ANSYS Additive User's Guide

Applies to:

- ANSYS Additive Print
- ANSYS Additive Suite

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

Release 19.2
September 4, 2018

Published in the U.S.A.

Table of Contents

Welcome to ANSYS Additive

ANSYS Additive is a simulation tool dedicated to the field of additive manufacturing. Offerings include Additive Desktop, a standalone application, and Additive Cloud, the same application deployed in a cloud environment. The capabilities are the same, with just a few minor exceptions as noted.

Why Use ANSYS Additive?

Companies today are designing products that are sophisticated, complex, and finely-tuned to operate in their working environments. With the use of CAE and FEA tools, designers are aggressively reducing their product's time-to-market, cost, and material consumption. ANSYS Additive harnesses the next level of design improvement by providing tools to simulate a part's behavior *during the manufacturing process* for those using the latest techniques in **additive manufacturing** (3D printing). Already shown to be a revolutionary technology with immense advantages over traditional manufacturing methods, additive manufacturing brings unique challenges as well as opportunities for even more time and cost savings.

The Additive application simulates the layer by layer build process of metal parts undergoing Laser Powder Bed Fusion (LPBF), a type of additive manufacturing that uses a laser to melt or fuse material powder together. As each layer is processed, the region under the laser experiences extremely intense, local heat that cools rapidly and results in thermal distortion. The simulation method uses a layer by layer accumulation of inherent strain to predict this distortion. As a user, you will gain critical insight into the complex physics-based phenomena associated with this layering process.

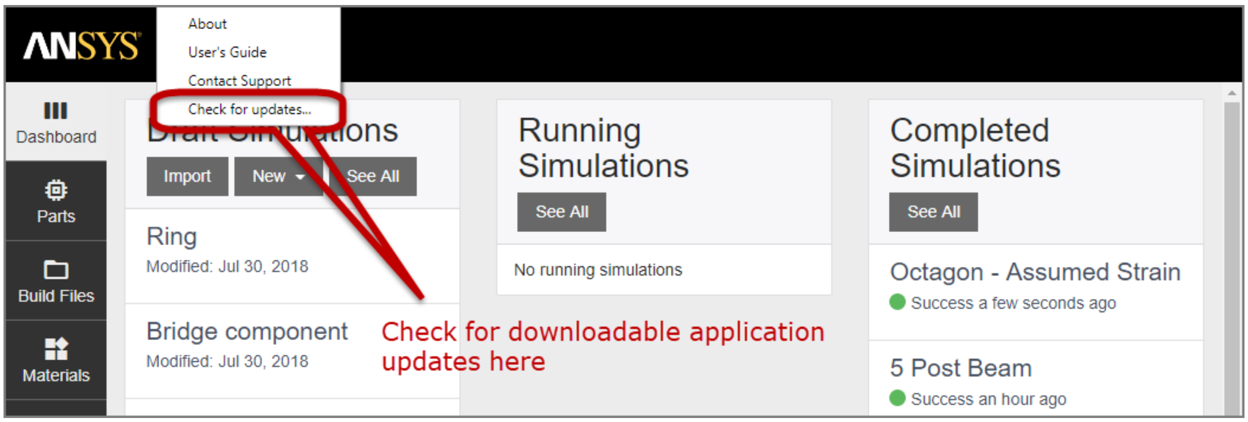
Simulating the build process may be performed at various points in the overall design/manufacturing process depending on your goals. Additive generates practical solutions to residual stress, distortion, and build failure, enabling you to:

- Improve Product Design – In complicated, asymmetrical parts, shrinkage and distortion due to rapid heating and cooling during the 3D printing process may result in components outside of tolerances. Simulations of the build process show magnitudes and locations of part distortion. Designers can quickly make design changes to assure part conformance without iterations of trial and error builds.
- Inform Build Preparation – Instead of building several part prototypes in different orientations on the build plate, simulations of these orientations reveal best orientation in a fraction of the time and expense.
- Validate Build Preparation – Parts designed using powerful topology optimization tools result in complex and intricate shapes that present difficult challenges on where to place supports required for build. Simulations in Additive include the generation of optimized support structures using predicted residual stress accumulation as criteria for support placement and thickness. Engineers can use the optimized support information in their production builds to reduce build failures due to insufficient supports.

Simulations accurately predict part distortion during the build. Additive's Distortion Compensation feature takes that information a step further and automatically creates a distortion compensated geometry, essentially reversing distortion effects. Engineers can use the compensated geometry file in their production builds and be assured of a final part that conforms to design intent.

Look for Frequent Updates

As is everything associated with the additive manufacturing industry, we are moving *fast*. We want to provide you with rolling updates of the ANSYS Additive software on a more frequent basis than what you might be used to. In Additive Desktop, you can look for updates under Help → Check for updates.



Our Additive Cloud application, updated more frequently still, may offer Beta features not included in Additive Desktop. Documentation on these Beta features, if available, is located here.

Finally, check here for the latest updates to this User's Guide.

The Simulation Process

A simulation in Additive consists of four steps:

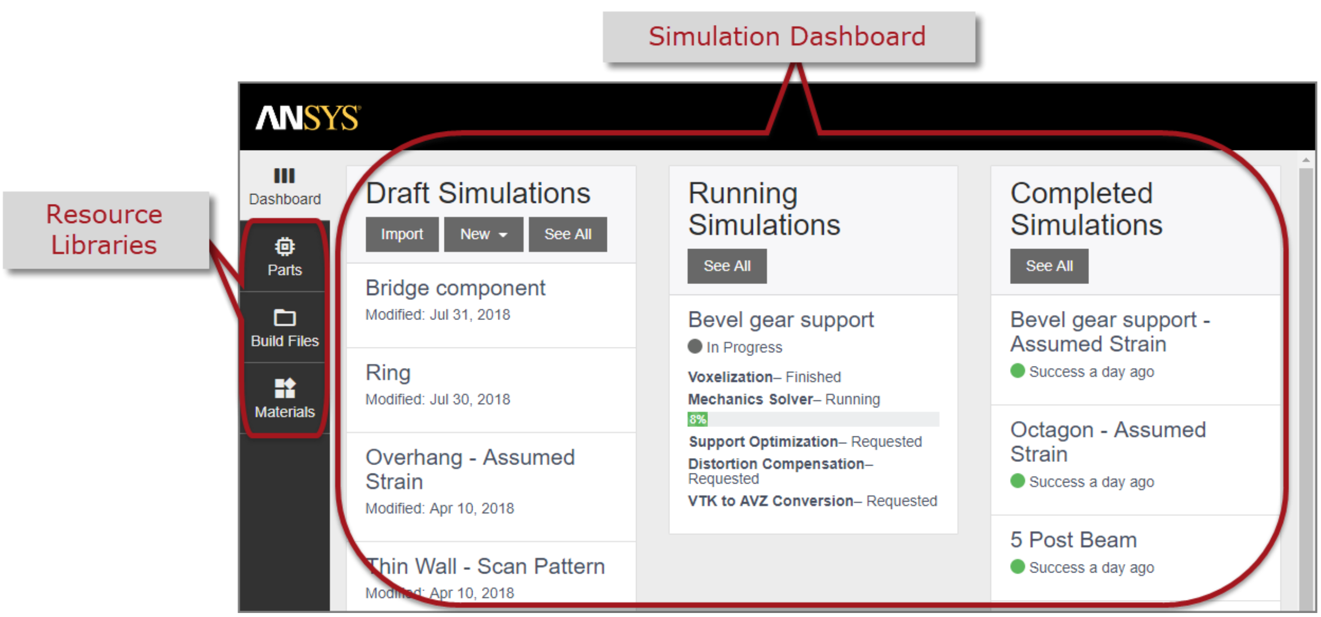
1. Prepare and Upload a Part
2. Set Up a Simulation
3. Run a Simulation
4. Review Results of a Simulation

Depending on your simulation goals, you may need to run multiple iterations of this four-step process.

Also, before beginning a simulation for the first time, you should run a series of calibrations to determine input factors that take unique aspects of your machine and material into consideration. The calibration procedures and parts are available online here. Note that the calibration procedure may continue to evolve as we work with more machine partners and key customers.

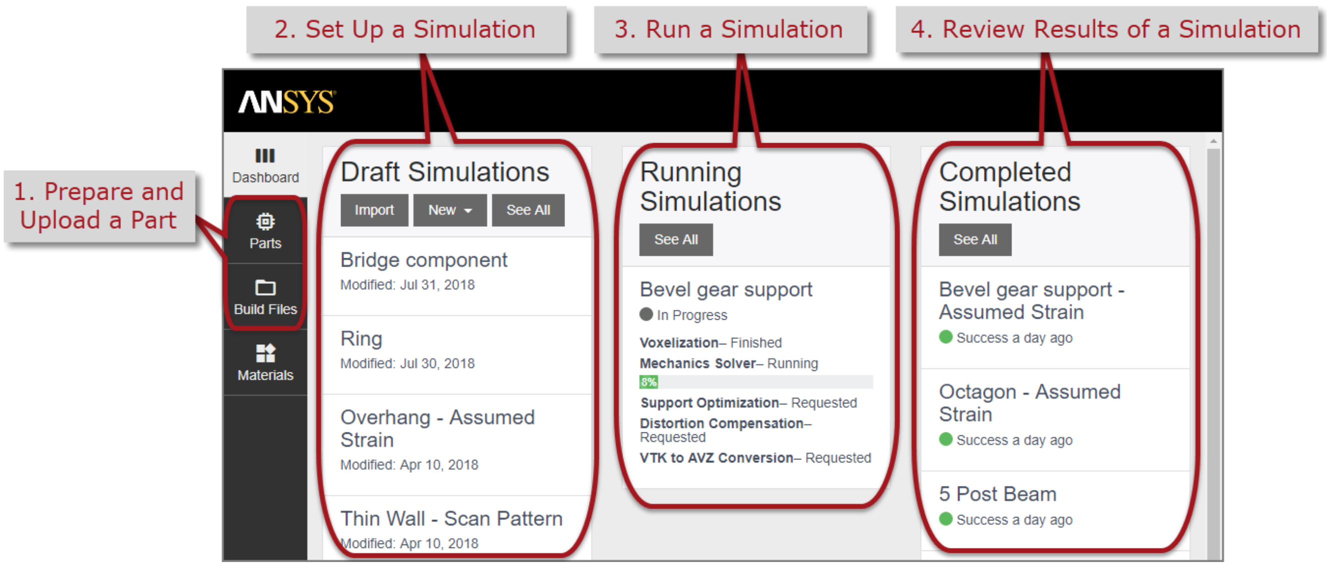
Understanding the Additive Interface

The Additive interface is quite simple and straightforward. It includes a simulation dashboard in the main portion of the window and resource libraries on the left. Clicking on the ANSYS logo (ANSYS) from anywhere in the program takes you back to the simulation dashboard.



Resource libraries are repositories for parts, build files, and materials. The first step in the simulation process, prepare and upload a part, involves uploading a part into the Parts Library (or the Build File Library if you have a build file).

The dashboard shows your most recent simulations organized from left to right as Draft Simulations, Running Simulations, and Completed Simulations. This structure parallels the next steps in the simulation process; set up a simulation, run a simulation, and review results of a simulation. Draft simulations are simply saved simulation forms that have not been run as simulations yet.



Step 1. Prepare and Upload a Part

A typical workflow begins with preparing a part for uploading to the Additive application. From within a CAD program, such as ANSYS SpaceClaim, you'll need to export an .stl file. The standard file format for most rapid prototyping and 3D printing applications, an .stl file (from stereolithography) is a tessellated representation of a 3D object that consists of triangle elements that define the external surface of the object. Rather than being a volume representation, surface normal vectors define the inside versus the outside of the object.

Guidelines for Part Orientation and Resolution

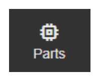
There are some important considerations to keep in mind before you export your geometry.

- Dimensions of the part must be in units of millimeters (mm). While .stl files are unitless, the Additive application does not provide the ability to switch unit systems and Metric units of millimeters are assumed.
- Currently, part size is limited to 600 millimeters in all directions (i.e., the maximum part is 600 x 600 x 600 mm).
- The .stl file must have the part positioned in the orientation in which it will be printed.
- A part with its longest dimension in the Z direction will require the longest simulation time. While the domain volume doesn't change with pFart orientation, the number of voxel *layers* changes, which means more calculations in the solver are being performed. Alternatively, a part with the smallest dimension in the Z direction will have the shortest simulation time.

(Note that parts oriented with their longest dimension in the Z direction will similarly take longer to print on the machine because more layers are required. Powder recoating/spreading time is an order of magnitude higher than laser printing time.)

- The time required to slice and voxelize an .stl file exponentially increases with the number of triangles. (Slicing, as used here, refers to the internal process of dividing a part into scan vectors according to the scan pattern input parameters that will be used in the 3D build process. Voxelization refers to the dividing of a part into voxels, or elements, used in the mathematical simulation. See Voxel Size.) Smaller .stl files can be processed much faster. Given the fidelity of voxelization and slicing, the level of detail included in large .stl files is not needed. If an .stl file is over 40,000 triangles, it will be reduced on upload. Triangle reduction works by calculating the collapse cost of each triangle by looking at the largest difference between normal vectors of its surrounding triangles. For example, if a triangle is basically co-planar with its surrounding triangles then the cost or visual impact of removing that triangle is very low. Triangles with the least collapse cost are removed until the desired threshold is met. There is also a maximum allowed collapse cost threshold to prevent the part from collapsing onto itself resulting in an invalid .stl file. Because of this threshold, the final part might have more than 40,000 triangles. Reducing the fidelity of the .stl file has little to no effect on simulation results because of the fidelity required by slicing and voxelization.

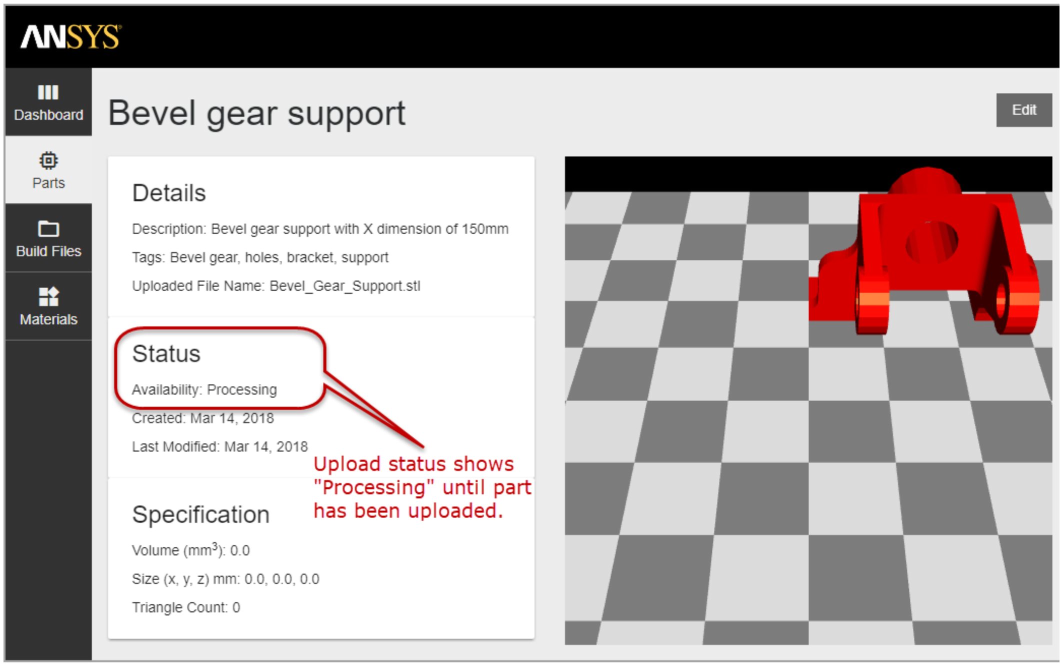
Uploading a New Part

To upload a new part to the Parts Library, click on the 

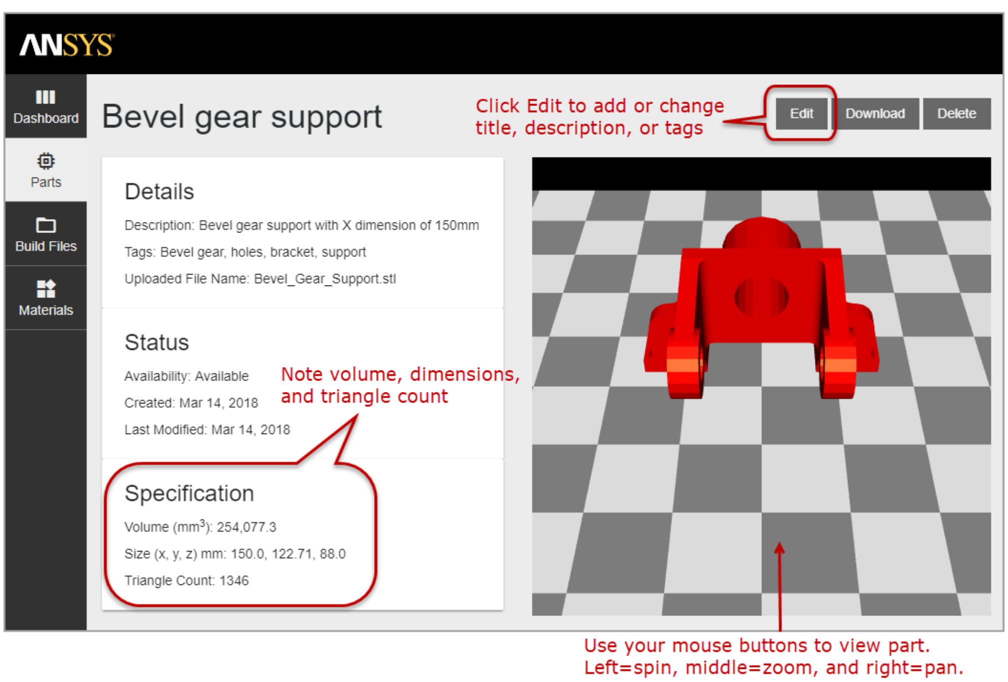
button in the left panel of the dashboard and then click on Upload Part. Part files are .stl files, either ascii or binary. While .stl files are unitless, dimensions of the part in Additive are assumed to be in units of millimeters (mm). The maximum file size is 100 MB.

The name, tags, and description fields allow you to identify the part in some way that makes logical sense to you. These fields are searchable at any location in the program that allows for searching on text fields. Tags should be at least three characters long.

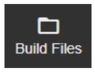
It may take a few minutes to upload a part, depending on the size of the file. The status indication in the Parts Library shows “Processing” while the part is uploading and “Available” when uploading is complete. A common practice is to upload the part and then begin setting up your simulation. When you get to the step of selecting geometry on the simulation form, the part is usually available for selection.



