SIEMENS

NX 11.0.2 Fixed Plane Additive Manufacturing Help

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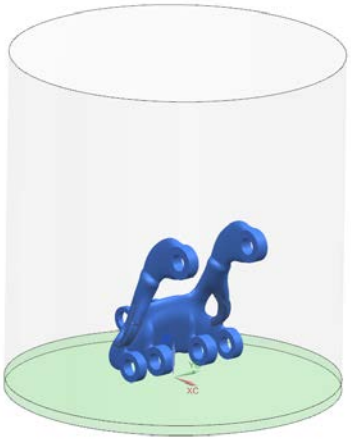
Fixed Plane Additive Manufacturing

The Additive Manufacturing application offers a set of data preparation tools to support the additive manufacturing process. This application is focused on two main additive manufacturing techniques:

1. Material Extrusion (e.g. Fuse deposition modelling)
2. Powder Bed Fusion (e.g. Multi Jet Fusion, Direct Metal Laser Sintering, Selective Laser Sintering, and Electron Beam Melting).

The application is powered by Materialise.



The software allows you to:

- Define 3D printer characteristics (e.g. build tray geometry)
- Optimize part locations in the build tray.
- Prepare parts for 3D printing by generating support structures.
- Develop an optimal build strategy (e.g. build speed, material consumption).
- Integration with numerous 3D printers: using Materialise build processors we offer seamless integration with numerous machines from different machine vendors.

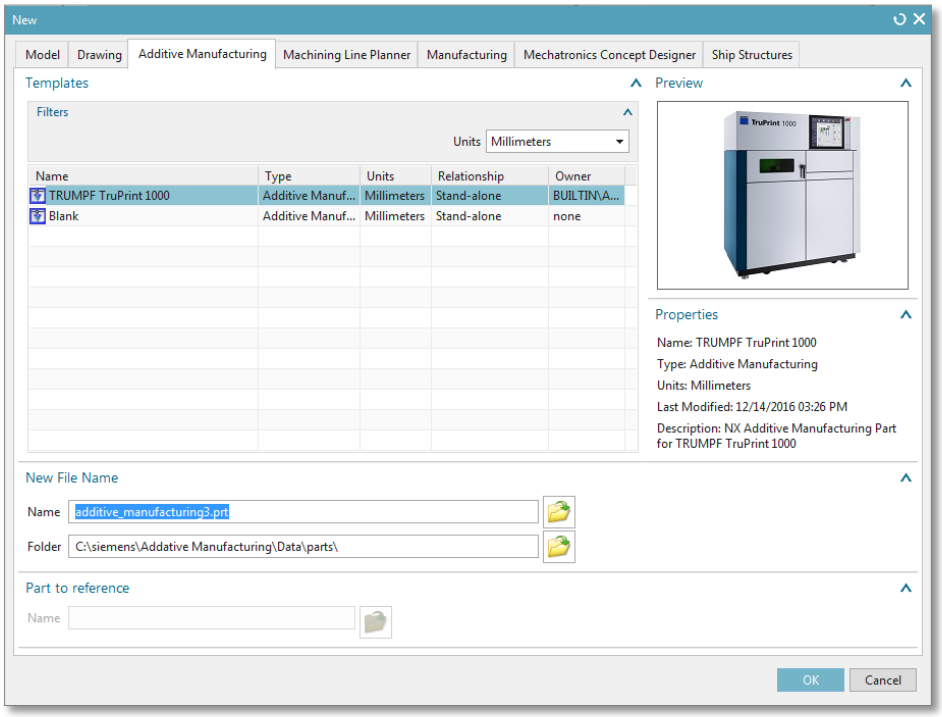
Where do I find it?

Command Finder	**Additive Manufacturing**

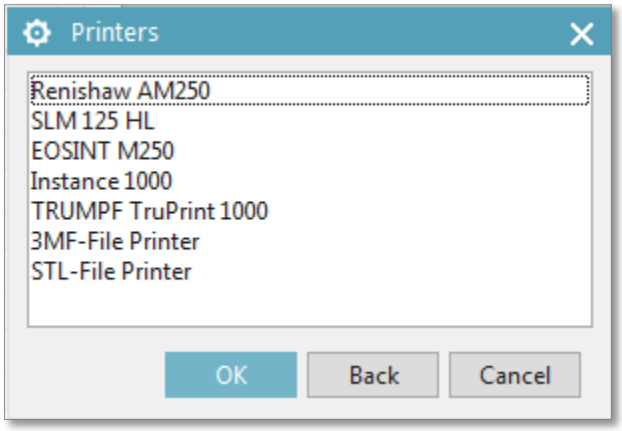
Additive Manufacturing workflow

The following is the basic workflow for preparing a build tray for 3D printing:

