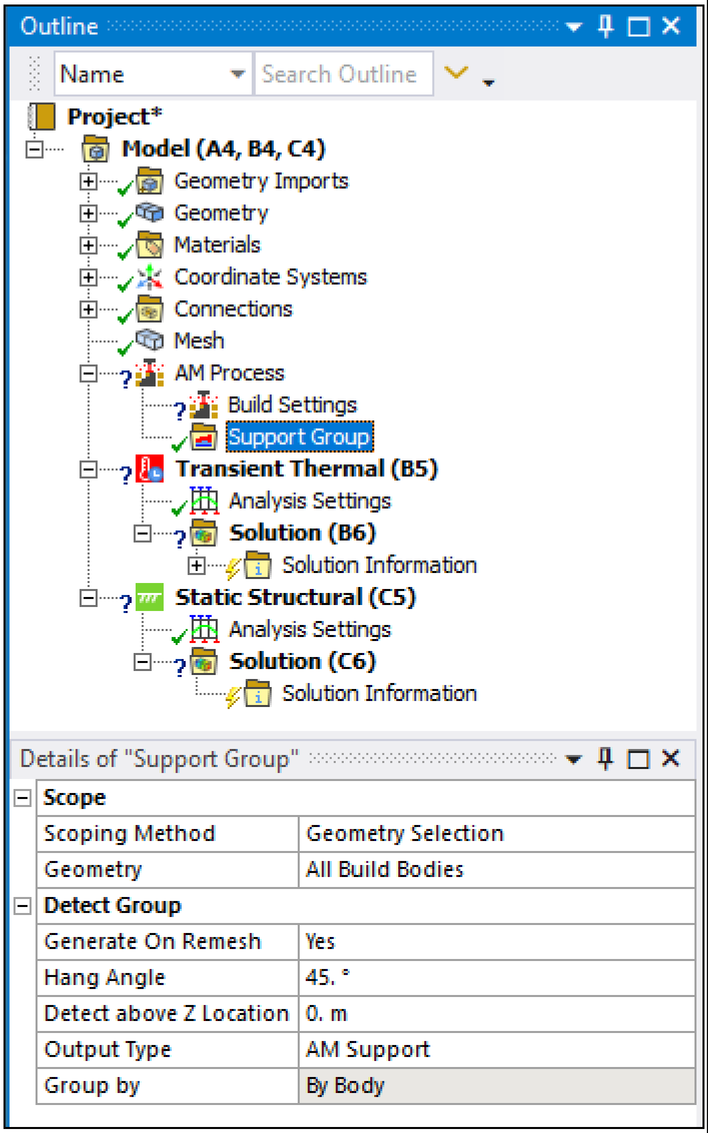
Additional Related Information

LPBF Simulation Guide (p. 1)

Identify and/or Generate Supports (p. 86)

8.11. Support Group

Groups **Predefined Support** and/or **Generated Support** objects in an AM process simulation. Also use this object to detect element faces for a Generated Support.



Object Properties (p. 243)
Tree Dependencies (p. 244)
Insertion Methods (p. 244)
Right-click Options (p. 244)
Additional Related Information (p. 244)

Object Properties

The Details Pane properties for this object include the following.

Category	Properties/Options/Descriptions
Scope	**Scoping Method**: Options include **Geometry Selection** (default) and **Named Selections**. **Geometry**: Displays when the **Scoping Method** is set to **Geometry Selection**. **Named Selection**: Displays when the **Scoping Method** is set to **Named Selection**. Select a desired Named Selection from the drop-down menu.

Category	Properties/Options/Descriptions
De-tect Group	The **Detect Group** category includes the properties listed below. You use these properties to automatically detect element faces to be applied to **Generated Supports**. See Identify and/or Generate Supports (p. 86) for more information. **Generate On Remesh** **Overhang Angle** **Detect Above Z Location** **Output Type** **Group By**

Tree Dependencies

- **Valid Parent Tree Object**: AM Process.
- **Valid Child Tree Objects**: Predefined Support, Generated Support, and STL Support.

Insertion Methods

- Select the **AM Process** object and then select the **Support Group** option from the **Supports** group on the AM Process Context tab.
- Right-click the **AM Process** object and then select the **Insert** > **Support Group**.

Right-click Options

In addition to common right-click options, relevant right-click options for this object include:

- **Insert**
  - Predefined Support
  - Generated Support
  - STL Support
  - Commands
- **Detect Supports**: Detect element faces to be applied to Generated Supports.
- **Detect and Generate Supports**: Detect element faces and generate supports for the build bodies.