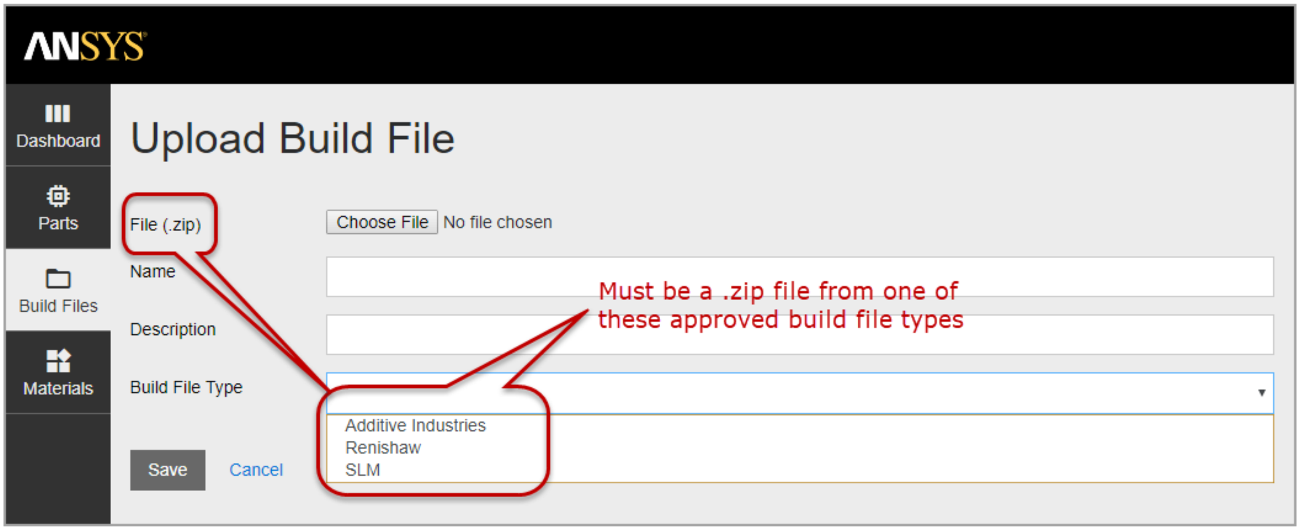
If you want to see details about a part that has already been uploaded, you may view it in the Parts Library by clicking on it or searching on any text term used in the name, tags, or description of the part. Click on the part name to bring up a summary of information and image preview of that part. Use your mouse buttons to move the part around in the image preview; left button for spin, middle for zoom, and right button for pan. Click on Edit to edit the name, tags, or description. You cannot edit the features of the geometry itself. (Note that if the original .stl file is changed, it does not affect the uploaded part. A new copy is made inside the application that is not linked to the original file.)



Uploading a Build File

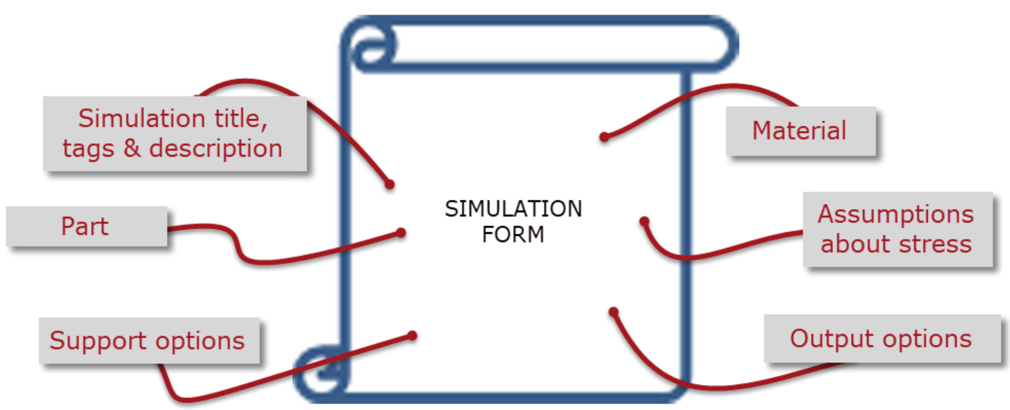
Build files are unique to each 3D printing machine and are required to be .zip files containing the part .stl file as well as files specifying machine scan vectors. To upload a .zip file to the Build Files Library, click on the 

button in the left panel of the dashboard and then click on Upload Build File. When you select Build File Type, you are specifying which of the approved translators Additive will use when reading the data. Note that we may add additional options as we continue to work with more machine partners.



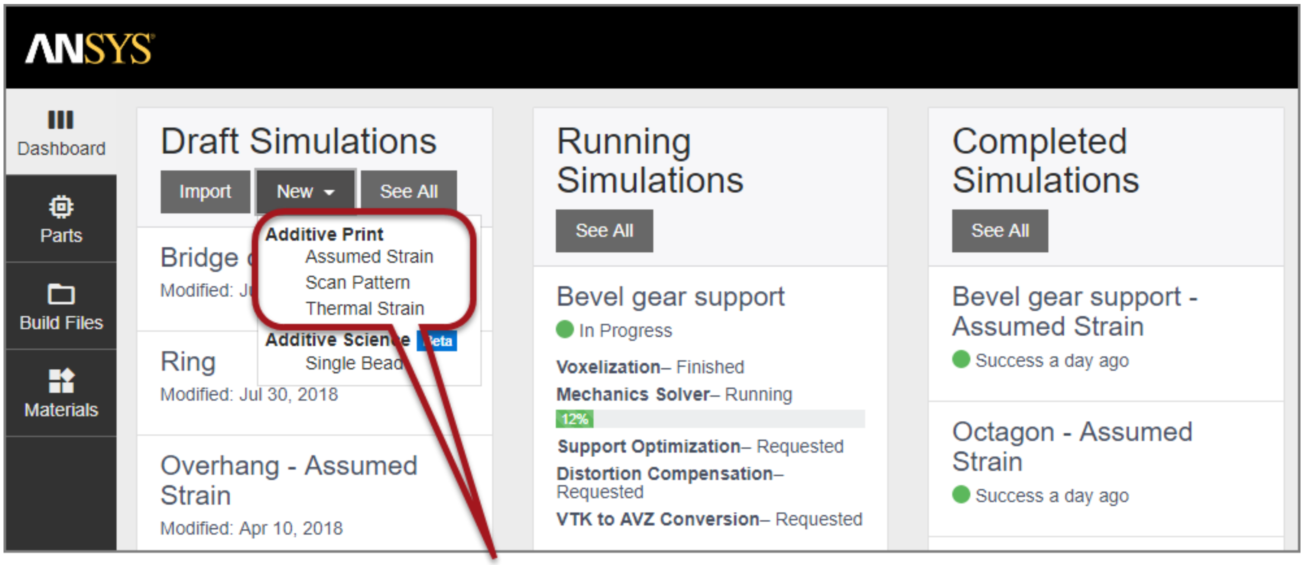
Step 2. Set Up a Simulation

Once you have added a part into the Parts Library (or a build file into the Build File Library), you are ready to begin setting up a simulation. You will use a simulation form to specify the criteria necessary for a simulation, including the part and its material and stress behavior, support options, and the desired output options of your simulation.



New Simulation

From the dashboard, click on the New dropdown box to choose a new form corresponding to a simulation type. There are three types of simulations available in Additive Print; Assumed Strain Simulation, Scan Pattern Simulation, and Thermal Strain Simulation. The simulation types (sometimes called strain modes) specify the different ways inherent strain is calculated as an input to the mechanics solver. All three strain modes offer the same simulation output options.



Choose one of three simulation types

Assumed Strain - Uniform Isotropic

Assumed strain mode is the fastest simulation type available. It assumes that a constant isotropic strain occurs at every location within a part as it is being built. 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/material/strain/stress mode combination of interest. Calibration procedures are available online here.

Scan Pattern Strain - Anisotropic

This strain mode uses the same average strain magnitude as assumed uniform strain, but it subdivides that strain into anisotropic components based on the local orientation of scan vectors within the part. This strain mode requires the creation of scan vectors using user-provided scan settings or by reading scan vectors from a machine's build file. This extra step results in a small, increased amount of simulation time compared to assumed uniform strain. For parts where the scan pattern is randomized, scan pattern and assumed strain should give a similar answer. For parts where the scan patterns are aligned, scan pattern strain will result in a more accurate prediction. As in an Assumed Strain Simulation, you will need to calibrate for Strain Scaling Factor.

Thermal Strain - Anisotropic

This strain mode provides the highest degree of accuracy by predicting how thermal cycling affects strain accumulation at each location within a part. A "thermal ratcheting" algorithm assigns a base strain to each location within the part as it solidifies. Each time a location within the part is heated above a temperature threshold (~ 0.4 of its absolute melting temperature) an increase in strain in that location occurs. If a location re-melts, the strain is reset to the base strain. The more times a location is heated above the threshold without melting, the higher the strain accumulates. Once the strain magnitude is calculated for each location within a part using the thermal ratcheting algorithm, that strain is passed to the mechanics solver and applied as an anisotropic strain based upon both local strain magnitude and local scan orientation. Because thermal strain requires a thermal prediction for every scan vector, this strain mode requires a much longer computational time. As in Assumed Strain and Scan Pattern Simulations, you will need to calibrate for Strain Scaling Factor.

Performing an Assumed Strain Simulation

This is the simplest and fastest simulation type. The following steps described for an Assumed Strain Simulation are also required for the other simulation types.

Set Details

Details include Simulation Title, Tags, Description, and Number of Cores.

Simulation Title, Tags, and Description

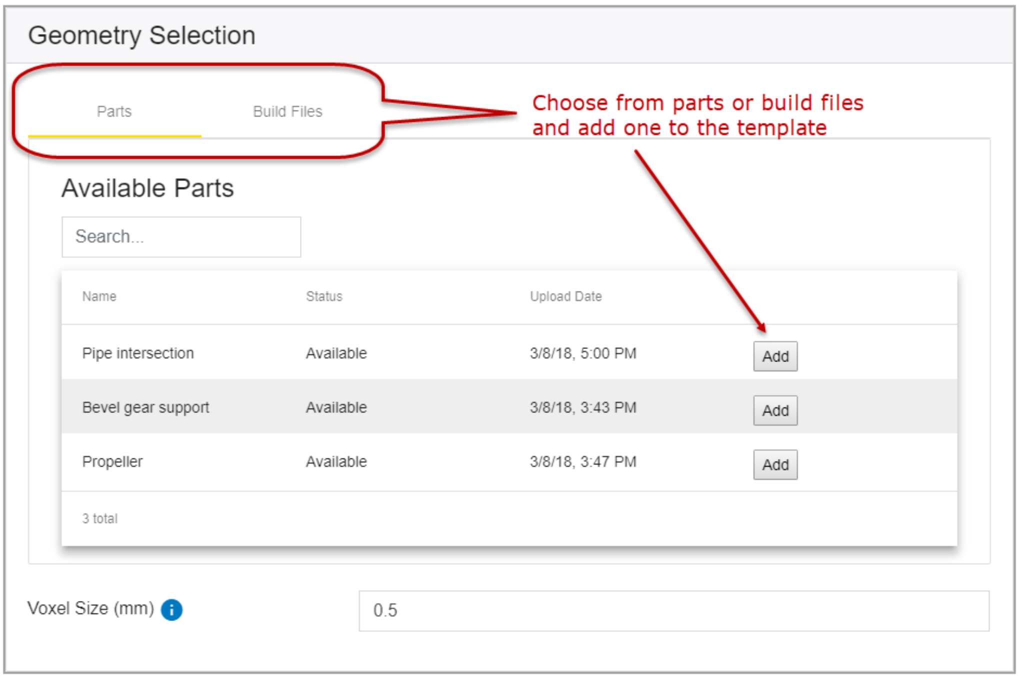
The title (required), tags, and description fields allow you to identify the simulation in some way that makes logical sense to you. These fields are searchable at any location in the program that allows for searching on text fields. Tags should be at least three characters long.

Number of Cores

To take advantage of High Performance Computing, the Additive Desktop application allows you to specify multiple processor cores. Depending on your Additive license, you may have up to 12 cores to use. The default is 4.

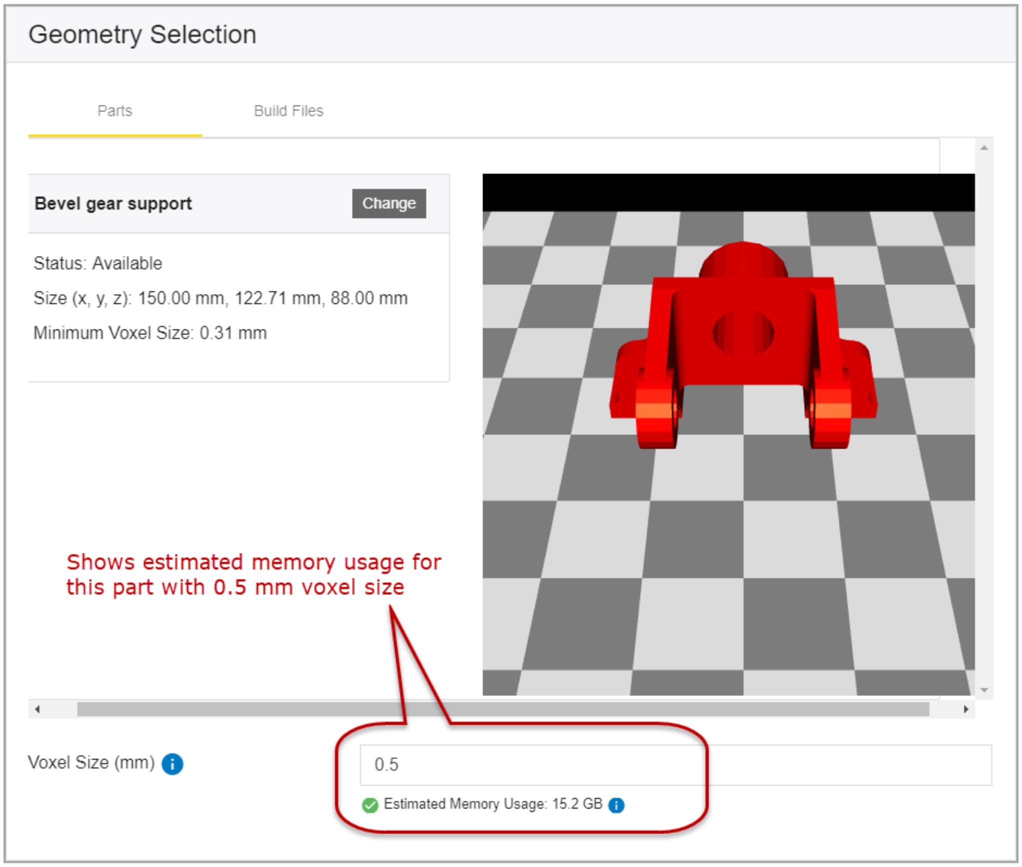
Select Geometry

You select a part for simulation by adding it to your simulation form. Regardless of whether you add a part or a build file, it must have been uploaded first to the Parts Library or Build File Library, respectively.

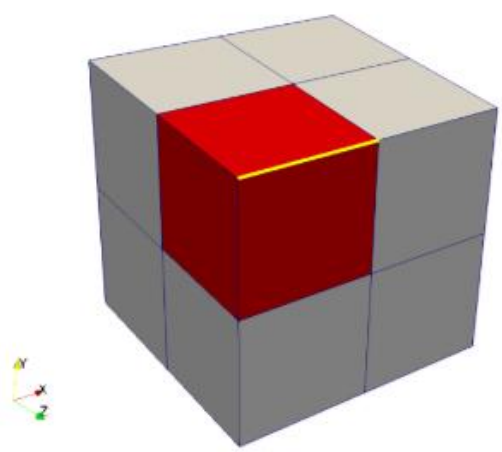


Voxel Size

Upon adding a part to the simulation form, you will see a preview of that part as well as a summary of the part's overall dimensions in millimeters in x, y, and z coordinates, a minimum voxel size recommendation, and an estimate for memory usage.



You will need to specify a voxel size to be used for your simulation. A voxel is a hexahedral (cubic) element used in the finite element method. In the following figure of a cubic part, a voxel is shown in red. There are eight voxels in the cube. Voxel size is the length of the yellow line.



When combined, voxels define the domain of the geometry. Minimum voxel size is the estimated voxel size that can safely run without the simulation risking failure due to insufficient memory. This is calculated automatically when the part is uploaded. Voxel size defaults to 0.5 mm.

How Do I Determine Voxel Size?

Currently, the calculated Minimum Voxel Size assumes a Minimum Overhang Angle of 45 degrees and a Minimum Support Height of zero. (See Define Support Options for further details about supports.) A smaller angle could result in fewer support voxels and a Minimum Support Height > 0 will include more support voxels. In both cases, memory requirements will change, and a larger voxel size may be required for a successful simulation.

Generally, there should be at least four voxels through the thickness of the finest feature of interest. This is accomplished by setting the voxel size to one-fourth the minimum feature dimension. It should be noted however, that for a geometry with highly disproportionate overall dimensions compared to its finest features, some accuracy may be sacrificed in the fine features to obtain a shorter run time by applying the above rule to a thicker area of the part.

How Does Voxel Size Affect Run Time?

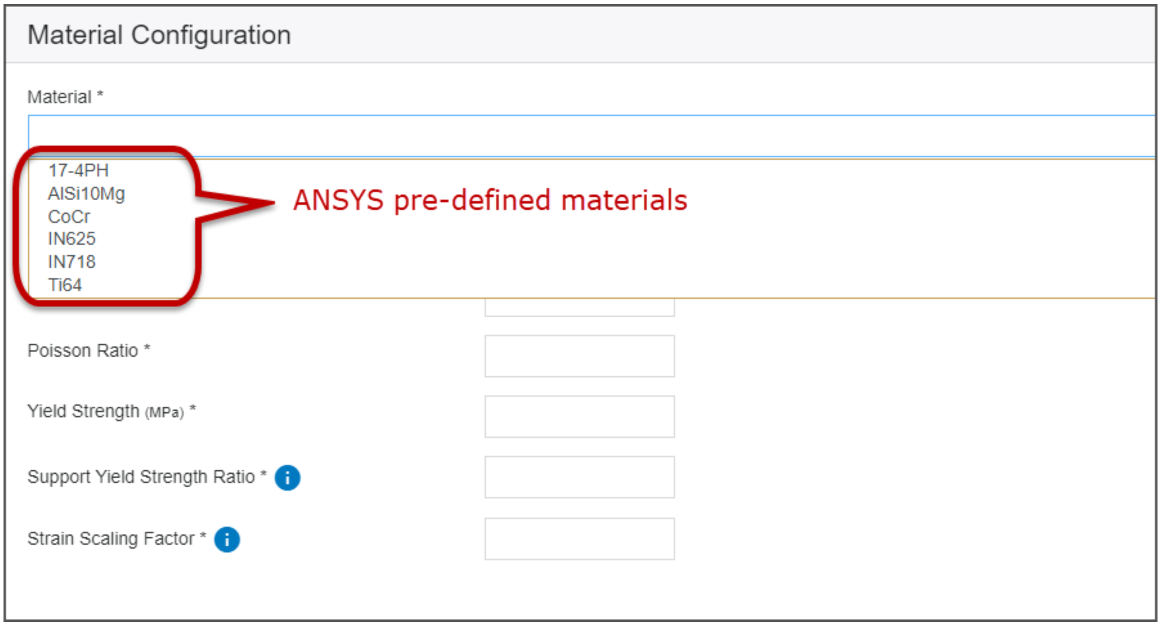
Decreasing voxel size by any factor can exponentially increase the solution run time by that factor raised to the power of 4 (assuming a uniform part, like a cube).

As an example, if the starting voxel size takes 5 minutes of run time, and then the voxel size is reduced by a factor of two, the run time could be expected to increase to approximately $5*(2)^4 = 80$ minutes.

Currently, voxel size is limited to between 0.02mm and 2mm. However, due to memory requirements of smaller voxel size simulations, this range of voxel size is not guaranteed to finish the simulation.