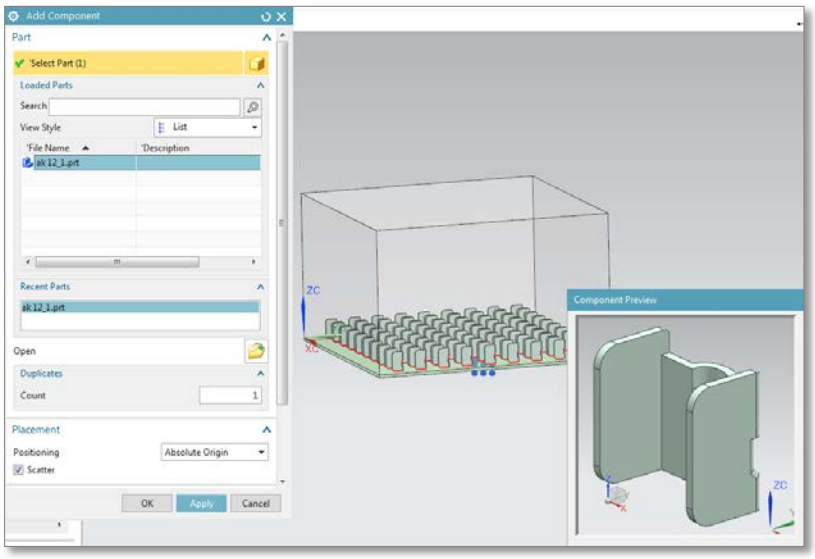
1. Create new Additive Manufacturing part by using the **New…** command.



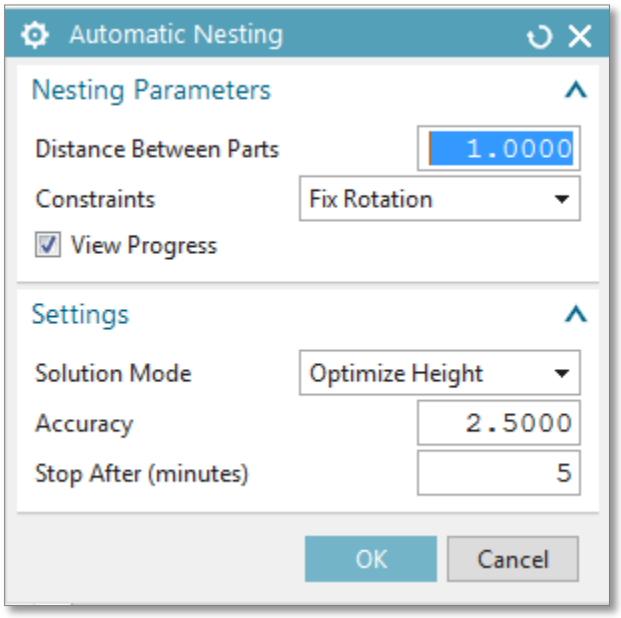
2. Use the **Select 3D Printer** command to set the target 3D printer.



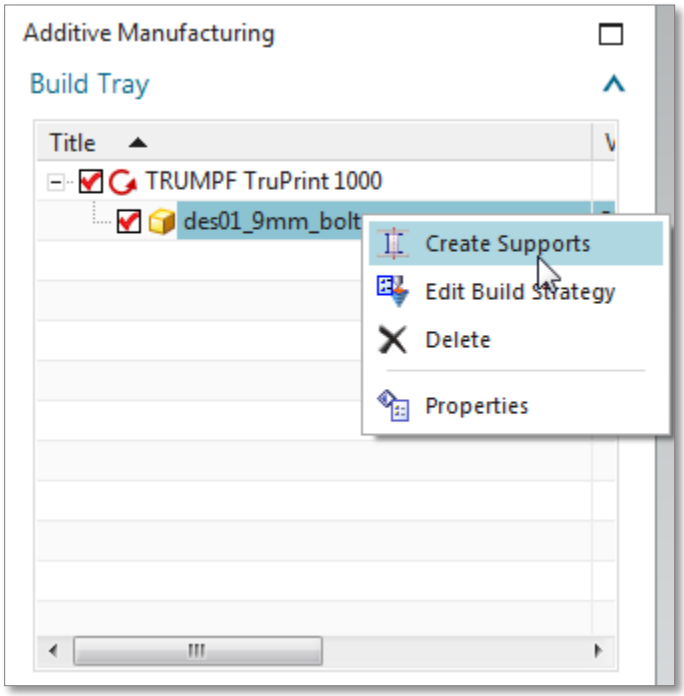
3. Add parts to 3D printer build tray using the **Add** command.



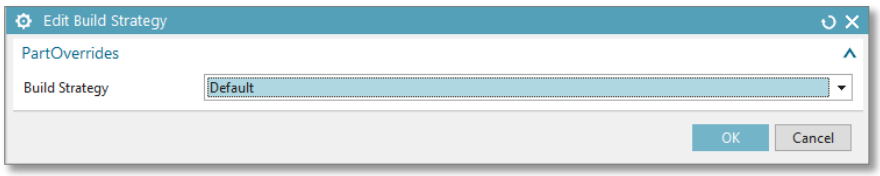
4. Optimize part locations using the **Pattern Component** or **Automatic Nesting** commands.