Additional Related Information

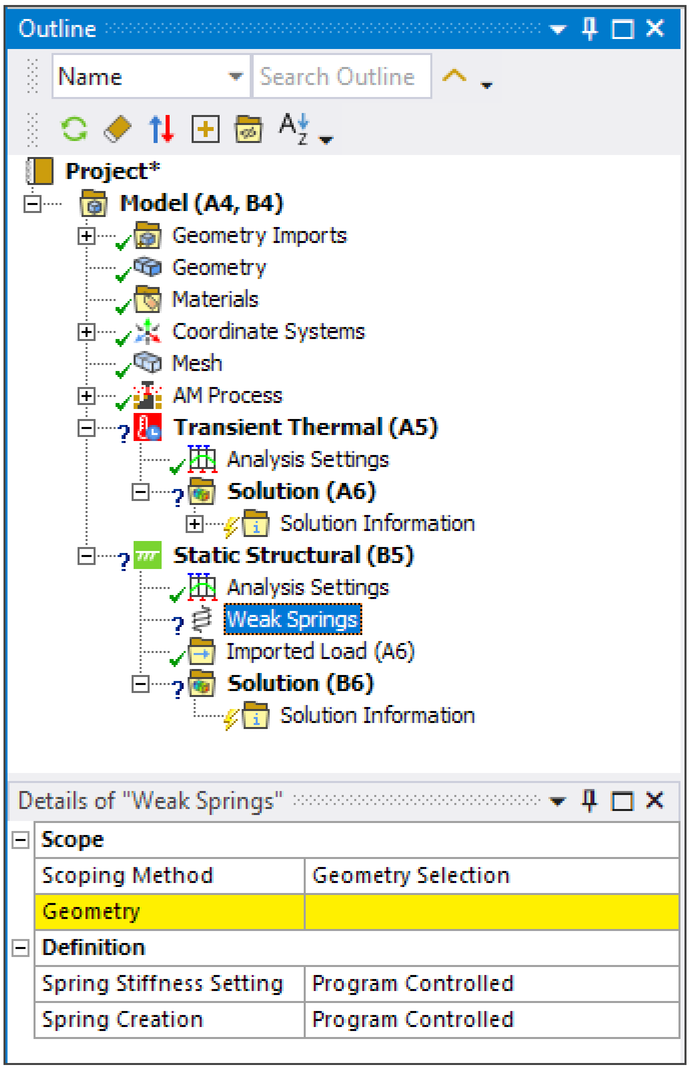
LPBF Simulation Guide (p. 1)

Identify and/or Generate Supports (p. 86)

8.12. Weak Springs

Loosely holds parts in place during AM postprocess steps to prevent rigid body motion.

Insert a Weak Springs object into an LPBF Static Structural simulation to prevent rigid body motion when the default nonlinear stabilization is insufficient. In most cases, simulations should be able to reach convergence without this object, but it may be necessary on some occasions. When added, the object automatically scopes all nodes at the bottom of the part and support geometries. It attaches weak springs on the scoping (or a subset of the scoping) to hold the part in place without significantly affecting results.



Object Properties (p. 245)

Tree Dependencies (p. 247)

Insertion Methods (p. 247)

Additional Related Information (p. 247)

Object Properties

The Details Pane for this object includes the following properties.

<table>
<tr><th>Category</th><th>Properties/Options/Descriptions</th></tr>
<tr><td>Scope</td><td>Scoping Method: Options include Geometry Selection (default) and Named Selections.
Geometry: Select either Faces or Mesh Nodes of the object. If the Grid option is chosen for Spring Creation the scoped nodes or faces need to be on a consistent Z-plane. When added, the object will automatically scope all nodes in the build geometries (part and support) at the interface with the base.
Named Selection: Select a Named Selection from the drop-down menu.</td></tr>
<tr><td>Definition</td><td>Spring Stiffness Setting: Choose Program Controlled, Factor, or Manual.
Program Controlled: Equivalent to using the Factor setting with a value of 1e-7.
Factor:
Spring Stiffness Factor: A factor used to indirectly specify the spring stiffness. Defaults to 1e-7. With this option,
$\text{Spring Stiffness} = Factor * E * \sqrt{Area}$, where Area is the area represented by the scoped nodes, E is the elastic modulus, and Factor is the Spring Stiffness Factor.

The area and elastic modulus in this equation are determined by Mechanical APDL whereas the Spring Stiffness Factor is user input. The elastic modulus is taken at either the room temperature specified in the AM Process object (for Thermal-Structural simulations), or the reference temperature of the body it's associated with (for Inherent Strain simulations).
Manual:
Spring Stiffness: Stiffness value to be used by each spring.
Spring Creation: Choose Program Controlled, Grid, All Scoped Nodes, or All Scoped Corner Nodes.
Program Controlled: When scoped faces or nodes are all on the same Z-plane, the application uses the same as using the Grid option where the grid length is the total_element_volume/total_area where the area is the area associated with each scoped node and the volume is the element volume of elements associated with the scoped nodes.

When scoped faces or nodes are not on the same Z-plane, the application defaults to All Scoped Corner Nodes.
Grid: When the Grid setting is selected, an X-Y grid of candidate points is created internally. For each point, the nearest corner node is identified, and if the node is within the Grid Spacing distance of the candidate grid point, weak springs are applied to it. This option is useful when a layered</td></tr>
</table>

Category	Properties/Options/Descriptions
	tetrahedral mesh is used, to provide a more uniform overall distance between weak springs when there may be smaller elements along part curves. The Grid option is only valid if the face or node scoping is in a consistent Z-plane. – Grid Spacing: Spacing used to identify weak spring locations. • **All Scoped Nodes**: Weak springs are placed on all scoped nodes or nodes associated with face scoping. • **All Scoped Corner Nodes**: Weak springs are placed on all scoped corner nodes or all corner nodes associated with scoped face(s). Corner nodes are those at the corners of an element (not the midside of an element edge).

Tree Dependencies

- **Valid Parent Tree Object**: Static Structural
- **Valid Child Tree Objects**: None.

Insertion Methods

- Click the **Weak Springs** button in the LPBF Process ribbon.
- Right-click **Static Structural** and then select **Insert** > **Weak Springs**.

Additional Related Information

Apply Structural Boundary Conditions (p. 120)