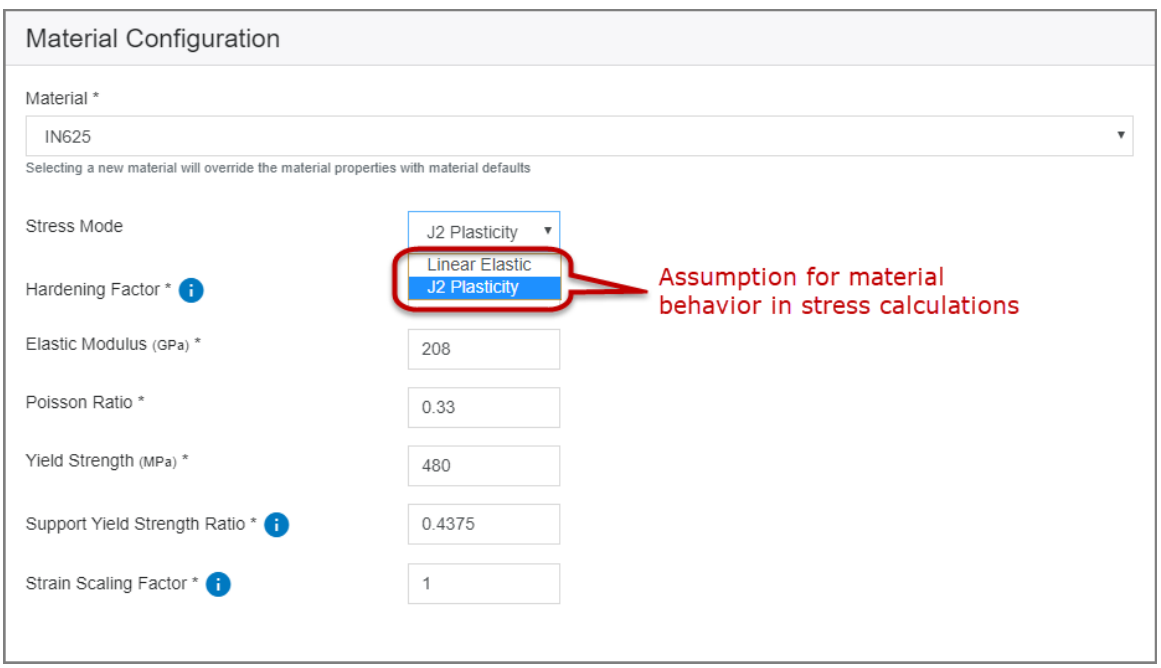
Select Material

You may choose from standard ANSYS pre-defined materials or you may customize your own material. Upon selecting a material from the drop-down box, the properties of Elastic Modulus (in GPa), Poisson Ratio, and Yield Strength (in MPa) for that material automatically populate the fields on the form. These values are for materials at room temperature. To see the other properties associated with a given material, or to customize a material, you will need to bring up the Materials Library. See Customizing a Material.

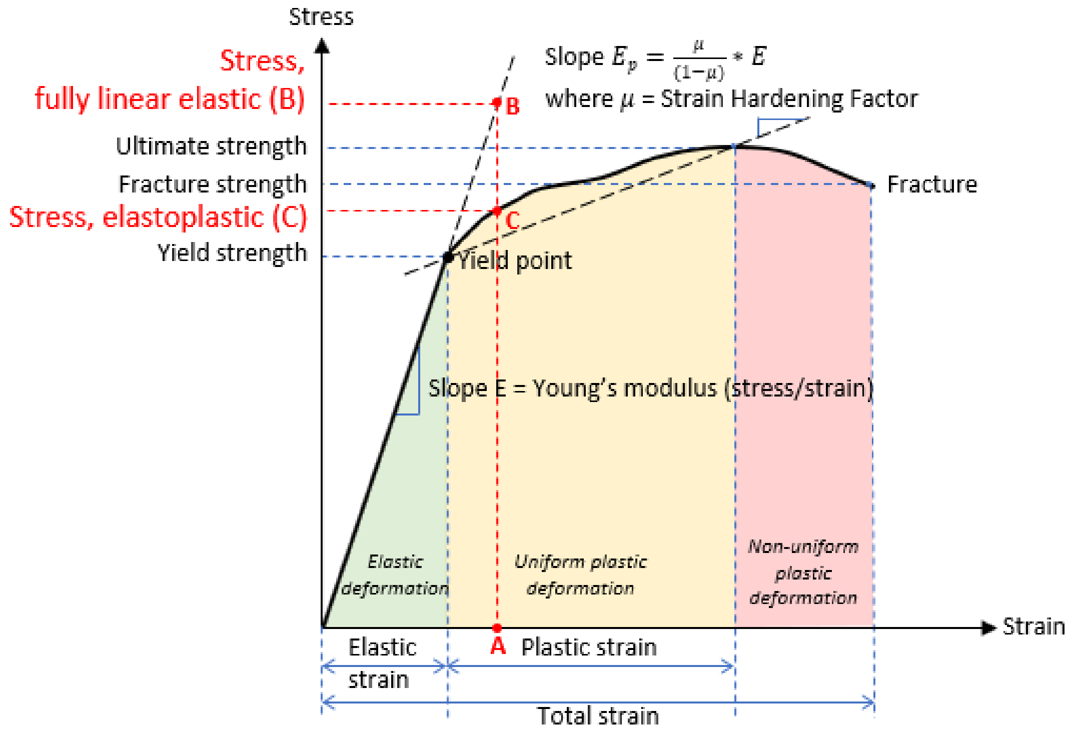


Linear Elastic versus J2-Plasticity Stress Mode

Once you choose a material, you have the option of choosing material behavior in calculations of stress that is either *linear elastic* or elastoplastic (exhibiting both elastic and plastic properties). The elastoplastic calculations are based upon the *J2 (von Mises) plasticity* model.



This stress mode option is associated with a material's ductility, a measure of a material's ability to undergo significant plastic deformation before rupture. The following figure shows stress-strain curves for a typical metal material. After yield, for a given strain, A, in the plastic deformation region, notice that the stress at point B (fully linear elastic) is higher than the stress at point C (elastoplastic). Stress values differ depending on your assumptions about material behavior.



Stress-strain curves based on notions of fully linear elastic and elastoplastic mechanisms

In the Additive application, an assumption of **linear elastic** behavior will result in a higher maximum stress value for a given strain beyond the yield point for the material. This overprediction may not be realistic for parts with larger distortions. The simulation will run faster, however, which may be beneficial if you care about on-plate results only (because you will heat-treat the final part to relieve residual stress, for example). It is important to note that while stress values beyond the elastic range will be artificially high, *distortion* values will generally be correct using the linear elastic option. Therefore, using linear elastic stress mode can be useful for analyzing distortion trends *while the part is still on the baseplate*.

An assumption of elastoplastic behavior (using the **J2-plasticity** model) applies best to ductile materials, such as most metals. Currently, small deformation plasticity has been used in these models where addition of elastic and plastic strains amount to total strain, since metals do not exhibit the large deformations we see in polymers, for example. Von Mises stresses are used to reduce the stress levels when strain values exceed elastic strain. Strain hardening algorithms are included in the stress calculations (see Hardening Factor).

The simulation will run longer with the J2-plasticity option, but this option is required if you want accurate distortion *after-cutoff* results, or accurate indications of stresses and strains.

Hardening Factor

If you select the J2-Plasticity option, a material-specific strain Hardening Factor is used in stress calculations to provide further information about the material's behavior in the plastic deformation region. The Hardening Factor is used to calculate the slope of the stress-strain curve (E_p) above the material's Yield stress:

$$Ep = \frac{\mu}{(1-\mu)} * E$$

This factor may be changed here or when customizing a material.

Note: Previous to the 19.2 update, a hardening factor of 0.1 was used for all materials. Following the update, the default materials each have their own hardening factor. Custom materials created by the user prior to this change will use 0.0198 as the hardening factor. A consequence of this change is that simulations run with this release may have slightly different output values than those run with previous releases. The magnitude of the difference depends on a variety of factors including part geometry and orientation, material, scan pattern, laser power and whether supports are used in the simulation.

Support Yield Strength Ratio

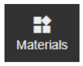
For a good understanding of supports, refer to Define Support Options. The Support Yield Strength Ratio is a parameter associated with the support structure's *material* and is therefore included under the material section of the simulation form. The Support Yield Strength Ratio is used in the finite element simulation to modify the strength of the support material. It affects both yield strength and elastic modulus of the support material. For example, a value of 1.0 results in a support strength equal to the solid material while 0.5 is half the strength of the solid material. The default value is 0.4375. This default was determined by studies where the support strength for default supports built on an EOS M270 machine were tested and compared to solid material built on the same machine.

Strain Scaling Factor

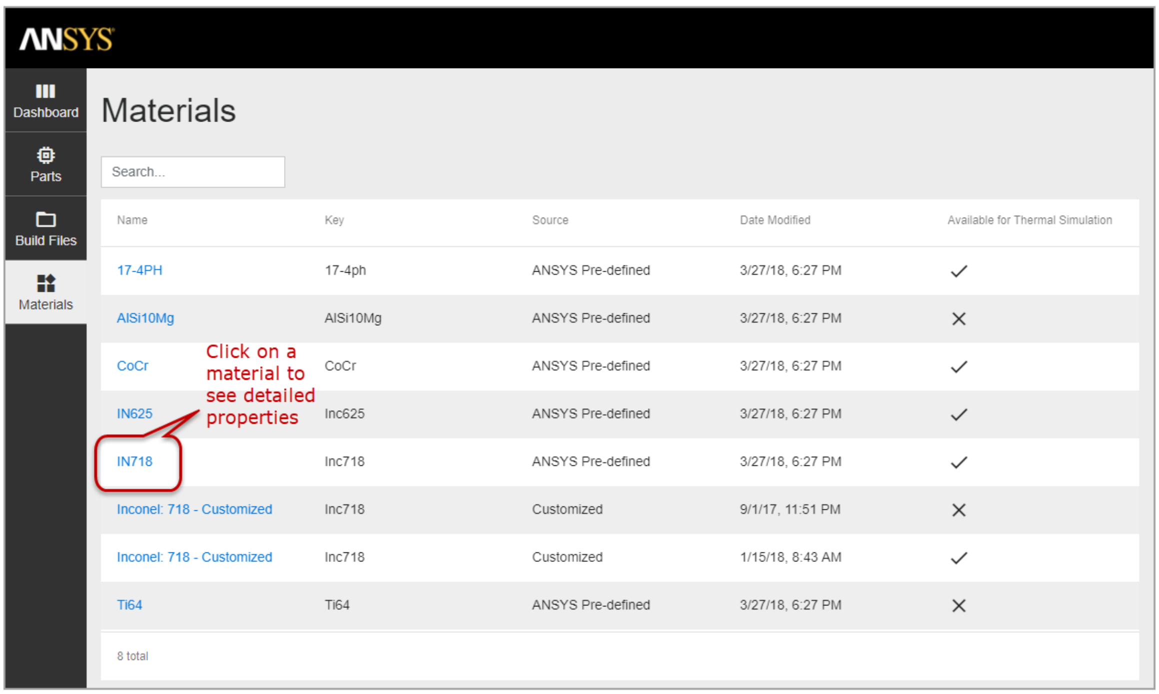
The Strain Scaling Factor (SSF) is a calibration factor that you may use to improve the accuracy of your simulations. This value is a direct multiplier to the predicted strain values. Values greater than 1 will amplify displacement and stress and values less than 1 will reduce them. The default Strain Scaling Factor is 1. You should use the default SSF of 1 for your first simulation in Additive and then work from there to calibrate the best value of SSF. Because these are single data point values and there can be variations even from batch to batch of material, we recommend that you calibrate for SSF for each specific machine and material combination. Calibration procedures are available online here.

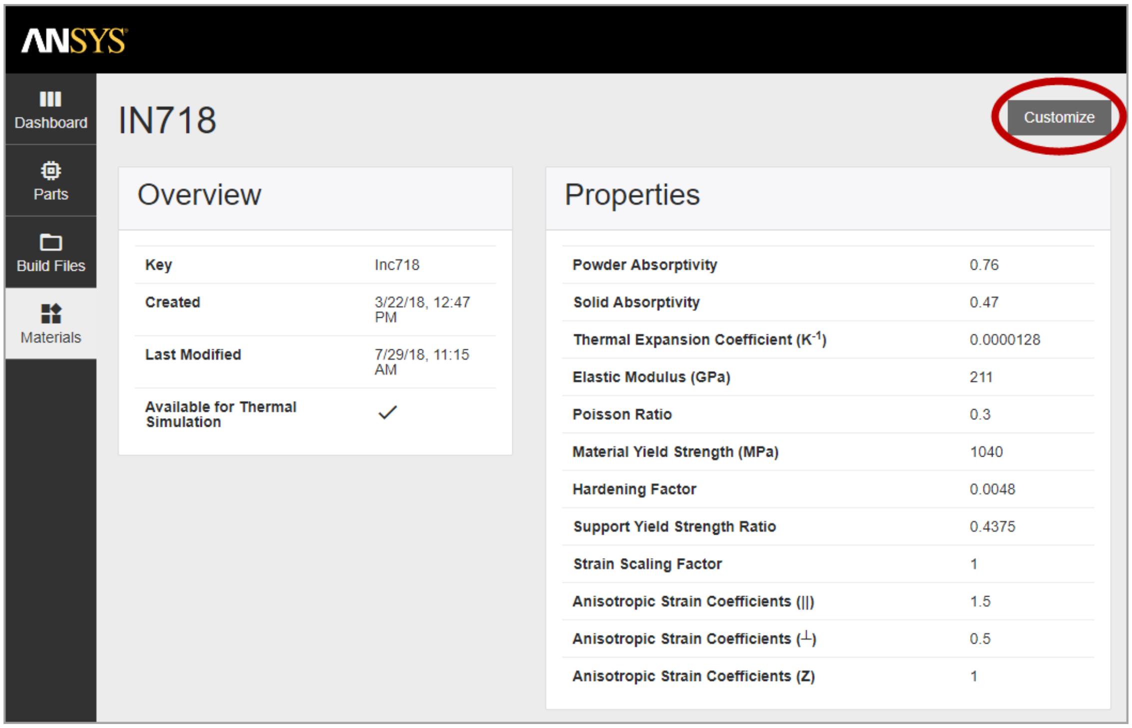
It is not necessary to perform this calibration if you are conducting a *trend analysis*, that is, if you will be examining the effects of variable changes on stress or distortion relative to each other.

Customizing a Material

Click on the 

button in the left panel of the dashboard to bring up the Materials Library. There you will see the list of ANSYS pre-defined materials. Clicking on any of these materials will bring up a panel of detailed properties for that material. Click on Customize to create a new material based upon one of these pre-defined materials. Note that you cannot edit an ANSYS pre-defined material, but you may edit your own customized materials.





Define Support Options

During Laser Powder Bed Fusion, as the laser passes over each layer of metal powder it creates a melt pool similar to a welding process. The melt pool area cools and is reheated again in the next laser pass. With each successive layer, the material underneath cools and contracts. This process of heating and cooling, expanding and shrinking, results in strain, distortion, and residual stress in the part that effectively act to lift the part off the baseplate. Therefore, support structures are required to hold printed parts in place during fabrication.

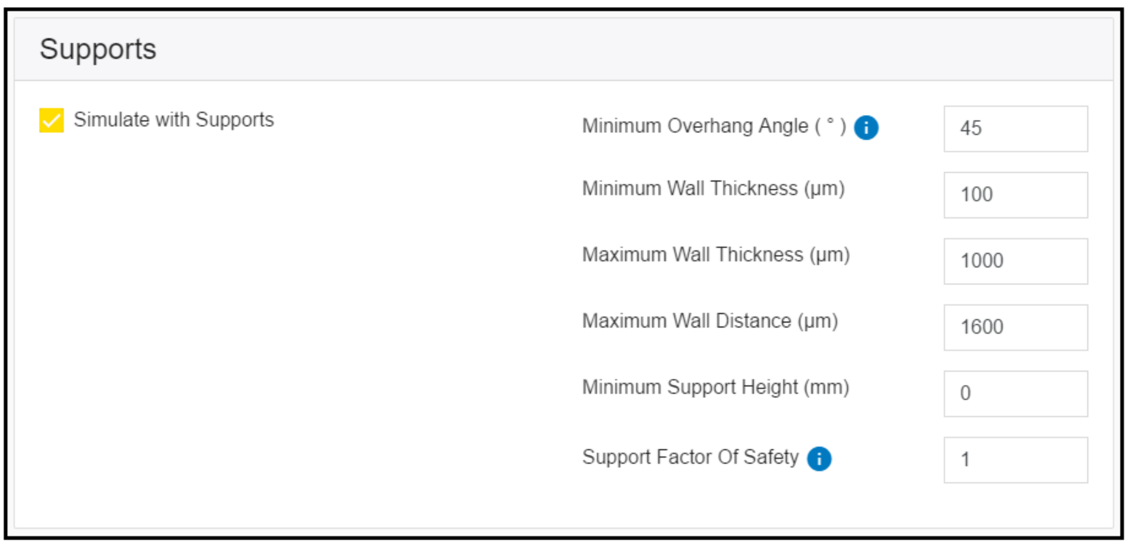
These support structures are commonly thin "walls" printed along with the part that are fixed to the baseplate and connect to the part at areas of the geometry that *overhang* the main body of the part. The supports are printed of the same metal material as the part and must be cut or machined off upon completion of the build. Too many supports, or support walls that are too thick, will require excessive post-build time to remove. Too few supports, or support walls that are too thin, may not be strong enough to hold a part in place and may result in cracks, excessive distortion, drooping between walls, or breaks.

The Additive application simulates the build process with an *initial* set of supports based upon geometry considerations only, and then generates two new sets of *optimized* supports based upon the simulation results. The initial supports are thin, single-bead-scan width support walls placed uniformly underneath overhang areas defined by a user-specified angle. The maximum residual stresses that supports must withstand are predicted in the simulation (in the mechanics solver). The optimized support structures are then automatically generated (in the support generation module) based upon an algorithm which varies the support density to carry these maximum residual stresses. Two sets of optimized supports are generated:

- **Optimized Thin Wall Supports** are of a uniform wall thickness, but wall *spacing* is varied such that more walls are placed in regions of higher residual stress and fewer walls in regions of lower residual stress.
- **Optimized Thick Wall Supports** are uniformly spaced walls with varying *thicknesses* such that thicker walls are placed in regions of higher residual stress and thinner walls in regions of lower residual stress.

Be aware that the conditions defining inherent strain are not applied to supports, they are applied only to the solid part material. Thus, stress is not accumulating in the supports as the layer-by-layer addition of material is simulated until solid part material is reached, at which time the part material will cause some stress to develop in the supports.

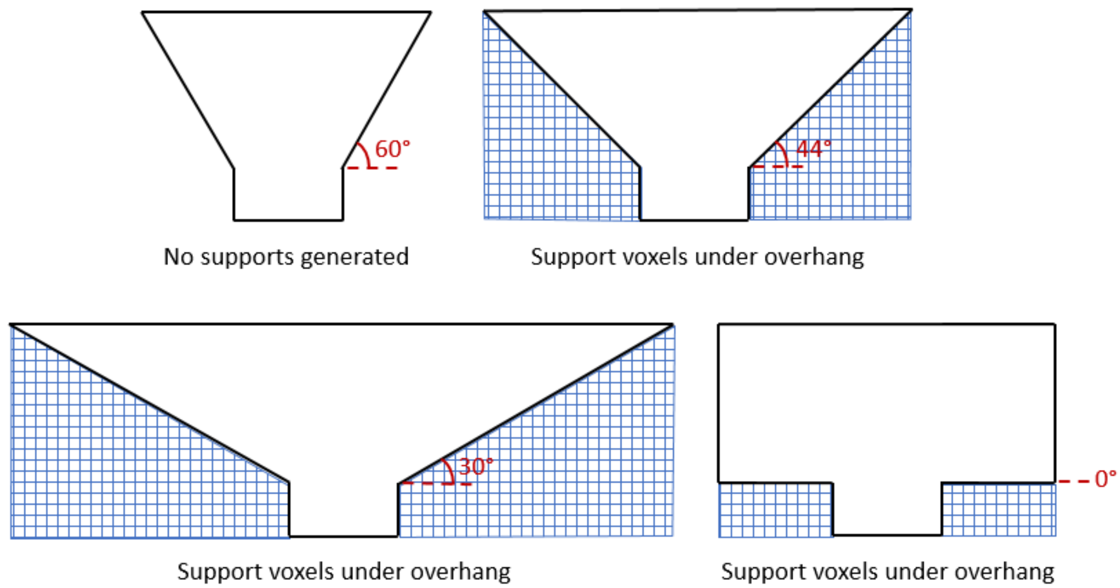
On your simulation form, you will need to specify certain parameters that guide the support generation process.