5. Automatically generate support structures to the loaded parts using the **Create Supports** command.



6. Using the **Properties** command, optimize the support structure by updating the support generation parameters.
7. Use the **Edit Build Strategy** command to define and assign specific build strategy to the build tray and parts.



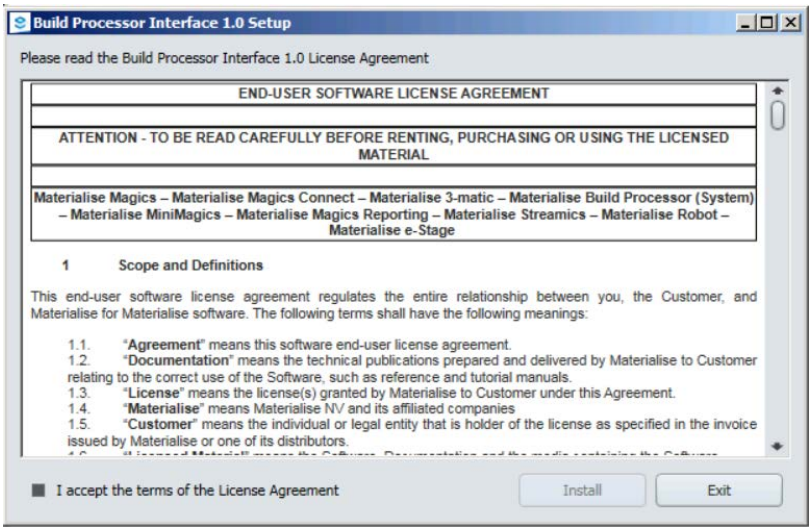
8. Generate machine's specific input files using the **Generate** command.
9. View slices and hatches using the **View Slices** command (if supported by vendor).

Build Processor installation

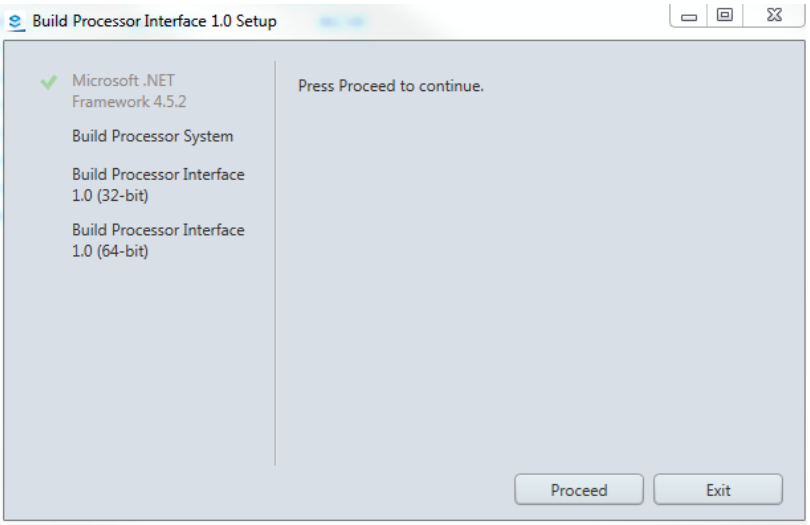
In order to communicate with a specific machine from different vendors, one has to install a dedicated build processor for the required machine.

The following procedure describes the required steps for the build processor installation:

1. Install Build processor interface:
   A. Run $UGII_BASE_DIR\ mach\auxiliary\mfgam\BuildProcessorInterface.exe
   B. Accept the License Agreement terms and click Install (no license is required).



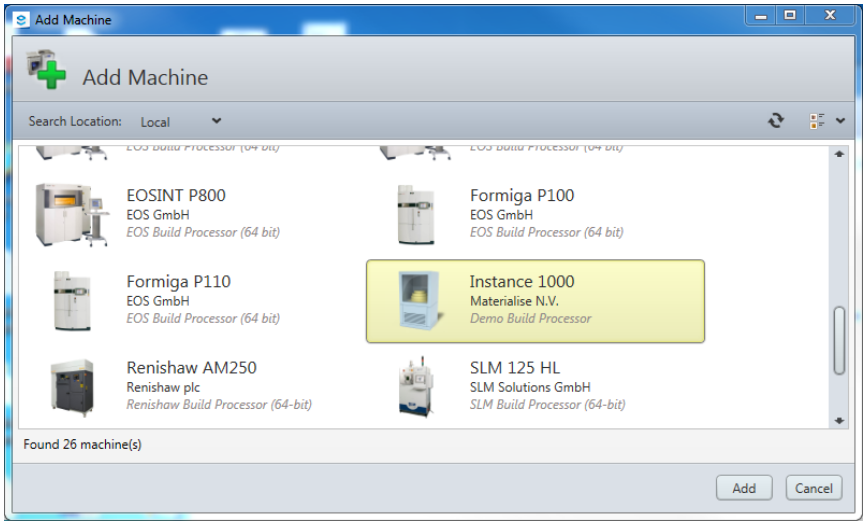
   C. Click **Proceed** to start installation.



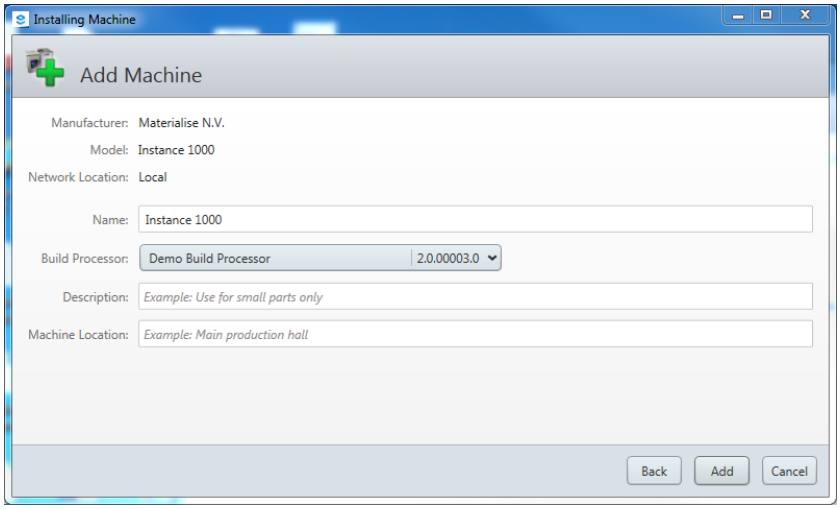
   D. Click **Finish**.

2. Install the specific build processor purchased from Materialise.
3. If supported, install the vendor's slice viewer.
4. Restart the machine.
5. Add the machine to the build processor manager:
    A. Launch the Build Processor Manager application.
    B. Click the **Add a Machine** command.

C. Select the required machine and click **Add**.



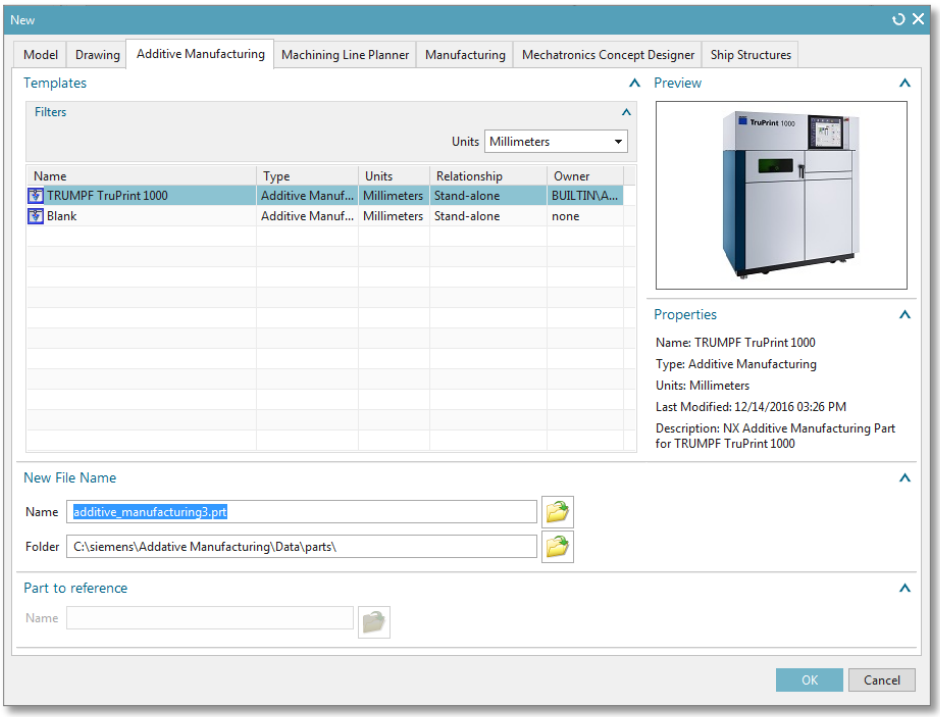
D. View the machine details and click **Add**.



E. Click **Cancel**.

F. Activate the build processor using the license key you got from Materialise.

Create New Additive Manufacturing file



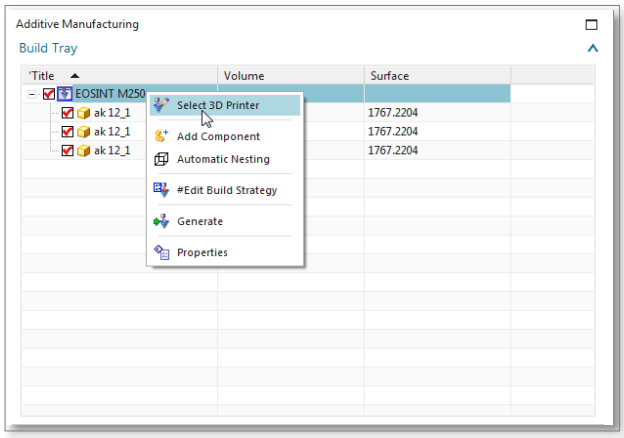
Use the **New** command to create a new Additive Manufacturing file.