Minimum Overhang Angle (°)

The overhang angle is measured from the powder bed surface (horizontal = 0 degrees) up to the surface of the part. Any point on the surface of the part having an angle *less than* the Minimum Overhang Angle will be supported. The default Minimum Overhang Angle is 45 degrees. Avoid using a value that is the same as the angle of the geometry of your part, as it can cause asymmetric support structures due to finite rounding errors. For example, if your geometry includes an overhanging feature of precisely 45 degrees, use 46 or 44 degrees for Minimum Overhang Angle.

Supports will be created for overhang areas even in cases where the supports cannot reach the baseplate because a portion of the part is in the way. In that case, supports will span part-surface to part-surface.



Supports will be generated under overhanging features with angles less than the Minimum Overhang Angle (defaults to 45°)

Minimum and Maximum Wall Thickness (μm)

Minimum and Maximum Wall Thickness are parameters used for the Optimized Thick Wall Supports. Minimum Wall is the thinnest possible support wall that the machine will build. Usually you will specify the thickness of a single bead scan. The default value is 100 microns. The thickness of support walls will not exceed the Maximum Wall Thickness. The default value is 1 mm.

Maximum Wall Distance (μm)

Maximum Wall Distance is a parameter used for the Optimized Thin Wall Supports. It is the allowed maximum distance between two neighboring support walls. Regardless of the predicted stress level in the support structure, the walls in supported regions will be spaced not more than this value. Too large of a wall distance might result in failures such as the part breaking away from the support or the development of cracks in the support structure. When a laser scans a relatively large area of powder where the support wall distance is too wide, cracking might happen since powder has no strength to hold the solidified part in place. The excessive distortion might cause blade and part collision. *We recommend that maximum wall distance should not exceed 2 mm when a single bead support wall is used.*

Minimum Support Height (mm)

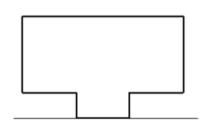
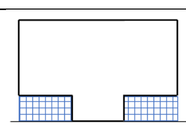
This is the height, in millimeters, that the part will be elevated off the baseplate. If you set a value of 3 mm then the part will be elevated such that the lowest point on the part is at least 3 mm above the baseplate. This value should be set to allow for an easy part cut-off from the baseplate while also considering how many voxels must be created to add that additional height. (More voxel layers = more simulation time.) We recommend that this value be set as low as is realistic for each simulation. The default value is 0. (See What Happens if I Do Not Use Supports?)

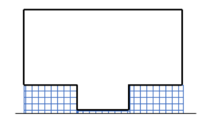
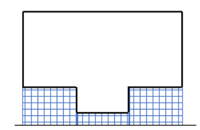
Support Factor of Safety

The support factor of safety is a parameter that drives the strength of the automatically generated *optimized* support structures. If you would like the supports to withstand 2x the expected load, then you would enter a 2 in this field and the predicted strength of the auto-generated support structure would be double the predicted stress. The strength of the support structure is driven by the number and thickness of support walls that are generated. The support factor of safety defaults to 1.

What Happens if I Do Not Use Supports?

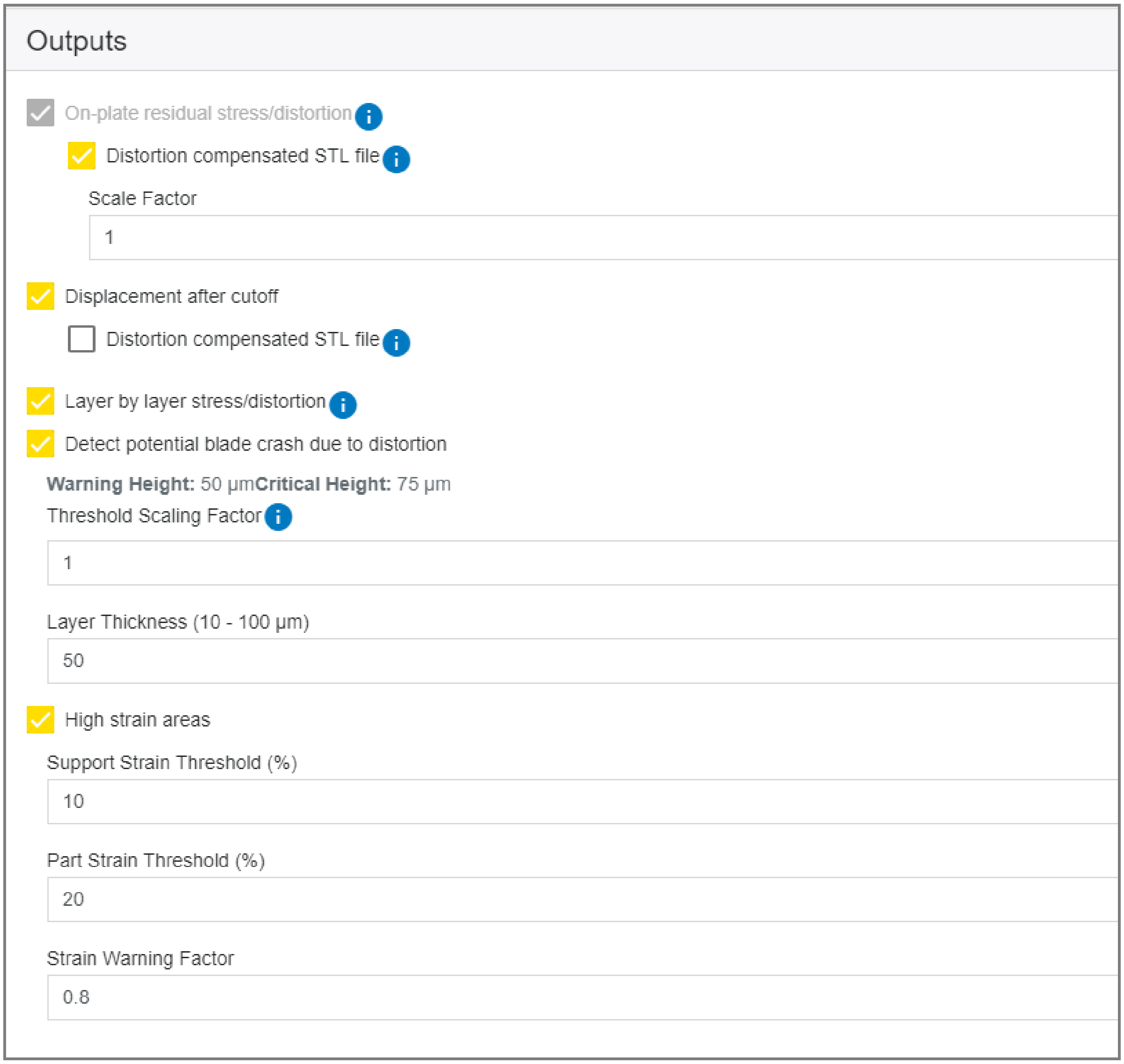
By default, supports will be generated in all simulations, but you may uncheck the Simulate with Supports box on the form to perform a simulation without supports. There are some subtle assumptions that occur in the simulation depending on the *outputs* that you select, however. The possible scenarios are described in the following table.

Simulate with Supports (checkbox)	**Minimum Support Height (a support option)**	**Displacement After Cutoff (an output selection)**	**What Happens?**	**Part**
☐	N/A (0)	☐	No supports generated Bottom nodes are rigid (no displacements)	
☐	N/A (0)	☑	One layer of support voxels added to simulate part after cutoff No other supports added anywhere Support Yield Strength Ratio set to 1	
☑	0	☐	Supports generated below Minimum Overhang Angle	

			Bottom nodes are rigid (no displacements)	
☑	0	☑	Supports generated below Minimum Overhang Angle One layer of support voxels added to simulate part after cutoff Support Yield Strength Ratio set by user	
☑	Non-zero, for example, 3 mm	☐	Supports generated below Minimum Overhang Angle 3mm of voxel layers created Support Yield Strength Ratio set by user	

Select Outputs

The powerful features of ANSYS Additive leading to the most useful insights for users are initiated by simple output option checkboxes. On a simulation form under Outputs, there are several options from which to choose, depending on your simulation goals. These options may affect simulation run-time but provide additional output files that will be available under Completed Simulations when the simulation is complete. In some cases, additional inputs are required.



On-Plate Residual Stress/Distortion

The basic output produced for all simulations (it appears grayed out because it is the default set of results) contains a voxelized representation of the part with predicted residual stresses and displacements at the end of the build while the part is still attached to the baseplate. Both end state stresses and maximum stress during the build are contained in this .vtk file. Results may be viewed directly in ANSYS Viewer or downloaded.

Distortion Compensated .stl File

The Additive application can predict the location and magnitude of distortion and then "reverse" distort the original .stl file. Then when you build your part using the compensated geometry, the result will be closer to the original design.

On a simulation form under Outputs (either on-plate or after cutoff), check the box for Distortion compensated .stl file. Once selected, you have the option to specify a Scale Factor. The Scale Factor will change the magnitude of the distortion applied to the original .stl file. A Scale Factor of 1 (default) will create an .stl file with distortion compensated by the same magnitude as the simulated results. A Scale Factor < 1 will compensate less than the simulation-predicted magnitude and a value > 1 will compensate more than the simulation-predicted magnitude.

Once the simulation is complete, a file labeled Compensated Geometry (and Compensated Geometry (after cutoff)) can be found in the Output Files section under Completed Simulations. Click the download link to get the distortion compensated .stl file. A second output is also created named Geometry with Distortion (and Geometry with Distortion (after cutoff)). This is a geometry representation in .vtk format with distortion vectors with each vertex.

How Does Distortion Compensation Work?

The Additive application simulates spatial distortion that occurs during the build process. With Distortion Compensation activated, each vertex in the Compensated Geometry .stl file is moved in the opposite direction of the closest distortion vector (i.e., new position = original position + deformed (dX, dY, dZ) **x -1**). To ensure the .stl file has fidelity on par with the simulation, any triangle with an area greater than an equilateral triangle with sides equal to the selected voxel size is split into smaller triangles.

Since the Distortion Compensation feature is a simple linear assumption, it may not always be correct. Parts can respond non-linearly. For example, if you use a Scale Factor of 1 (default) in a simulation and then use the compensated file to rerun the simulation, you will see if the part is predicted to distort to the correct shape. If not, you'll know that the compensated .stl file was under-compensated or overcompensated and you can adjust the Scale Factor up or down from there. *Generally, the Distortion Compensation feature tends to be an iterative process. In fact, our experience thus far shows good results using two iterations with Scale Factor = 0.5 for each simulation. Alternatively, a good starting point for one iteration is a Scale Factor = 0.75.*

Displacement After Cutoff

Choose Displacement After Cutoff to view the voxelized representation of the part with predicted displacements after the part is *cut off the baseplate.* In the case where you have chosen to simulate without supports (i.e., you have unchecked Simulate with Supports in the Supports section of the simulation form) and you choose Displacement After Cutoff, one layer of support voxels will be added to the base of the part to simulate part after cutoff. (See What Happens if I Do Not Use Supports.)

Layer by Layer Stress/Distortion

Choose this option for detailed voxel layer-by-voxel layer results to learn insights about the behavior *as the part is being built*. Results are for the surface of the part, not the interior information. The output consists of a .zip file containing a series of .vtk files, one for each voxel layer. The .zip file may be large. Use these files to “animate” the build process. View locations throughout the part of potential blade crashes and high strain areas that may indicate cracks.

Blade Crash Detection

Considering the amount of distortion that is possible within a part being built, a scenario of concern is a collision of the recoater blade and the distorted part. The Additive application provides a blade crash detection feature that predicts if and where such a collision will occur.

On a simulation form under Outputs, check the box for Detect potential blade crash due to distortion. Once selected, you have the option to specify a Threshold Scaling Factor and Layer Thickness. (Note that for Scan Pattern and Thermal Strain simulations, the Layer Thickness parameter appears in the Machine section of the simulation form.)

Once the simulation is complete, a file labeled Potential blade crash locations can be found in the Output Files section under Completed Simulations. Click the download link to get the .csv file. Indications of blade crash are also available on the On-plate stress/displacements .vtk and .avz files and the Layerwise .vtk files.

Each voxel will be assigned one of the following values for predicted blade crash:

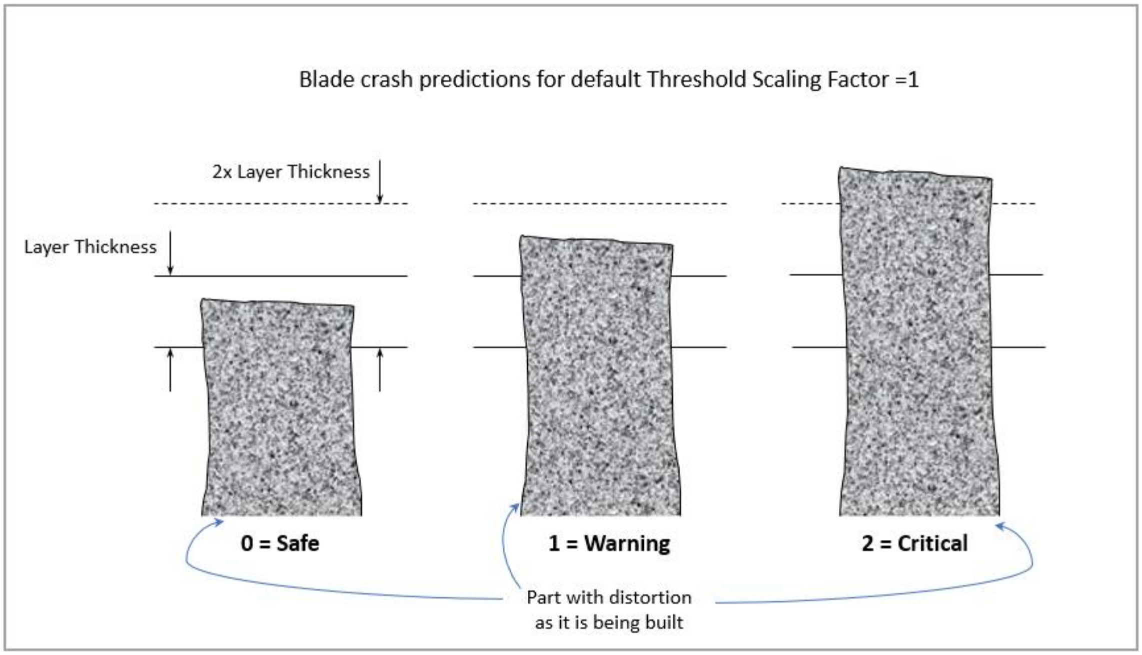
- 0 (none predicted)
- 1 (warning—potential blade crash)
- 2 (critical—likely blade crash)

Layer Thickness (10-100 µm)

Layer Thickness is the thickness of the powder layer coating that is applied with every pass of the recoater blade. The default value is 50 microns. We recommend that you use the actual thickness unique to your machine.

Threshold Scaling Factor

This value is used to modify blade crash calculations so that you can allow for flexibility in the recoater blade. Using the default value of 1 and a Layer Thickness value of 50 µm, any distortion in the positive Z direction over 50 µm will be marked as a warning (potential blade crash) and any distortion over 100 µm will be marked as a critical area (likely blade crash). In another example, if you know there is not a lot of flexibility in the recoater blade, use a threshold value of 0.8. With a 50-micron Layer Thickness, any distortion in the positive Z direction over 40 microns will be marked as a warning and any area with distortion greater than 80 will be marked as critical.



High Strain Areas

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. This is a common issue that can affect parts built with additive manufacturing. The Additive application allows you to quickly look at the design and process settings that would alleviate potential cracking. The High Strain Areas feature 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. Required inputs for this output type include the Support Strain Threshold, Part Strain Threshold, and Strain Warning Factor.

Strain threshold values should be entered as a percent for the support and part material respectively. (Engineering strain can be calculated as the change in length divided by the original length. For example, a 3.0" titanium bar that has been stretched to 3.3" is said to have experienced a tensile strain of 0.1, or 10 percent.) The default values of 10 and 20 percent strain are simply sample values, and you should adjust these values as needed for different materials or other factors that may affect the total elongation of your build material. When a calculated strain exceeds these threshold values, it will be labeled as "Critical."

The Strain Warning Factor allows you to establish a "Warning" range, to identify further areas where strain is approaching the critical range.

Support Strain Threshold (%)

Percentage strain in the *supports* above which strain will be considered critical. Defaults to 10%.

Part Strain Threshold (%)

Percentage strain in the *part* above which strain will be considered critical. Defaults to 20%.

Strain Warning Factor

This value is multiplied by each of the strain thresholds above to define limits where strain is labeled as a warning. Defaults to 0.8.

Using default values for all inputs, strains in the *supports* between 8 and 10 percent will be in the warning range, while strains over 10 percent will be considered critical. Strains in the *part* between 16 and 20 percent will be in the warning range, while strains over 20 percent will be considered critical.

The output of High Strain Areas can be viewed as part of the On-plate stress/displacement .vtk and .avz files and the Layerwise .vtk files, as well as a High strain regions .csv file. When viewed through the On-plate stress/displacement output, critical strain locations are given a value of 2 and warning locations are given a value of 1. All other locations retain a value of 0 to show low risk. The High strain regions .csv file consists of strain values for all points with strains at or above the warning threshold. Information about each point includes the x, y, and z locations along with the strain value and the deposit layer. The deposit layer represents the actual powder layer during a build. When using the Assumed Strain analysis type, a layer thickness of 50 µm is used to identify the deposit layer of each location.