The Additive Manufacturing file allows you to prepare your design for the 3D printer: define the printer and its parameters, position your parts on the build tray, create support structures, and generate the machines input file.

In order to create a new Additive Manufacturing file:

1. Choose **File** tab → **New**.
2. Ensure that the **Additive Manufacturing** tab is selected.
3. Select the Additive Manufacturing template according to your machine vendor.
4. Enter a file name and a folder in the **New**, **File**, and **Name** fields.
5. Click **OK**.

Select 3D Printer

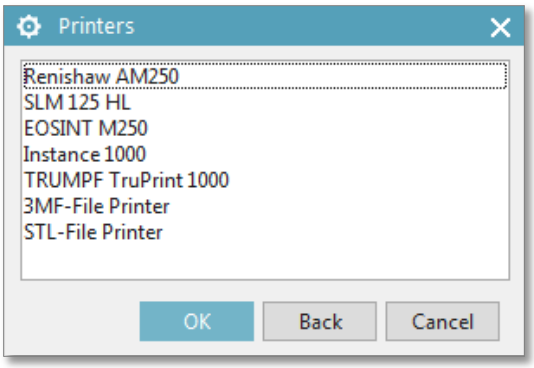


Use the **Select 3D Printer** command to define the specific 3D printer you are going to use in order to build your assembly. Setting the printer will adjust the build tray and build strategy properties.

The command allows the user to select a 3D printer according to the installed build processors.

In order to select a 3D Printer:

1. In the Additive Manufacturing Viewer, right-click on the build tray.
2. Select the **Select 3D Printer** command from the right-click menu.
3. Select the target printer.



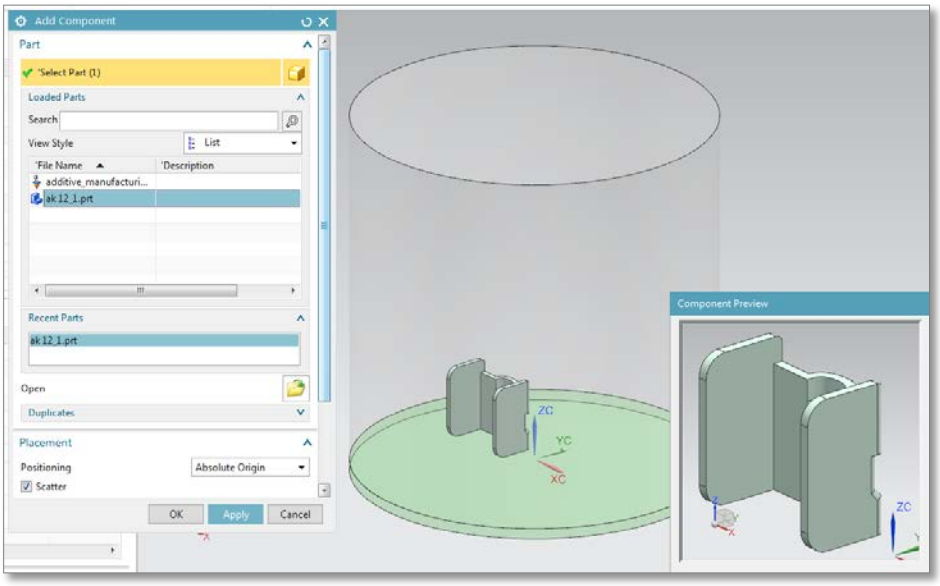
4. Click **OK**.

Where do I find it?

Application	Additive Manufacturing
Prerequisite	Additive Manufacturing part file
Additive Manufacturing Navigator	Build tray Right click menu

Add component

Use the NX Assemblies **Add** command to add new components that you would like to the build tray.



In order to add components to the build tray:

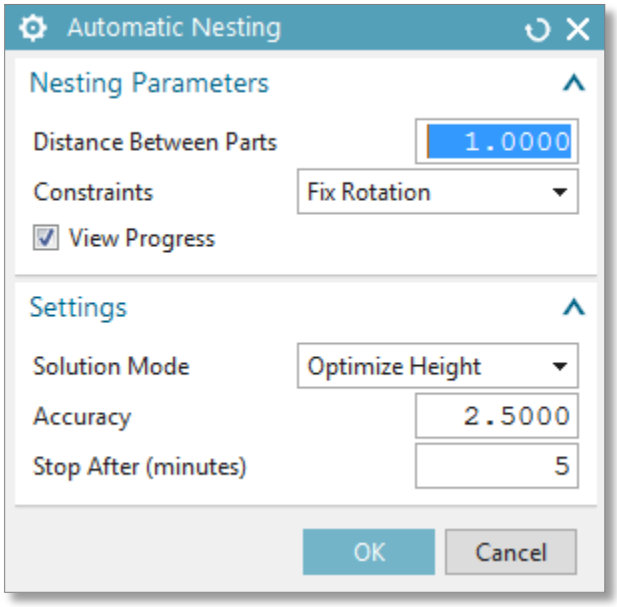
1. Create New Additive Manufacturing part file.
2. In the Ribbons Home Tab click on the **Add** command.
3. Select the NX CAD file (.prt) using the **Open…** command.
4. Place the part using the **Placement** options.
5. Click **OK**.

For additional information please refer to NX documentation.

Where do I find it?

Application	**Additive Manufacturing**
Prerequisite	**Additive Manufacturing part file**
Additive Manufacturing Navigator	**Build tray Right Click menu**

Automatic Nesting



Use **Automatic Nesting** to automatically position the parts in the build tray in an optimal way. The **Automatic Nesting** command helps to maximize the 3D printer output by saving powder and time.

You can do the following with this command:

- Increase nest density and automatically pack more parts into the same build tray.
- Decrease your build time by minimizing build height.
- Reduce build failures by avoiding collisions and interlocking.

In order to use the **Automatic Nesting** command:

1. Create New Additive Manufacturing part file.
2. Add more than one component to the build tray.
3. Select the build tray in the Additive Manufacturing viewer.
4. In the build tray, right-click and select the **Automatic Nesting** command from the menu.
5. Set the nesting parameters.
6. Click **OK**.

Where do I find it?

Application	**Additive Manufacturing**
Prerequisites	• **Additive Manufacturing part file** • **More than one part were added to the build tray**
Additive Manufacturing Navigator	**Build tray Right Click menu**

Automatic Nesting dialog box

Nesting parameters		
Set geometric constraints to the automatic nesting solution.		
Distance between parts	Required distance between the parts. The minimum value depends on the specific build tray volume.	
Constraints	**Free**	Parts are free to rotate and translate.
	Fix Z Direction	Parts are free to translate, rotate around z and rotate 180 degrees around other axes.
	Fix Bottom Plane	Parts are free to translate and to rotate around z only.
	Rotate 180	Parts are free to translate and rotate 180 degrees over any axis.
	Fix Bottom And XY	Parts are free to translate, rotate 180 degrees around z only.
	Fix Rotation	Parts are free to translate only.
	Fixed	Parts are fixed in place and will not be optimized.
View Progress	Review the nesting process in the graphics.	
Settings		
Selection Mode	**Optimize Height**	Minimize building height, to minimize build times and material consumption
	Distribute in Height	Uniformly distribute the parts over the height of the container
	Optimize Slice Volume	The volume of the slices is made as identical as possible amongst the maximal used height of the platform.
	Optimize Height and Slice Volume	The volume of the slices is made as identical as possible and the height of the platform is brought back to its minimum. This calculation method will take some more time than the other options.
Accuracy	Define the accuracy of the Automatic nesting algorithm. A smaller value will increase the accuracy of the nesting process but will take a longer time to achieve the optimal results.	
Stop After (minutes)	Stop criteria for the Automatic Nesting optimization. Define the Maximum duration of the optimization process.	

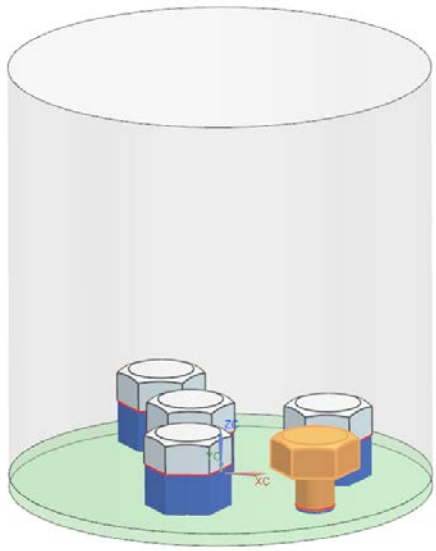
Create Automatic supports

Use the **Create Automatic Supports** command to automatically create support structures for the selected parts.