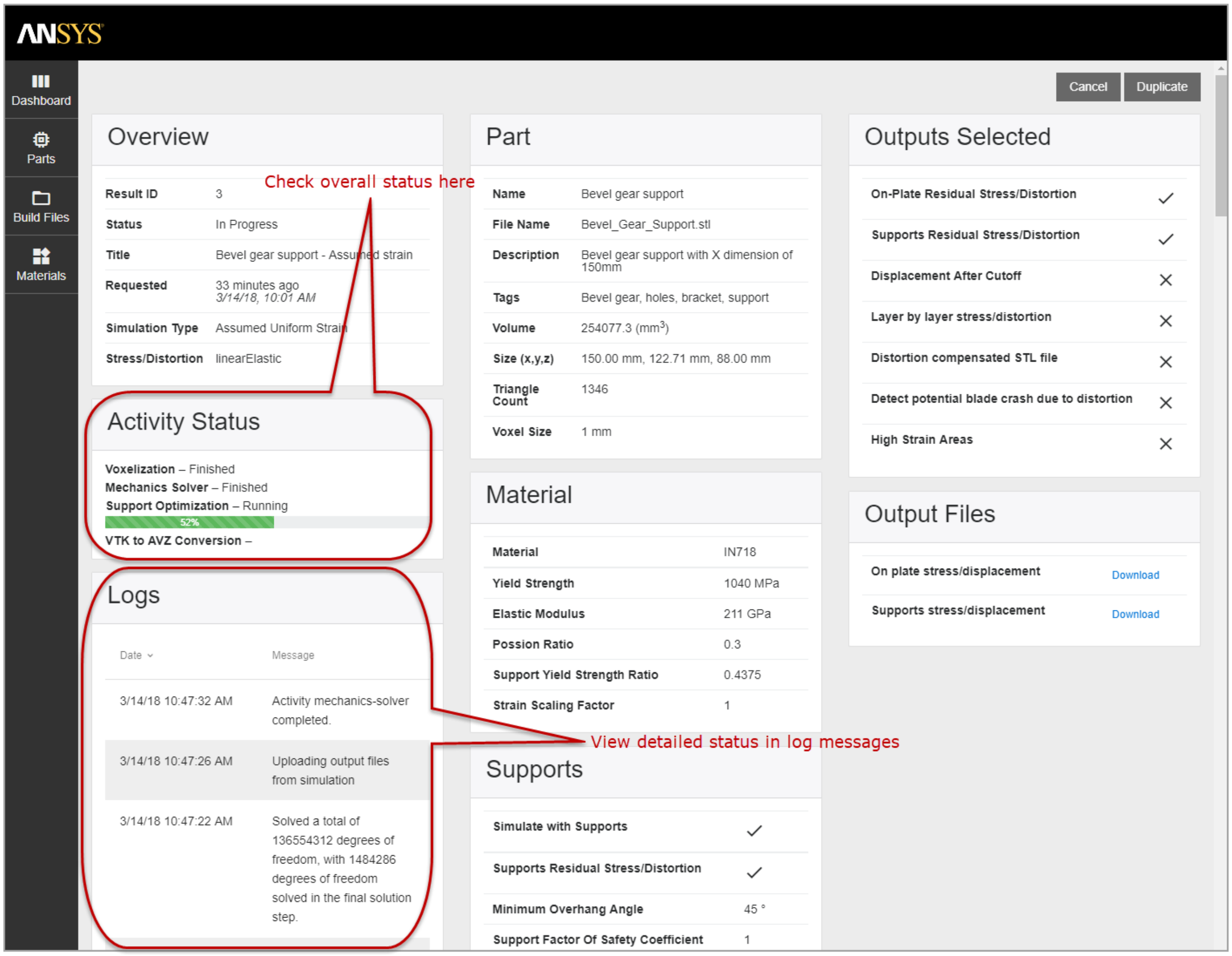
Save/Export a Draft Simulation

You may want to save your simulation form periodically before starting the simulation. While it is not saved to a specific file on your computer, it is saved internally, and you will see it listed under Draft Simulations. It is removed from Draft Simulations when you start a simulation (i.e., when it is no longer a "draft"). All your input options are stored when you run a simulation so that you may see your options at any time when you click on a simulation in the Running Simulations and Completed Simulation areas of the dashboard.

To save your inputs to a file, use the Export button. (A Save action is required before you can Export.) Exported files have an .aasp extension and may be imported using the Import button under Draft Simulations. Exported files do not include the part.

Step 3. Run a Simulation

Start a simulation by clicking on the Start button at the bottom of your simulation form. You will immediately see status activity in a convenient summary format.



How Long Will My Simulation Take?

ANSYS Additive is fast! For simple simulations, your run time will take minutes or even just seconds. For more complicated geometries and simulation options, the program may run for many hours, or even days, but you will see status activity in the log indicating progress throughout the simulation. Keep in mind these general considerations:

- A simulation with J2-plasticity (stress mode) will take slightly longer than a simulation assuming linear elastic stress behavior. Stress results will be more accurate. See Linear Elastic versus J2-Plasticity Stress Mode.
- A Scan Pattern simulation will take slightly longer than an Assumed Strain simulation. See Scan Pattern (Anisotropic) Strain.

- A Thermal Strain simulation will take much longer than any other type of simulation, but it provides the highest level of accuracy in results. See Thermal (Anisotropic) Strain.
- Too small of a voxel size can increase simulation time significantly. See How Does Voxel Size Decrease Run Time?
- A part oriented with the longest dimension in the Z direction will take longer during simulation than parts oriented with their longest dimension in the plane of the baseplate. While the domain volume doesn't change with part orientation, the number of voxel *layers* changes. See Guidelines for Part Orientation and Resolution.
- Run time will get slower per voxel layer as the simulation proceeds through the voxel layers. There are more calculations being performed for each new layer as the simulation progresses.
- Simulation time does not equal build time but it is related. If the time to build your part takes a week, it is reasonable to expect the simulation to take a few days.

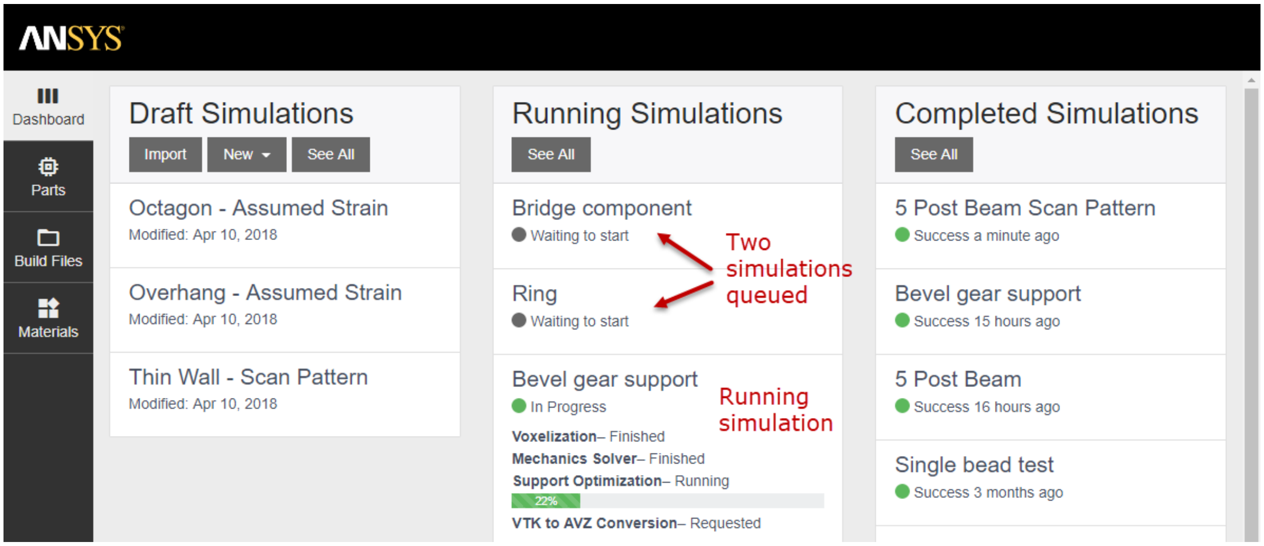
Common Error Messages During a Simulation

There may be occasions when a simulation fails and cannot continue. If that is the case, a warning message will pop up on your desktop and you will get an error message in the activity log. The following table lists a common error message related to the number of iterations in the simulation and recommended actions.

Error Message Invalid state: Hit maximum iterations (12000) with a residual of 10071.14…	
Cause	**Suggested Solution**
Too coarse of a mesh, leading to poor connectivity between elements (i.e., elements only sharing corners/edges).	Try reducing voxel size to improve connectivity of the mesh.
As the number of iterations required to reach convergence scales nonlinearly with the problem size, this may be indicative of too large of a problem.	Try increasing voxel size to reduce the problem size.
May happen in regions of a part with large unsupported cantilevered overhangs, resulting in excessive local distortion.	May try adjusting support structure settings to avoid large unsupported overhangs.
Convergence is dependent on the level of interconnectedness of the mesh. Thin shell parts or parts connected only by slender members may not be so well interconnected.	May try modifying geometry to improve interconnectedness or try simulating the part in a different orientation.

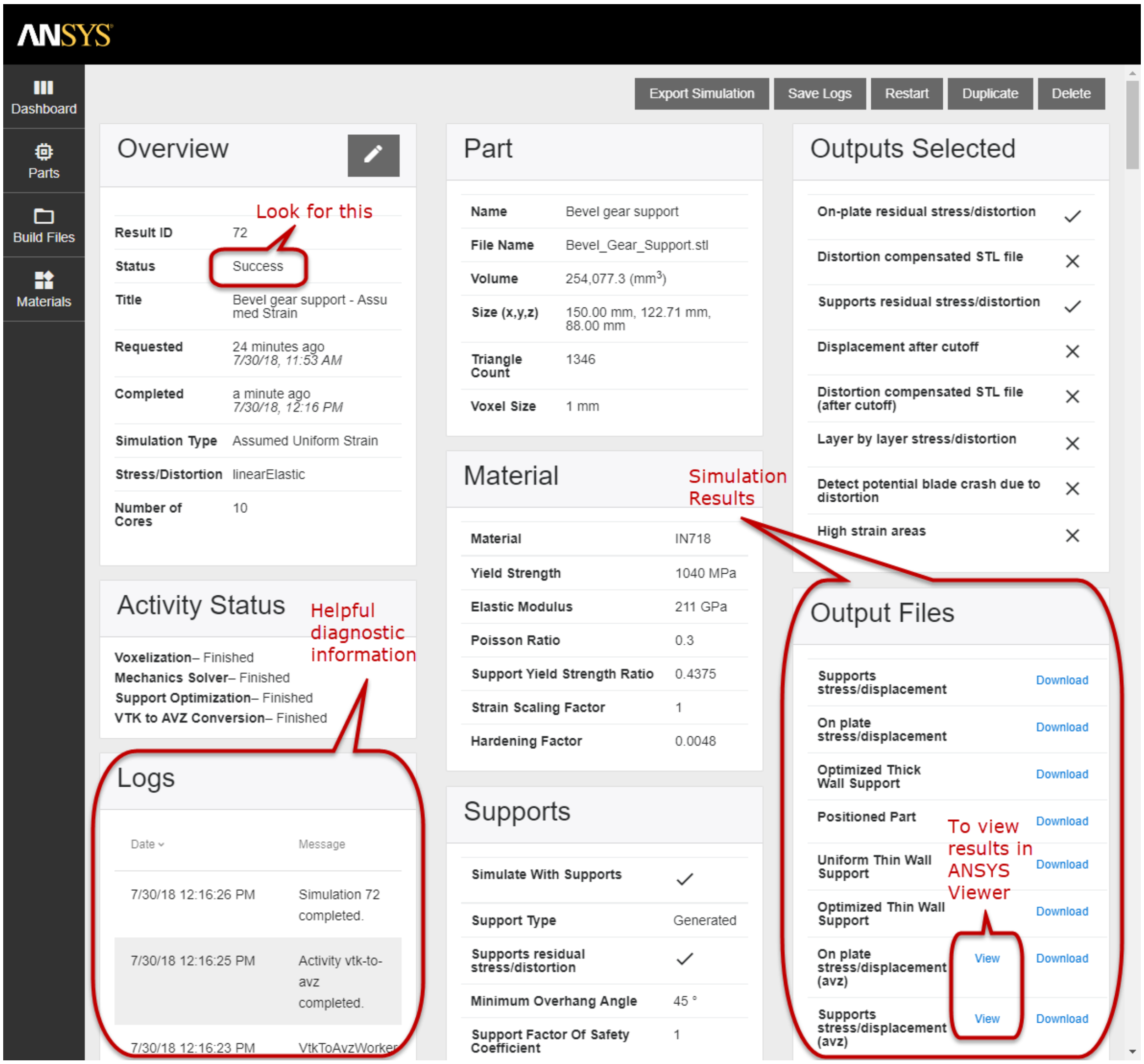
Can I Run Multiple Simulations at Once?

You may queue multiple simulations so that one will start immediately after the one before it completes. This is extremely convenient if you want to line up a few simulations to run overnight, for example.



Step 4. Review Results of a Simulation

Look for the Success status indication to know your simulation has completed. In the Overview and Logs sections, you will see beginning and ending time stamps and other useful information. Simulation results are found in the Output Files section.



Output Files

Results from your simulation are viewable in ANSYS Viewer, or are downloadable files (.avz files are ANSYS Viewer files), or both, as described here:

On-plate stress/displacement (both .vtk and .avz)

Contains the voxelized representation of the part with predicted displacements and stresses at the end of the build (i.e., end state) while the part is *still attached to the baseplate*.

- Displacement in mm: magnitude, x, y, and z components
- Current state stress in Pa: von Mises, xx, yy, zz, xy, yz, zx directions
- Max stress during build in Pa: von Mises, and xx, yy, zz in compression & tension for each
- Blade crash severity (if Detect potential blade crash output option is chosen)
- High strain severity (if High Strain Areas output option is chosen)

After cutoff displacement (both .vtk and .avz)

Contains the voxelized representation of the part with predicted displacements after the part is *cut off the baseplate*.

- Displacement in mm: magnitude, x, y, and z components

Potential blade crash locations (.csv)

Includes locations of all potential and likely blade crashes and the predicted magnitude of the Z displacement at those points.

- Global x, y, z coordinates, and total Z displacement in mm

Layerwise .vtk files (.zip)

When unzipped, a series of .vtk files that show voxelized representation of part layer-by-layer *during build*. You will have as many .vtk files as voxel layers.

For each voxel layer:

- Displacement in mm: magnitude, x, y, and z components
- Current state stress in Pa: von Mises, xx, yy, zz, xy, yz, zx directions
- Blade crash severity (if Detect potential blade crash output option is chosen)
- High strain severity (if High Strain Areas output option is chosen)

Use these files to “animate” the build process. View locations throughout the part of potential blade crashes and high strain areas that may indicate cracks.

Supports stress/displacement (both .vtk and .avz)

Contains the voxelized representation of the *support structure* with predicted displacements and stresses at the end of the build (i.e., end state) while the part is *still attached to the baseplate*.

- Displacement in mm: magnitude, x, y, and z components
- Current state stress in Pa: von Mises, xx, yy, zz, xy, yz, zx directions
- Max stress during build in Pa: von Mises, and xx, yy, zz in compression and tension for each

View this file together with On plate stress/displacement to see full build together (part and supports). Note that these are the default uniform supports, not the optimized supports. Recall that stress is not accumulating in the supports until they are affected by the solid part material. See Define Support Options.

High strain regions (.csv)

List of high strain warning areas during the build.

- Global x, y, z coordinates

Optimized Thin Wall Support (.stl)

Contains optimal *thin wall support layout* based on predicted stresses and distortions that can be used to minimize risk of support failure.

Optimized Thick Wall Support (.stl)

Contains optimal *thick wall support layout* based on predicted stresses and distortions that can be used to minimize risk of support failure.

Positioned Part (.stl)

Input geometry (non-compensated) positioned into its start location and orientation, that is, offset to account for supports between the baseplate and the part.

Uniform Thin Wall Support (.stl)

Representation of uniform thin walled supports used in the finite element simulation *before* support optimization occurs.

View this file to see areas of the part that need supports based on Minimum Overhang Angle specified on the simulation form. Compares to typical third-party software that generates supports on part areas that meet the overhang minimum.

Compensated Geometry (.stl)

Contains the distortion-compensated 3D surface representation (tessellated triangles) of the part *while the part is still attached to the baseplate*. The compensated geometry is placed flush with the baseplate surface and does not include the offset for supports between the baseplate and the part.

You may want to use this file as the part geometry for final production builds.

Compensated Geometry (after cutoff) (.stl)

Contains the distortion-compensated 3D surface representation (tessellated triangles) of the part *after the part is cutoff from the baseplate*.

You may want to use this file as the part geometry for final production builds.

Geometry with Distortion (.vtk)

Contains the 3D surface representation (tessellated triangles) of the original, undistorted part with predicted displacements at the end of the build while part is *still attached to the baseplate*. The geometry does not include the offset for supports between the baseplate and the part.

- Displacement in mm: magnitude, x, y, and z components

Geometry with Distortion (after cutoff) (.vtk)

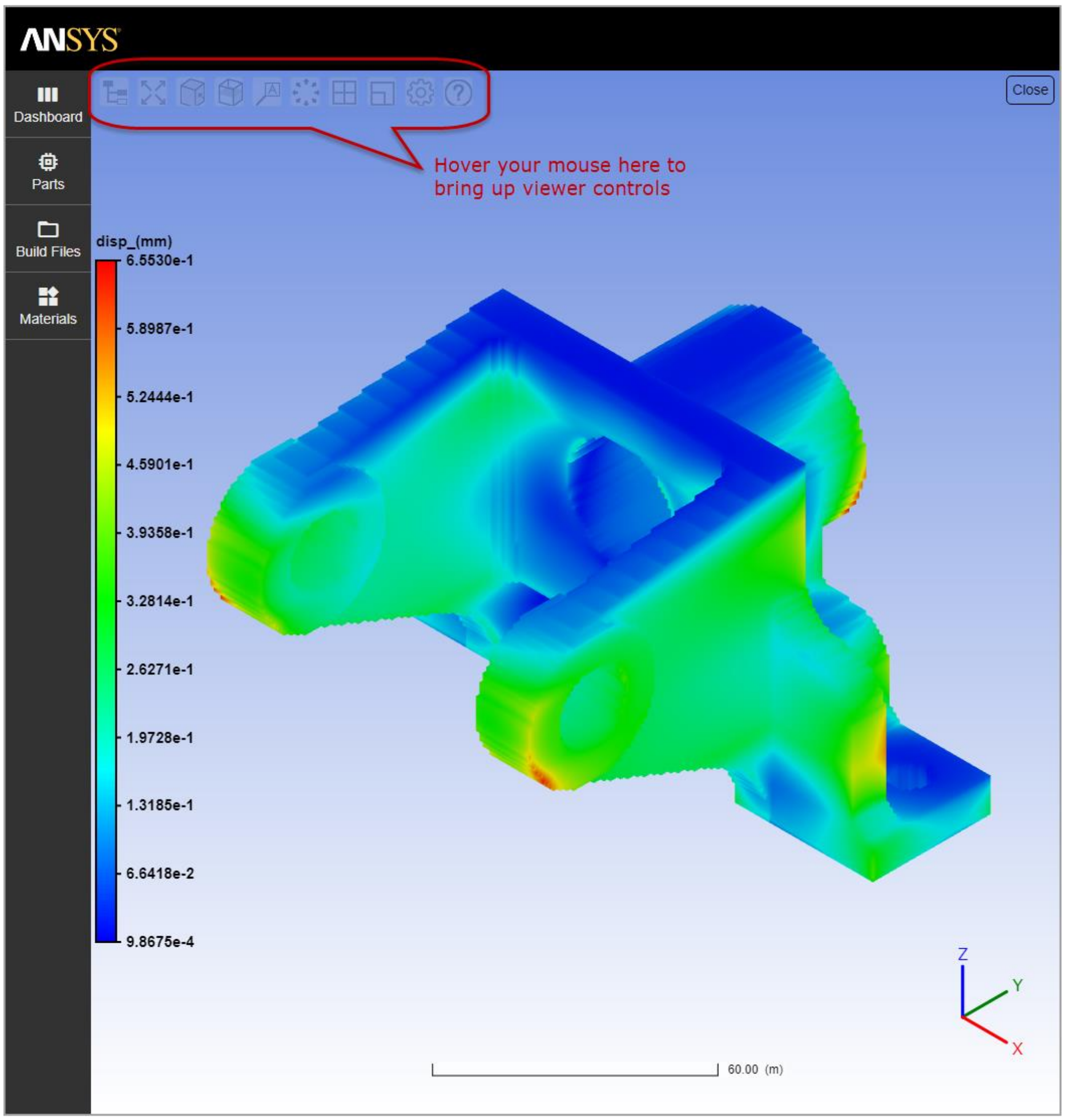
Contains the 3D surface representation (tessellated triangles) of the original, undistorted part with predicted displacements *after the part has been cut off from the baseplate*.

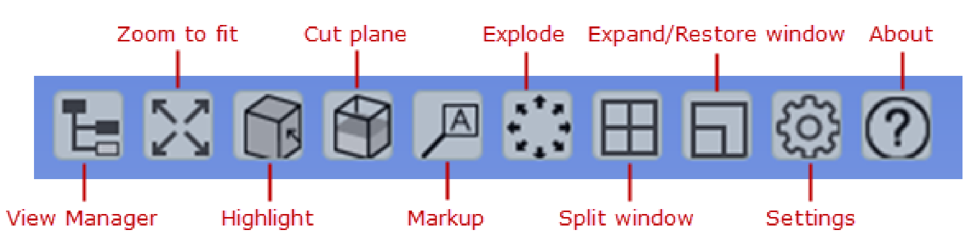
- Displacement in mm: magnitude, x, y, and z components

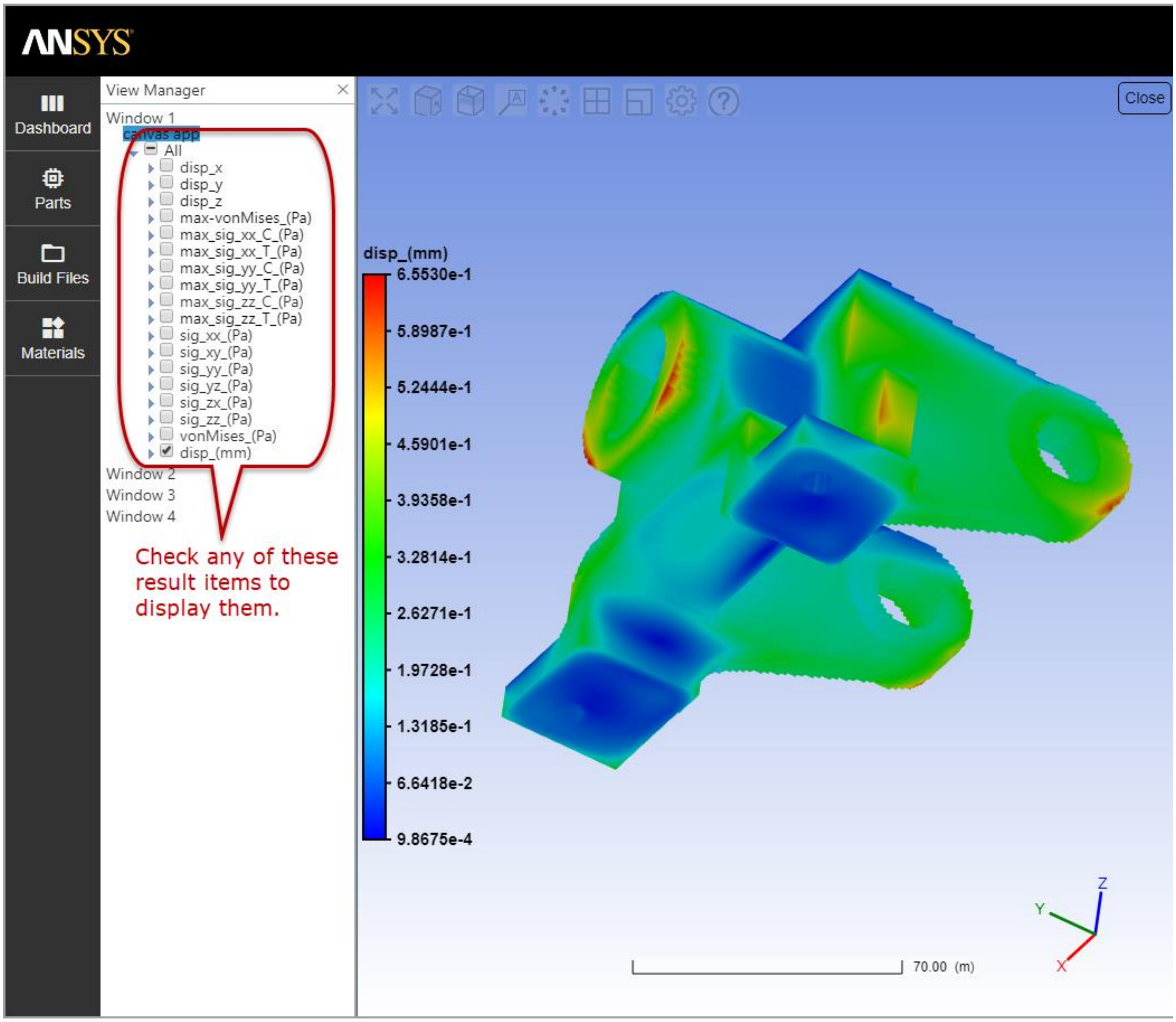
Using ANSYS Viewer to Review Results

ANSYS Viewer is an interactive 3D image viewer that is embedded in ANSYS Additive. It allows you to easily visualize simulation results in 3D models.

To bring up ANSYS Viewer, click on "View" next to any of the outputs with that link in the Output Files section under Completed Simulations.



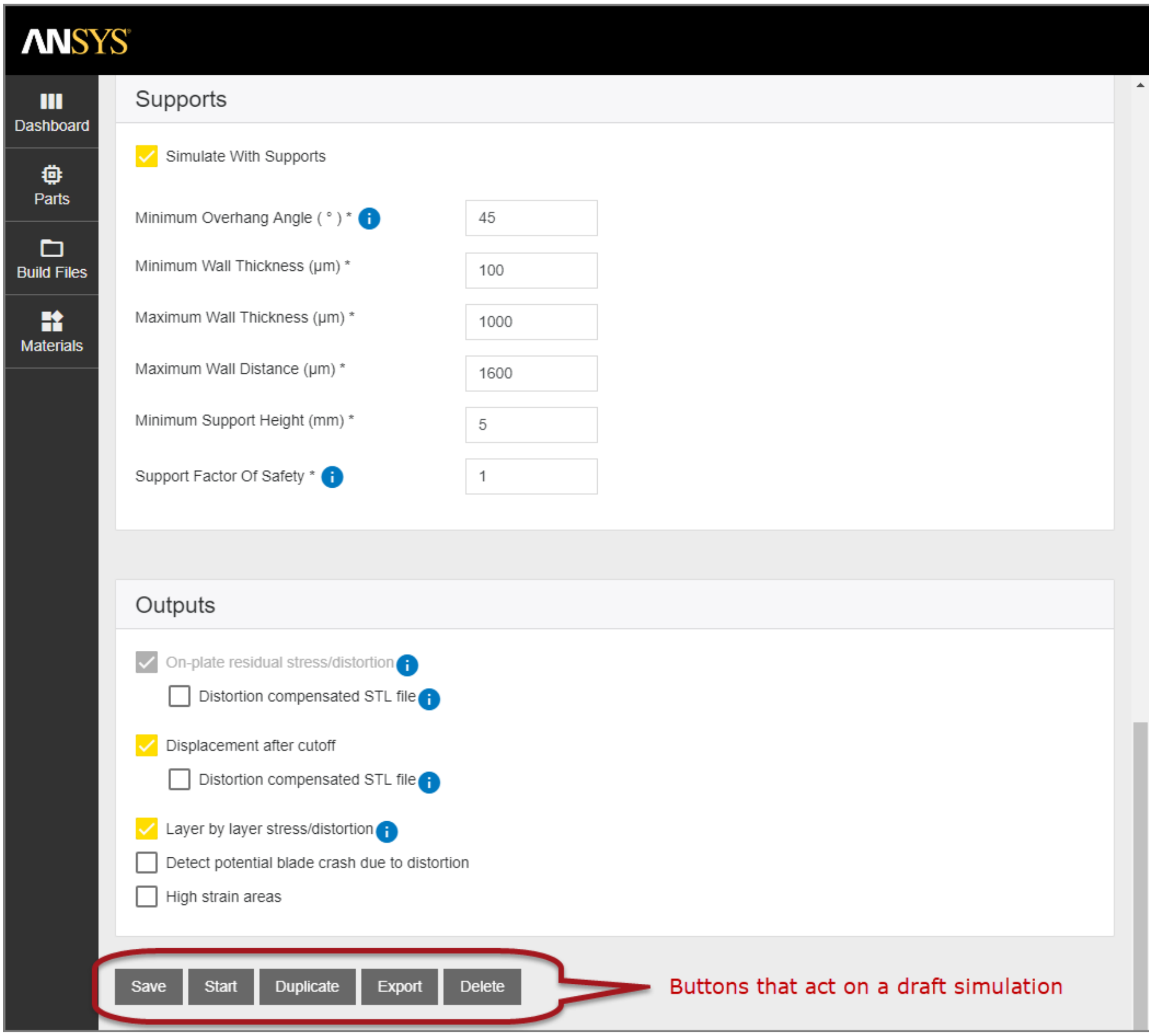


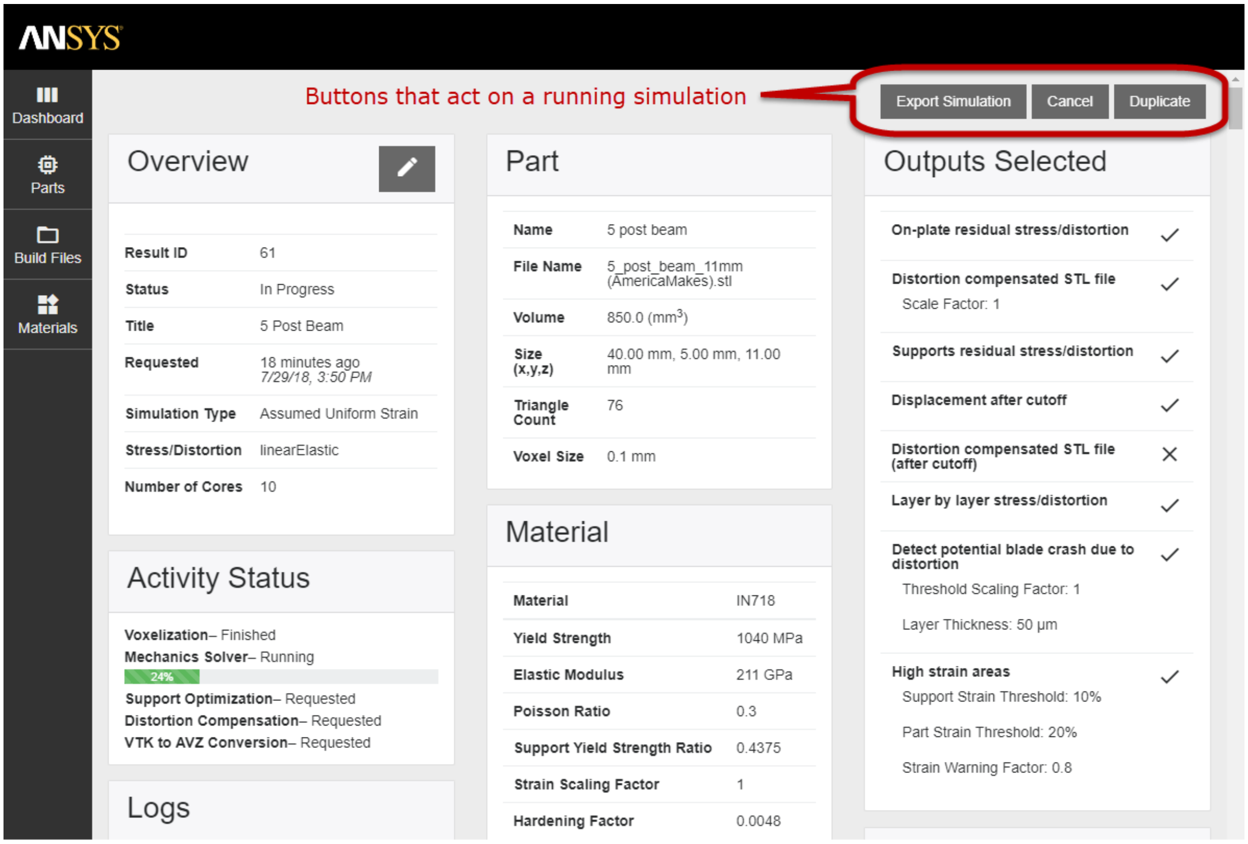


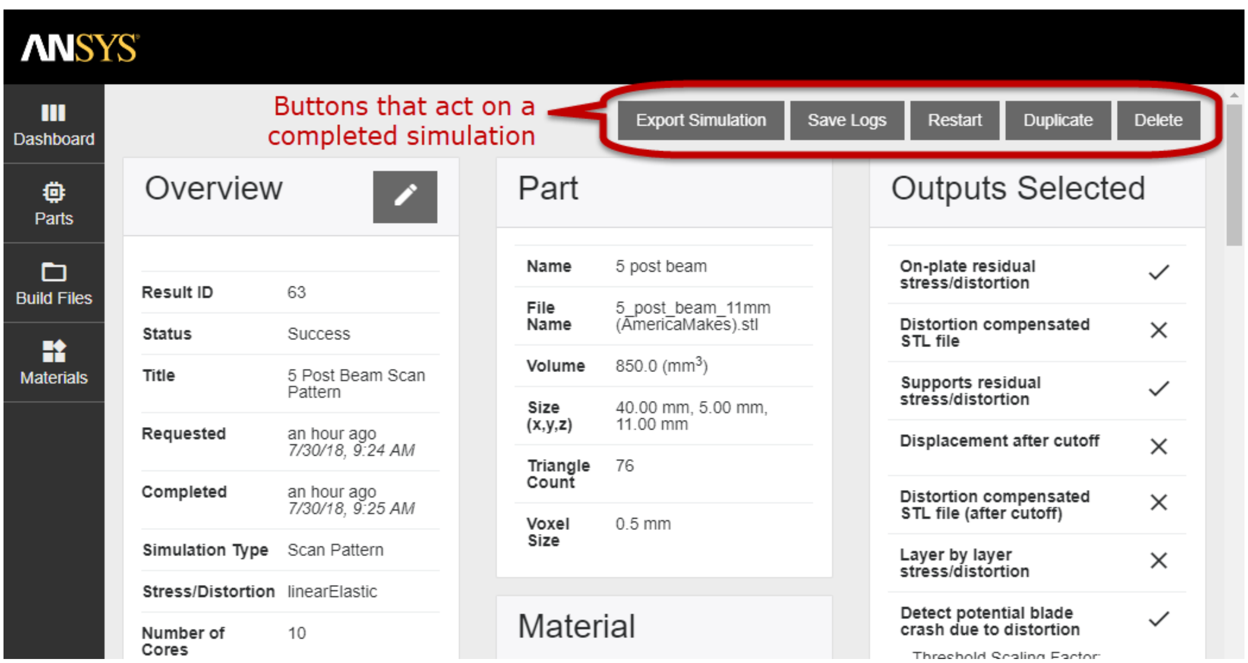
You can click on different result items in the View Manager to see them displayed. Use your mouse buttons to move the part around in the image preview; left button for spin, middle for zoom, and right button for pan.

Exporting, Restarting, Deleting and Other Actions on Simulations

At most points while using Additive, you have opportunities to Export, Save, Start/Restart, Duplicate, Cancel, and/or Delete a simulation, depending on where you are in the program. The operations are designed to work within a database paradigm, as described below.







Save: Clicking the Save button under Draft Simulations saves your simulation form internally (but not as a file on your computer) and you will see it listed under Draft Simulations. It is removed from Draft Simulations when you start a simulation (i.e., when it is no longer a "draft"). All your input options are stored when you run a simulation so that you may see your options at any time when you click on a simulation in the Running Simulations and Completed Simulation areas of the dashboard. Use Export to save your simulation form to a file.

Start: Click Start under Draft Simulations to start execution of a simulation. At this point, the simulation is removed from Draft Simulations and is shown under Running Simulations.

Cancel: Clicking the Cancel button under Running Simulations stops the simulation. (It takes a moment for the processes to stop.) After canceling a simulation, you'll see it as canceled under the Completed Simulations list and you can Restart the simulation again with the Restart button.

Restart: Click on Restart under Completed Simulations to restart a canceled or interrupted simulation. (If you need to change an input value, click on Duplicate, at which point you will need to start the simulation from the beginning again.)

Duplicate: Clicking Duplicate from any point in the application makes a copy of the inputs of that simulation and creates a Draft simulation of the same name with those inputs. If you are working in Draft Simulations, a Save is required before you can Duplicate.

Export Simulation: Clicking Export Simulation brings up your file manager so you can save simulation form inputs to an .aasp file. This proprietary file format contains simulation input data that can be imported into ANSYS Additive (Desktop or Cloud). Your geometry selection (i.e., part) and simulation results are not included on the .aasp file. If you are working in Draft Simulations, a Save is required before you can Export.

Import: Click on Import on the dashboard to bring up the file manager and load an .aasp file. This action will populate a new simulation form with saved inputs. Note that the part is not included in saved inputs and will need to be added to the simulation form.

Delete: Clicking the Delete button from either Draft Simulations or Completed Simulations removes the simulation from the application. The operation will delete all metadata and output files. Data will be permanently deleted and is not recoverable.

Save Logs: The Save Logs button under Completed Simulations is needed only if you have a problem with your simulation and you need to contact customer support for a resolution. Clicking Save Logs brings up your file manager and allows you to write a zipped file containing files used for diagnostic purposes. Contact us at the ANSYS Customer Portal.

Where is My Data Stored?

For users of Additive Desktop, your completed simulations, parts, materials, and build files as well as log files and other program data, are stored as "AppData" on your computer. This is usually a hidden file but you can find it by typing `%appdata%` in your Windows File Explorer and then look for the **ansys-additive** folder. You can backup this folder periodically if having data backups is important for your workflow. (%appdata% is an environment variable determined by your operating system.)

ANSYS Additive does not support installation across a network so AppData will be local to your computer.

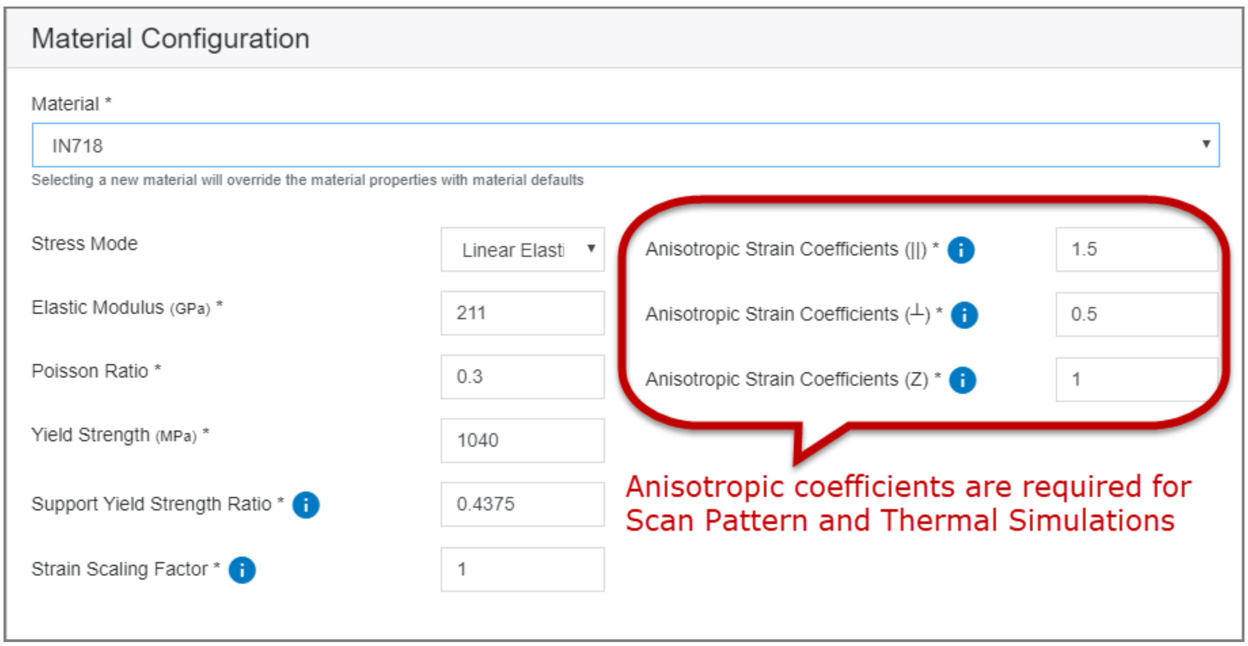
Performing a Scan Pattern Simulation

A Scan Pattern Simulation uses anisotropic strain calculations to improve upon the assumed strain method. Anisotropic strain is rapidly calculated for each powder layer based on the major orientation of the fill scan vectors or the specific scan vector files if loaded through a build file. Then these individual layer strain values are collected and averaged to the voxel size. The predicted strain is then used for a rapid mechanics analysis.