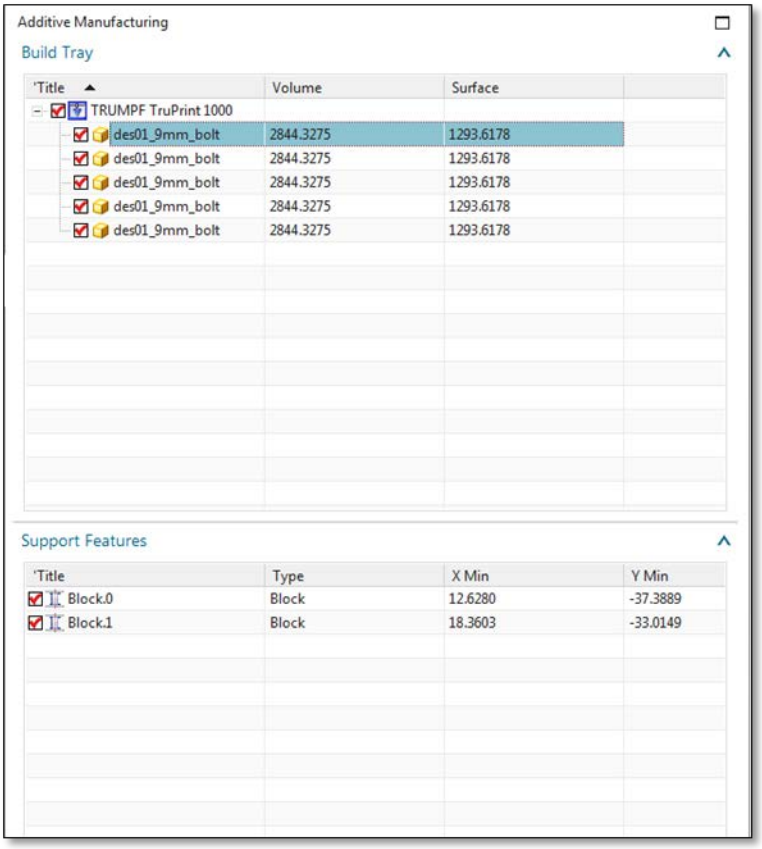
Different additive manufacturing processes, such as metal Powder Bed Fusion, require support structures for overhanging part areas during the powder solidification. During the support generation process we simultaneously strive to reduce support areas in order to decrease production cost, while enhancing the part quality and stability by increasing the support stiffness.

The **Fixed Plane Additive Manufacturing** application offers automatic support generation based on the user defined parameters. During the part export to the 3MF format, the application includes and slices the support as well.

The application generates supports of type block, line and point.



You can start the support generation by selecting one or more parts in the Additive Manufacturing viewer. In the right-click menu, select the **Create Automatic Supports** command.



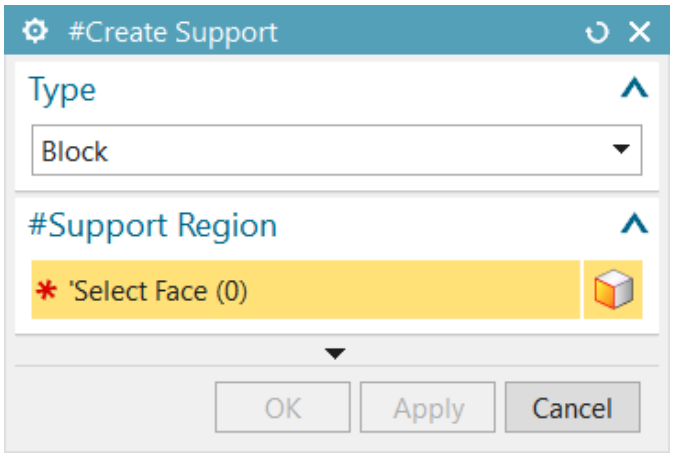
Once the support generation process is over, you can select the part and view its' support structure in the support features panel.

To learn more about the support structure generation see also Support **Feature Properties**.

Where do I find it?

Application	**Additive Manufacturing**
Prerequisite	**One or more parts selected in the Additive Manufacturing Viewer**
Additive Manufacturing Navigator	**Right click menu**

Create Manual support



Use the **Create Manual Support command** to manually define the region on the part to which you would like to add support structure.

Using the **Create Manual Support command** you can:

1. Create block support structure to a predefined face. For further explanation, please refer to **Define face for manual creation of support region workflow** section.
2. Create line support structure to a predefined curve. For further explanation, please refer to **Create line support structure** section.
3. Sketch a curve and add to it line support structure. For further explanation, please refer to **Create line support structure** section.
4. Create a point support structure to a selected point.

The support structure properties will be defined by the default support structure parameters settings. In order to set the default support structure parameters please refer to the **Support Structure Profile library.** After the generation of the support strucutre the user can view them in the support feature panel and change their paramters using the **Support Feature Properties** command.

Where do I find it?

Application	Additive Manufacturing
Prerequisite	Part selected in the Additive Manufacturing Viewer
Additive Manufacturing Navigator	Right click menu

Create Manual support dialog box

Type		
Set the type of support structure that you are about to create.		
Type	**Block**	Create support structure of type block.
	Line	Create support structure of type line.
	Point	Create support structure of type point.
Support Region		
Define the region on the part to which you would like to add support		
Select Face	Appears when Support Type is set to Block. Select one or more face to which you would like to add block support structure.	
Select Curve	Appears when Support Type is set to Line. Select one or more curve to which you would like to add line support structure.	
Draw Curve	Appears when Support Type is set to Line. Draw curve to which you would like to add line support structure. After you draw the curve you need to select it in the select curve field.	
Specify Point	Appears when Support Type is set to Point. Select a point to which you would like to add point support.	