In addition to the standard inputs as described for an Assumed Strain Simulation, there are two unique sets of input required for a Scan Pattern Simulation. These are the anisotropic strain coefficients of your material and the type and print parameters of your 3D printing machine.

Anisotropic Strain Coefficients

The Material Configuration section of the simulation form will change to include anisotropic coefficients if you have chosen a Scan Pattern or Thermal Simulation type.



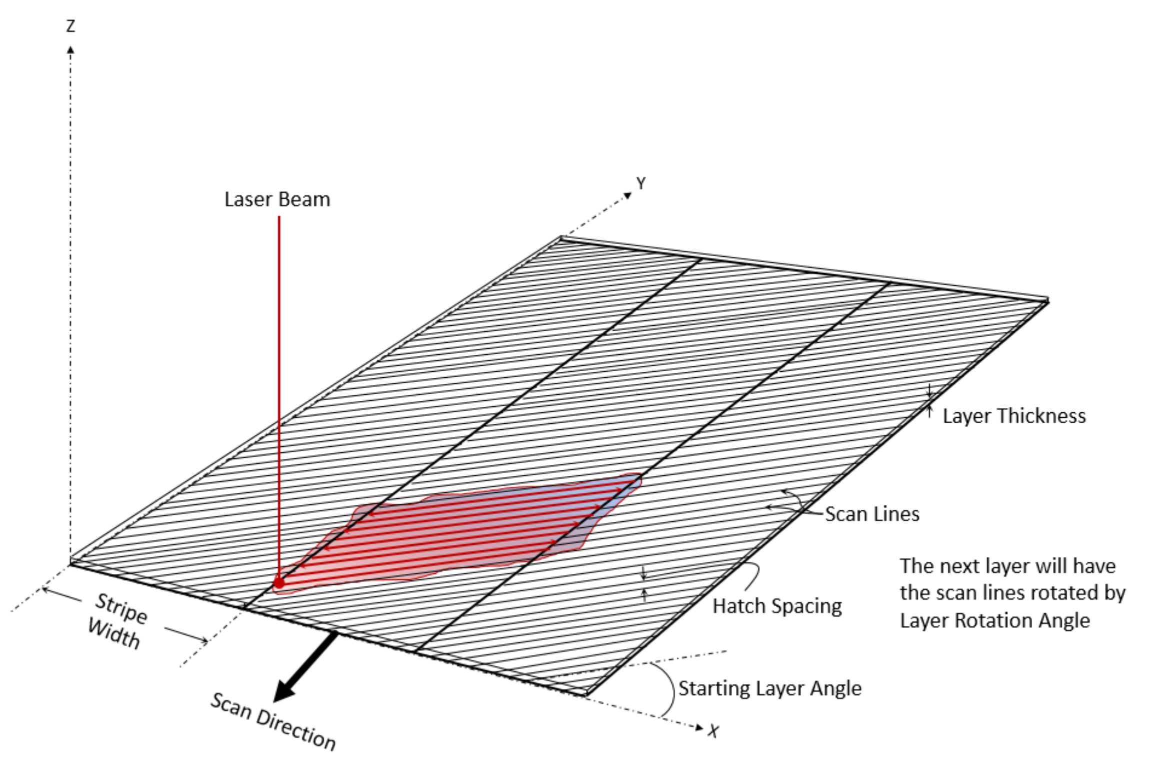
Anisotropic strain coefficients are used to represent anisotropic strain behavior on coordinate systems aligned with the local longitudinal, transverse, and depth scan directions. Positive values result in compressive base strain (contraction), whereas negative values result in tensile strain (expansion). Default values are shown in the following table.

Anisotropic strain coefficient (||) = 1.5	Longitudinal: The predicted strain in the direction that the laser is scanning for the major fill rasters will be multiplied by 1.5
Anisotropic strain coefficient (⊥) = 0.5	Transverse: The predicted strain orthogonal to the direction that the laser is scanning for the major fill

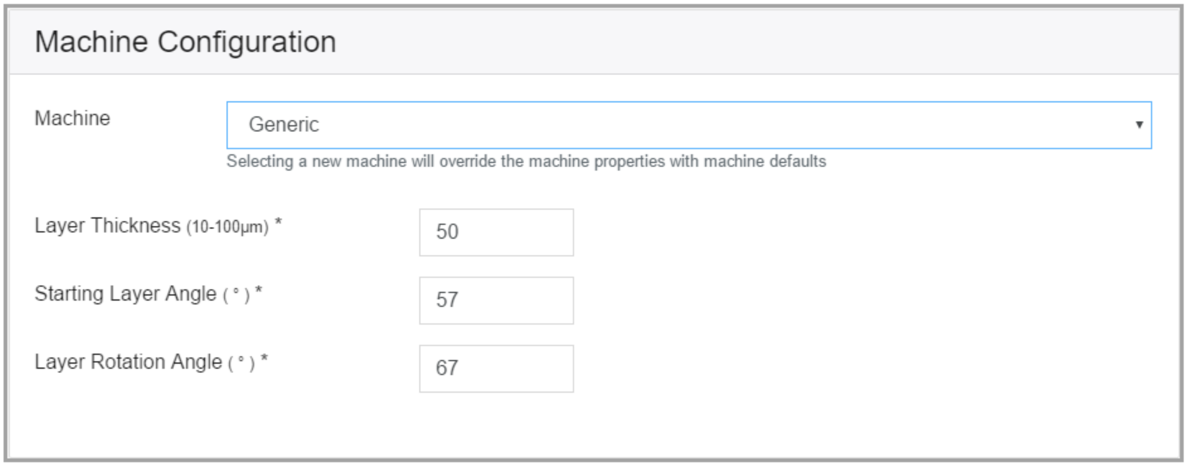
	rasters and in the plane of the surface of the build plate will be multiplied by 0.5
Anisotropic strain coefficient (z) = 1	Depth: The predicted strain in the Z direction will be multiplied by 1

Machine Parameters for a Scan Pattern Simulation

The following figure shows a typical scan pattern. Currently, the Additive application supports a rotating stripe scan pattern. (Checkboard and islands patterns are not supported. Contour scans are not simulated.) Machine parameters required for a Scan Pattern Simulation include Layer Thickness, Starting Layer Angle, and Layer Rotation Angle.



Machine parameters for a rotating stripe scan pattern



Layer Thickness (10-100 µm)

Layer Thickness is the thickness of the powder layer coating that is applied with every pass of the recoater blade. The default value is 50 microns. We recommend that you use the actual thickness unique to your machine.

Starting Layer Angle (°)

This is the orientation of fill rasters on the first layer of the part. It is measured from the X axis, such that a value of 0 degrees results in scan lines parallel to the X axis. The starting layer angle is commonly set to 57 degrees (default).

Layer Rotation Angle (°)

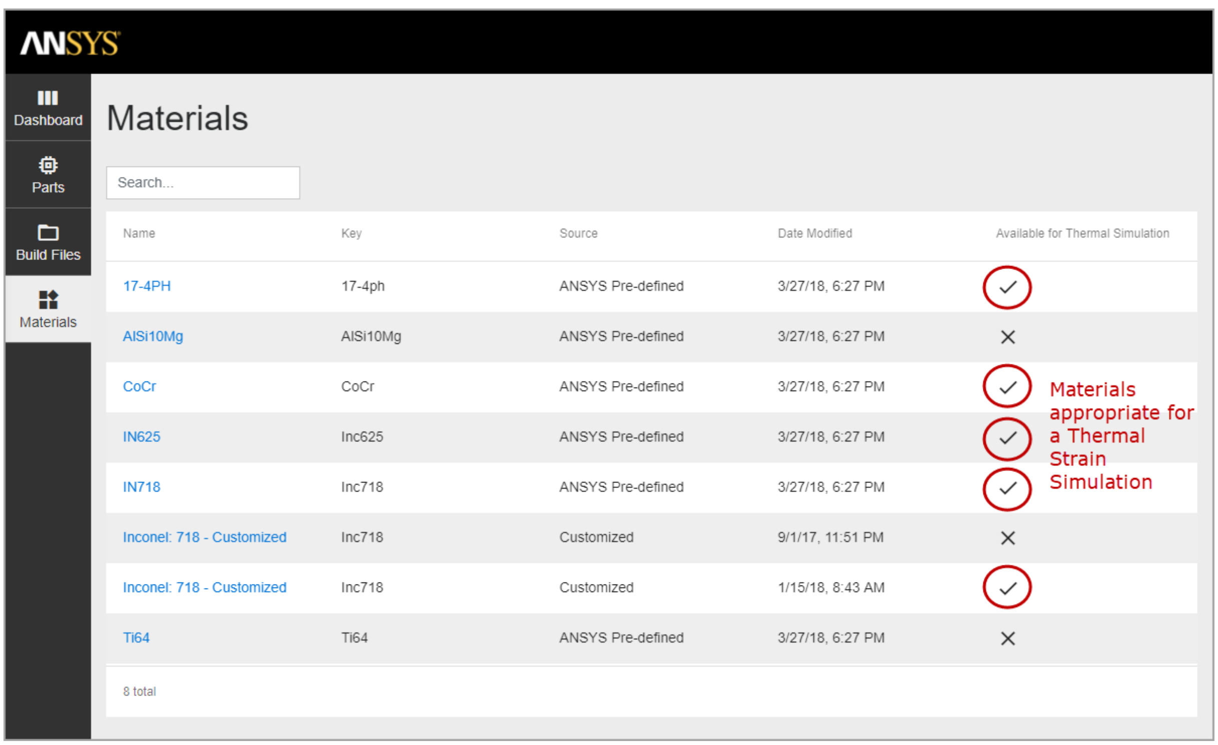
The Layer Rotation Angle is the angle at which the major scan vector orientation changes from layer to layer. This is commonly 67 degrees (default).

Performing a Thermal Strain Simulation

Thermal Strain Simulations provide the highest degree of accuracy by predicting how thermal cycling affects strain accumulation at each location within a part. It uses inherent strain but also implements a thermal ratcheting algorithm to locally modify the inherent strain value. In addition to the standard inputs as described for an Assumed Strain Simulation, you will need to select an ANSYS pre-defined, thermally validated material and define some additional configuration parameters of your 3D printing machine.

Select Material for a Thermal Strain Simulation

ANSYS pre-defined materials are designed to capture the effect of a material's chemical composition, powder-to-liquid and liquid-to-solid state transitions, and high cooling rates. Nonlinear thermo-physical properties such as thermal conductivity, density, and specific heat are essential for capturing solidification phenomena in metal additive manufacturing. At the time of this software release, the materials that have had complete validation tests performed and are recommended for Thermal Strain Simulations are shown with a checkmark as Available for Thermal Simulation in the Materials Library and in the details panel for a material. Only those materials will be available for selection in the Material Configuration section of the Thermal Strain Simulation form.

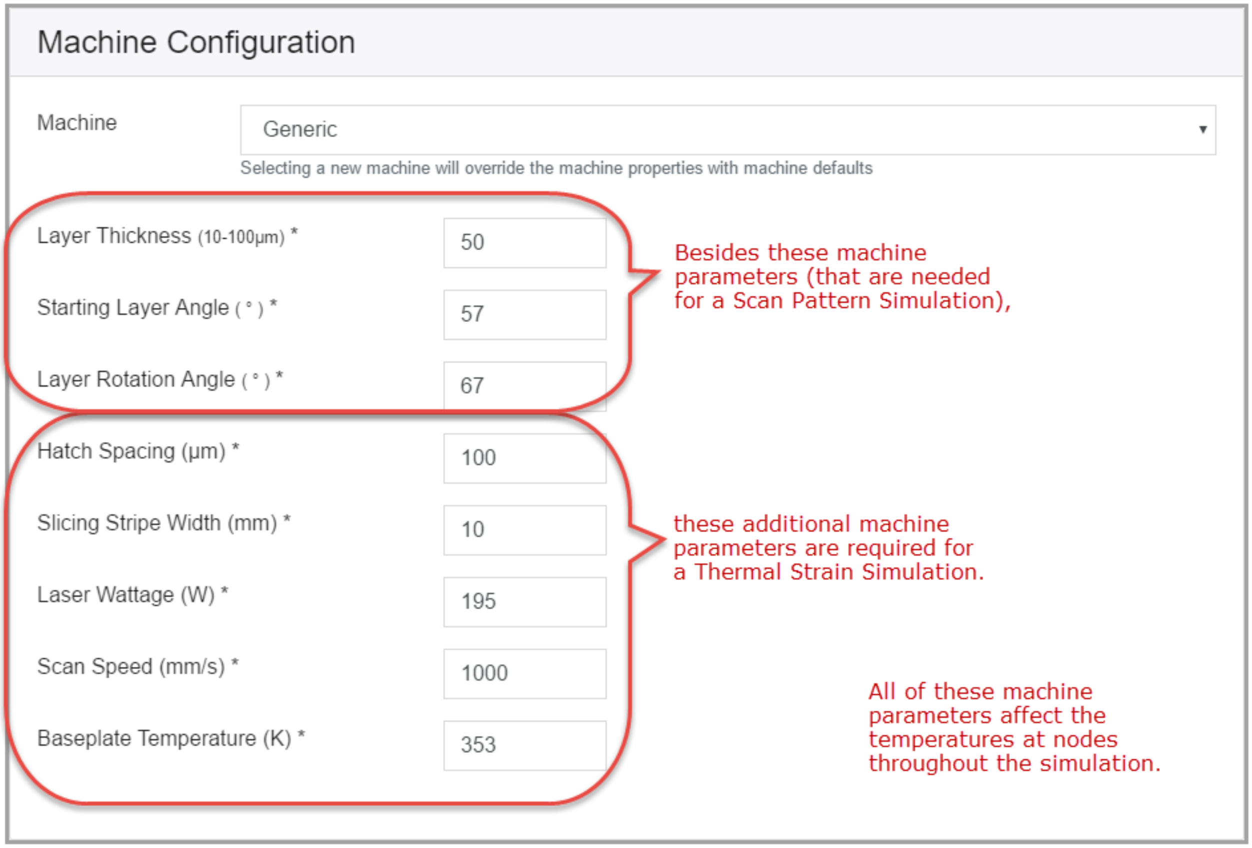


Name	Key	Source	Date Modified	Available for Thermal Simulation
17-4PH	17-4ph	ANSYS Pre-defined	3/27/18, 6:27 PM	✓
AlSi10Mg	AlSi10Mg	ANSYS Pre-defined	3/27/18, 6:27 PM	×
CoCr	CoCr	ANSYS Pre-defined	3/27/18, 6:27 PM	✓
IN625	Inc625	ANSYS Pre-defined	3/27/18, 6:27 PM	✓
IN718	Inc718	ANSYS Pre-defined	3/27/18, 6:27 PM	✓
Inconel: 718 - Customized	Inc718	Customized	9/1/17, 11:51 PM	×
Inconel: 718 - Customized	Inc718	Customized	1/15/18, 8:43 AM	✓
Ti64	Ti64	ANSYS Pre-defined	3/27/18, 6:27 PM	×

8 total

Machine Parameters for a Thermal Strain Simulation

Additional machine parameters required for a Thermal Strain Simulation include Hatch Spacing, Slicing Stripe Width, Laser Wattage, Scan Speed, and Baseplate Temperature.



Hatch Spacing (µm)

This is the 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. The default Hatch Spacing is 100 microns.

Slicing Stripe Width (mm)

In rotating stripe scan patterns, the geometry is broken up into sections, which are called stripes. The stripes are scanned sequentially to break up very long continuous scan vectors. Slicing Stripe Width is commonly set to 10 mm wide (default).

Laser Wattage (W)

This is the power setting for the laser in the machine. Defaults to 195 Watts.

Scan Speed (mm/s)

This is the speed at which the laser scans, excluding jump speeds and ramp up and down speeds. The default value is 1000 mm/sec.

Baseplate Temperature (K)

The controlled temperature of the baseplate. The default value is 353 degrees Kelvin.

Glossary

.aasp File

From "ANSYS Additive simulation package," the .aasp file format is a proprietary file format that contains simulation inputs that can be imported into ANSYS Additive (Desktop or Cloud). When imported, a new simulation form is populated under Draft Simulations. Note that this file does not contain the part or simulation results.

Activity Status

The section of results where graphical indicators show the status of each function within the simulation workflow.

Output

After cutoff displacement

A result file, output in both .vtk and .avz file formats, that contains the voxelized representation of the part with *predicted displacements after the part is cut off the baseplate*. Data on file include magnitude and x, y, and z components of displacement. Included in Output Files when a user has selected to output the displacement after cutoff.

Anisotropic

When the properties of a material vary with different orientations, the material is said to be anisotropic. The mechanical and thermal properties of these materials differ in different directions. (Alternately, when the properties of a material are the same in all directions, the material is said to be isotropic.)

Anisotropic Strain Coefficients

Coefficients used to represent anisotropic strain behavior on coordinate systems aligned with the local longitudinal, transverse, and Z (depth) scan directions. Positive values result in compressive base strain (contraction), whereas negative values result in tensile strain (expansion).

Input

Anisotropic Strain Coefficient (||)

Multiplier on the predicted strain *in the direction that the laser is scanning* for the major fill rasters.

Input

Anisotropic Strain Coefficient (⊥)

Multiplier on the predicted strain orthogonal to the direction that the laser is scanning for the major fill rasters and in the plane of the surface of the build plate.

Input

Anisotropic Strain Coefficient (Z)

Multiplier on the predicted strain *in the Z direction*.

ANSYS Pre-defined Materials

Materials in the Materials Library that are available for use and may not be edited. ANSYS pre-defined materials are designed to capture the effect of a material's chemical composition, powder-to-liquid and liquid-to-solid state transitions, and high cooling rates.

ANSYS Viewer

ANSYS Viewer is an interactive 3D image viewer that is either embedded into your ANSYS application or is available as a downloadable file (on the ANSYS Customer Portal here) to run in your web browser. Designed specifically for sharing and collaboration, ANSYS Viewer enables you to visualize 3D models created in ANSYS CAE software even if you do not have ANSYS software installed. ANSYS Viewer files have a .avz extension. ANSYS Viewer is *embedded* in ANSYS Additive (both Desktop and Cloud) for seamless 3D visualization.

Assumed Strain Simulation

A simulation method that assumes a constant, isotropic strain (inherent strain) occurs at every location within a part as it is being built. This is the fastest simulation method.

.avz File

A file format used by ANSYS Viewer for 3D visualization.

Baseplate

The surface of the 3D printing machine upon which the part is built. Also called the build plate.

Blade Crash

A scenario in which the recoater blade of the printing machine hits into the part already printed, most likely due to distortion of the part as a result of residual stress.

Build File

Build files are unique to each 3D printing machine and are required to be .zip files containing the part .stl file as well as files specifying machine scan vectors.

Build File Library

The repository for Build Files that you have uploaded into the program. Build Files are formatted .zip files written for specific 3D printing machines.

Build File Type

The machine type, such as Additive Industries, Renishaw, or SLM, corresponding to your build file. When uploading a build file, selecting the build file type assures that the appropriate translator will be used.

Output

Compensated Geometry

An .stl file containing the distortion-compensated 3D surface representation (tessellated triangles) of the part while the part is still attached to the baseplate. The compensated geometry is placed flush with the baseplate surface and does not include the offset for supports between the baseplate and the part.

Output

Compensated Geometry (after cutoff)

An .stl file containing the distortion-compensated 3D surface representation (tessellated triangles) of the part after cutoff from the baseplate.

Completed Simulation

Simulations that have either completed or that have been canceled or failed due to error. These simulations are no longer running. Click on a simulation in the Completed Simulations area of the dashboard to see simulation results along with input parameters and log files for that simulation.

.csv File

From "comma separated values," this is a file that allows data to be saved in a table-structured format. Traditionally, a .csv file is in the form of a text file containing information separated by commas, hence the name.

Customized Materials

Materials that have been edited from the original ANSYS pre-defined materials are labeled as customized materials.

Dashboard

The main area, or “home,” of the Additive user interface that shows an overview of Draft Simulations, Running Simulations, and Completed Simulations.

Deposit Layer

The layer of metal powder spread over the baseplate or melted material. Simulations begin at a deposit layer of 0 and layers are numbered sequentially thereafter as each successive layer is added.

Detect Potential blade crash due to distortion

An output option that activates the blade crash detection feature of Additive. If this box is checked, there will be a check to determine if the +Z displacement at every point in each new layer exceeds a threshold value. Locations of potential blade crash and associated displacement values are provided in a .csv output file, as well as in the On-Plate Residual Stress/Distortion and Layerwise .vtk files.

Distortion

The deformation that occurs as a result of residual stress in a part.

Distortion Compensated .stl File

An output option that activates the distortion compensation feature of Additive, which predicts the location and magnitude of distortion and then reverse distorts the original .stl file. When you build your part using the compensated geometry, the result will be closer to the original design.

Distortion Compensation

One of the functions within the simulation workflow responsible for reverse distorting the original .stl file to compensate for the effects of predicted distortion. Shown with a status indicator in the Activity Status area of Running and Completed Simulations.

Domain

The entirety of the voxels in the simulation as defined by the bounding dimensions of the part plus the support height voxels (if any). Some voxels are solid part material, some are support material and some are powder.

Mat'l

Elastic Modulus

A material property that is a measure of the material's stiffness. Elastic Modulus, or Young's Modulus (E), describes tensile elasticity, or the tendency of an object to deform along an axis when opposing forces are applied along that axis; it is defined as the ratio of tensile stress to tensile strain. Input in units of MPa.

Estimated Memory Usage

For Additive Desktop, an estimate of the memory required to run the simulation based on the dimensions of the part and voxel size. This estimate is provided in the Geometry Selection section of the simulation form as soon as you add a part. The estimate is calculated without considering support generation. (Estimated memory is not applicable to Additive Cloud.)

Experimental

A label applied to any new feature that has not been fully validated, but that we feel is stable and useful for users.

Fill Rasters

Laser scans associated with the interior regions of the part.

Generic

The Generic machine selection in the Machine Configuration section of the simulation form applies default input parameters that we have determined to be appropriate through initial testing. The default machine parameters most closely approximate an EOS machine, but may be similar to most commonly used metal laser powder bed fusion machines.

Geometry Selection

This section of the simulation form where you select a part (or a build file) for simulation. Parts (or build files) must first have been uploaded to the Parts (or Build File) Library.

Output

Geometry with Distortion

A .vtk file containing the 3D surface representation (tessellated triangles) of the original, undistorted part with predicted displacements at the end of the build while part is still attached to the baseplate. The geometry does not include the offset for supports between the baseplate and the part.

Mat'l

Hardening Factor

Also known as strain hardening coefficient, a material-specific factor used to calculate the slope of a material's stress-strain curve (E_p) above the material's Yield stress.

Input

Hatch Spacing

The 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. The default Hatch Spacing is 100 microns.

High Strain Areas

An output option that 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.

Output

High Strain Regions

A .csv file containing a list of high strain warning areas during the build with their associated strain values.

Inherent Strain

The residual, irrecoverable strain caused by a heat source melting or partially melting a material in a very localized spot such that the thermal contraction of cooling solidified material is constrained by the surrounding material. Typically associated with a welding process.

Isotropic

Isotropic materials have identical properties in all directions. For an isotropic medium, the stiffness tensor has no preferred direction: an applied force will give the same displacements (relative to the direction of the force) regardless of the direction in which the force is applied.

J2-Plasticity

One of the options for stress mode in ANSYS Additive strain-based simulations. J2-plasticity is a part of plasticity theory that applies best to ductile materials, such as some metals. Ductility is a measure of a material's ability to undergo significant plastic deformation before rupture. J2-plasticity uses Von Mises stresses to reduce the stress levels when strain values exceed elastic strain with strain hardening algorithms included. Simulations run longer with the J-2 Plasticity option but stress and strain results will be more accurate.

Input

Laser Wattage

The power setting for the laser in the machine.

Laser-Powder Bed Fusion (LPBF)

In Additive Manufacturing, a method of Powder Bed Fusion (PBF) that involves spreading a layer of metal powder and then using a laser to melt or fuse metal powder material together to build a part. This is the method being simulated in ANSYS Additive.

Input

Layer Rotation Angle

The angle at which the major scan vector orientation changes from layer to layer. This is commonly 67 degrees.

Input

Layer Thickness

The thickness of the powder layer coating that is applied with every pass of the recoater blade. The default value is 50 microns. We recommend that you use the actual thickness used for your machine and build material.

Output

Layerwise .vtk Files

A series of .vtk files that show voxel representation of the part layer-by-layer during the build. You will have one .vtk file for every voxel layer. The .vtk files are compressed into a .zip file.

Linear Elastic

An assumption that a material will undergo strain linearly proportional to the magnitude of applied stress and that the material will return to its original shape when the loads are removed. (A simple straight line on a stress strain curve.) One of the options for stress mode in ANSYS Additive strain-based simulations. Using this option can result in a higher maximum stress value for a given strain beyond the yield point for the material. This overprediction may not be realistic for parts with larger distortions. Stresses and strains may be unrealistically high. Distortion values will generally be accurate, however, so the linear elastic option may be *useful for analyzing distortion trends* while the part is still on the baseplate. The simulation *runs faster* with the linear elastic option and is a good choice if you are just beginning with Additive and you want to run practice simulations with default options.

Logs

The section of results where the time-stamped log entries are collected. Reading log messages is useful for following the progress of a simulation.

Machine Configuration

The section of the simulation form where you identify machine and process parameters. You will see this section for Scan Pattern and Thermal Strain simulations only. Assumed Strain simulations do not require inputs related to the 3D print machine.

Material Configuration

The section of the simulation form where you identify the material. When you select a material, properties associated with that material are automatically populated and any related background information is tied to the simulation.

Material Library

The repository for saved materials, including ANSYS pre-defined materials and user customized materials.

Input

Maximum Wall Distance

A parameter used for the optimized thin wall supports. It is the allowed maximum distance between two neighboring support walls. Regardless of the predicted stress level in the support structure, the walls in supported regions will be spaced not more than this value. Too large of a wall distance might result in failures such as the part breaking away from the support, the development of cracks at the bottom of the part, or drooping between support hatches. When a laser scans a relatively large area of powder where the support wall distance is too wide, cracking might happen since powder has no strength to hold the solidified part in place. The excessive distortion might cause blade and part collision. We recommended that Maximum Wall Distance should not exceed 2 mm when a single bead support wall is used.

Input

Maximum Wall Thickness

A parameter used for the optimized thick wall supports. Support wall thicknesses will not exceed Maximum Wall Thickness even in areas of high stress. Defaults to 1 mm.

Mechanics Solver

One of the functions within the simulation workflow responsible for calculating displacements and stresses. Shown with a status indicator in the Activity Status area of Running and Completed Simulations.

Input

Minimum Overhang Angle

The overhang angle is measured from the powder bed surface (horizontal = 0) up to the surface of the part. Any point on the surface of the part having an angle *less than* the Minimum Overhang Angle will be supported. Recommendation: Avoid using a value that is the same as the angle of the geometry of your part, as it can cause asymmetric support structures due to finite rounding errors.

Input

Minimum Support Height

The height (in mm) that the part will be elevated off the baseplate. If you set a value of 3mm then the part will be elevated such that the lowest point on the part is at least 3mm above the baseplate. This value should be set with consideration of approximating a realistic support height along with care about how many voxels must be created to add additional support height. It is recommended that this value be set as low as is realistic for each simulation.

Input

Minimum Wall Thickness

A parameter used for the optimized thick wall supports. Minimum wall thickness is the thinnest possible support wall that a machine will build under certain process parameters. It is usually the thickness of a single bead scan.

Output

On-plate stress/displacement

A result file, output in both .vtk and .avz file formats, that contains residual stresses and displacements of the part prior to its removal from the baseplate. Both end state stresses and maximum stress during the build are contained in this file, as well as potential blade crash locations and high strain areas if those output options are selected.

Output

Optimized Thick Wall Support

An .stl file of the auto-generated support structure as defined by the thick wall input parameters. The thick wall supports are uniformly spaced, but wall *thickness* is varied based on the residual stress levels predicted.

Output

Optimized Thin Wall Support

An .stl file of the auto-generated support structure. The thin wall supports are of a uniform wall thickness, but *wall spacing* is varied based on the residual stress levels predicted in the part.

Overhang Angle

Angle measured from the horizontal baseplate (0 degrees) to the surface of the part. Any surface measuring *less than* the Minimum Overhang Angle will be supported.

Part

The geometry for the simulation as defined by an .stl file that must be uploaded to the Parts Library. This is the most common method for defining geometry.

Input

Part Strain Threshold

An input parameter required when you choose the High Strain Areas output option. Defined as the percentage strain in the *part* above which strain will be considered critical. Defaults to 20%. (Critical regions are shown in the On-plate stress/displacements, Layerwise .vtk, and High Strain Regions output files.)

Parts Library

The repository for all parts (as .stl files) that have been uploaded into the system. Individual .stl files must be smaller than 100MB.

Mat'l

Poisson's Ratio

A material property that is the ratio of transverse contraction strain to longitudinal extension strain in the direction of stretching force. Tensile deformation is considered positive and compressive deformation is considered negative.

Output

Positioned Part

An .stl file of input geometry (non-compensated) positioned into its start location and orientation, that is, offset to account for supports between the baseplate and the part.

Output

Potential blade crash locations

A .csv file that contains locations of all potential and likely blade crashes and the magnitude of the +Z displacement at those points.

Residual Stress

Residual stress is the internal stress distribution locked into a material after all external loading forces have been removed. Stresses are a result of the material obtaining equilibrium after it has undergone elastoplastic deformation. In additive manufacturing processes, a part undergoes repeated expansion and contraction from the heating and cooling of the build process. This repeated heating and cooling can lead to residual stress—a result that shows up as cracks, warpage, and other forms of deformation in an object.

Result ID

A unique identifier for each simulation. When reporting a problem or looking for clarification on a specific simulation, this is the number that needs to be included with a support request. You will see the Result ID in the Overview section of Running and Completed Simulations.

Reverse Distort

Predicted distortion of a part is automatically passed to a distortion compensation function providing you with an .stl file that is pre-distorted, or reverse distorted, to compensate for process generated distortion.

Running Simulation

A simulation that is either actively running or has been queued to begin as soon as resources are available. Click on a simulation in the Running Simulations list on the dashboard to see input parameters, activity status, and log files for that simulation.

Scale Factor

An input parameter required when you choose the Distortion Compensated .stl File output option. The Scale Factor will change the magnitude of the distortion applied to the original .stl file. A Scale Factor of 1 (default) will create an .stl file with distortion compensated by the same magnitude as the simulated results. A Scale Factor < 1 will compensate less than the simulation-predicted magnitude and a value > 1 will compensate more than the simulation-predicted magnitude.

Scan Pattern Simulation

This simulation type uses the same average strain magnitude as in the Assumed Strain Simulation but it subdivides that strain into anisotropic components based on the local orientation of scan vectors within the part. This strain mode requires the creation of scan vectors using user-provided scan settings or by reading scan vectors from a supported machine's build file. This extra step results in a small, increased amount of simulation time compared to Assumed Strain simulation. For parts where the scan pattern is randomized, scan pattern and assumed strain should give a similar answer. For parts where the scan patterns are aligned, scan pattern strain will result in a more accurate prediction.

Scan Speed

The 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.

Scan Vector

Direction and velocity of one laser scan across the part. Multiple scan vectors make up a layer's scan pattern.

Simulate with Supports

A checkbox (on/off) option in the Supports section of the simulation form that controls whether supports are automatically generated in the simulation.

Simulation Form

All simulations are initiated from a simulation form. It holds the inputs and selections for your simulation. Saved simulation forms are shown under Draft Simulations in the dashboard. Once you start a simulation, it is removed from Draft Simulations (i.e., it is no longer a "draft") but all your input options are shown for Running Simulations and Completed Simulations.

Simulation Overview

A section of the simulation results where you can quickly see a summary status of the simulation.

Slicer

One of the functions within the simulation workflow responsible for "slicing" the domain into scan vectors according to the scan pattern input parameters. Shown with a status indicator in the Activity Status area of Running and Completed Simulations.

Input

Slicing Stripe Width

When using the stripe pattern for scan strategy, the geometry can be broken up into sections, which are called stripes. The stripes are scanned sequentially to break up what would otherwise be very long continuous scan vectors. Stripe Width is commonly set to 10 mm wide.

Start (button)

Starts a simulation from the simulation form. All your input options are stored when you run a simulation so that you may see your options at any time when you click on a simulation in the Running Simulations and Completed Simulation areas of the dashboard.

Input

Starting Layer Angle

The orientation of fill rasters on the first layer of the part. This is currently 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.

Status

The status of a part indicates the readiness of the part for running a simulation. When you first upload a part it will show as "uploaded", but there are some basic pre-processing steps that are completed at this time, so the part is not available for a simulation until "Available" appears in the status. (As a common practice, you can upload a part and then go to a simulation template and by the time the template is ready to run the part will usually be available. When uploading large parts then there is a chance that you may need to wait for upload to occur.)

.stl File

From "stereolithography," this is a 3D rendering file that approximates the surfaces of a solid model with triangles. .stl files describe only the surface geometry of a three-dimensional object without any representation of color, texture or other common CAD model attributes. The .stl format specifies both ASCII and binary representations. Binary files are more common, since they are more compact.

Strain Mode

Strain mode refers to the simulation type (Assumed Strain, Scan Pattern, or Thermal Strain).

Input

Strain Scaling Factor

The Strain Scaling Factor (SSF) is a calibration factor that you may use to improve the accuracy of your simulations. This value is a direct multiplier to the predicted strain values. Values greater than 1 will amplify displacement and stress and values less than 1 will reduce them. The default Strain Scaling Factor is 1. A simple calibration test should be performed to account for variations in material, machine, and temperatures as well as simulation assumptions (such as simulation type and stress behavior). Calibration procedures are available online here.

Strain Warning Factor

An input parameter required when you choose the High Strain Areas output option. This factor is multiplied by both the Support Strain Threshold and the Part Strain Threshold to define limits where strain is labeled as a warning (i.e., approaching critical range). Defaults to 0.8.

Stress Mode

An input option that allows you to choose between linear elastic or elastoplastic (using the J2-Plasticity model) material behavior in calculations of stress.

Support (or Support Structure)

Support structures act as fixtures to anchor a part to a baseplate during part fabrication. In an ideal scenario, the support density should be as low as possible so that less material is consumed and supports can be easily removed. However, if the support density is too low, supports can fail due to the intense strain resulting from thermal stress accumulation in the part. The Additive application uses predicted residual stress accumulation as criteria for support generation.

Support Factor of Safety

An input a parameter that drives the strength of the automatically generated optimized support structures. If you would like the supports to withstand 2x the expected load, then you would enter a 2 in this field and the predicted strength of the auto-generated support structure would be double the predicted stress. The strength of the support structure is driven by the number and thickness of support walls that are generated. Defaults to 1.

Support Optimization

One of the functions within the simulation workflow responsible for automatically generating supports. Shown with a status indicator in the Activity Status area of Running and Completed Simulations.

Support Strain Threshold

An input parameter required when you choose the High Strain Areas output option. Defined as the percentage strain in the *supports* above which strain will be considered critical. Defaults to 10%. (Critical regions are shown in the Supports stress/displacement and High Strain Regions output files.)

Support Yield Strength Ratio

The Support Yield Strength Ratio is a factor that is used in the simulation assumptions to assign a strength to the support material as compared to the solid material. This factor affects yield strength and elastic modulus of the support material. For example, a value of 1.0 results in a support strength equal to the solid material while 0.5 is half the strength of the solid material. The default value is 0.4375. This default was determined by studies where the support strength for default supports built on an EOS M270 machine were tested and compared to solid material built on the same machine.

Output

Supports Stress/Displacement

A result file, output in both .vtk and .avz file formats, containing the voxelized representation of the support structure with predicted displacements and stresses at the end of the build (i.e., end state) while the part is still attached to the baseplate.

Tags

Tags are used throughout the Additive application to provide optional input for reference and searching criteria.

Thermal Simulation

This is the method for calculating the thermal interaction of the laser and the material at every point in a part throughout the entire build. This method takes much longer than either of the other simulation methods, but is a much higher fidelity result.

Thermal Solver

One of the functions within the simulation workflow responsible for calculating inherent strain fields dependent upon scan patterns (Scan Pattern simulation), or scan patterns and thermal history (Thermal Strain simulation). Shown with a status indicator in the Activity Status area of Running and Completed Simulations.

Input

Threshold Scaling Factor

An input parameter required when you choose the Detect potential blade crash due to distortion output option. This factor is used to modify blade crash calculations so that you can allow for flexibility in the recoater blade. Defaults to 1. (Potential blade crash locations are shown in the On-plate stress/displacements, Layerwise .vtk, and Potential blade crash locations output files.)

Title

The name used for a simulation. Required input on a simulation form.

Triangle Count

The number of triangular tessellation elements that define the outer surfaces of your imported .stl geometry, making up the 3D representation of the part. You will see the triangle count for a part on the detailed description of each part in the Parts Library.

Output

Uniform Thin Wall Support

An .stl file of the geometry-based auto-generated support structure. The uniform thin wall supports use a uniform wall thickness and spacing and are strictly based on geometry. These supports are not to be used as stress optimized supports and are not recommended to use in building parts (use the optimized supports instead).

Version

Unique identifier of the release of the ANSYS Additive application. You can find the version number under Help → About.

Volume (mm^3)

The volume of the part is calculated based upon a rough estimation of the part.

Voxel

A hexahedral (cubic) element used in the finite element method. When combined, voxels define the domain of the geometry.

Voxelization

One of the functions within the simulation workflow responsible for creating the voxelized geometry, that is, dividing the domain into voxels for simulation. Shown with a status indicator in the Activity Status area of Running and Completed Simulations.

Input

Voxel Size

The length of any side of the voxel (cubic element), specified in millimeters.

.vtk File

From "Visualization Toolkit," the .vtk file format is an open source specification for storing 3D computer graphics, images, and visualization data. A right-handed cartesian coordinate system is used.

.vtk to .avz Conversion

One of the functions within the simulation workflow responsible for converting generic-format results files to a format appropriate for ANSYS Viewer. Shown with a status indicator in the Activity Status area of Running and Completed Simulations.

Mat'l

Yield Strength

The material property defined as the stress, in MPa, at which a material begins to deform plastically. Prior to the yield point the material will deform elastically and will return to its original shape when the applied stress is removed. Once the yield point is passed, some fraction of the deformation will be permanent and non-reversible.

Mat'l

Young's Modulus

Also known as the elastic modulus, Young's modulus is a mechanical property of linear, elastic solid materials and is a measure of their stiffness. It defines the relationship between stress (force per unit area) and strain (proportional deformation) in a material.