Define a face for manual creation of support region workflow

To add block support structure to a certain area in the part, you must define a face that its boundary define the region to which you would like to add support.

For this example you will do the following:

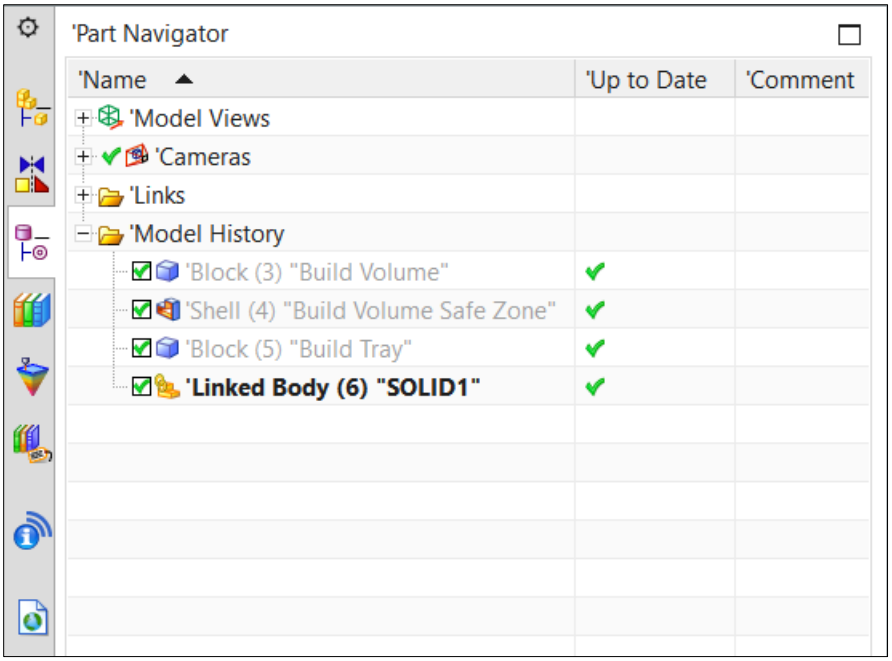
- Create Wave Geometry Linker
- Draw a curve on the surface of the part
- Create a face from the curve
- Create Block support type to the new face

Create Wave Geometry Linker

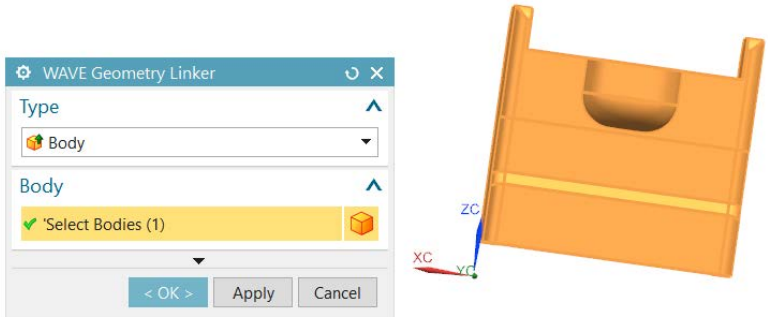
In order to allow modification of the part in the Additive Manufacturing part file. You must first create a Wave Geometry Linker to the part.

The **Create Automatic Support** and the **Create Manual support** commands automatically create Link body to the part. You will need to manually create the linked body if you want to create a face on the part before you execute those commands.

If a wave geometry link was already created to the part, you can skip this step. Please refer to the part navigator to check if a linked body already exist to the part.

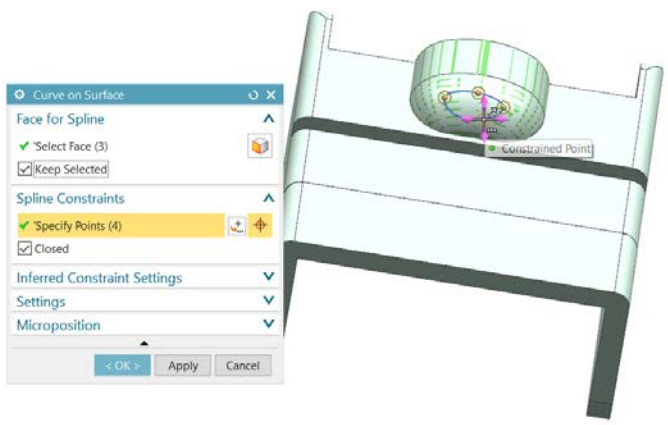


1. In the Assemblies tab, select the **Wave Geometry Linker** command.
2. Select the body to which you would like to create a face and click **OK**.



Draw a curve on the surface of the part

1. In the application Tab, select the **Modeling** application.
2. In the Curve Tab, select the **Curve on Surface** command.
3. Using the curve on surface command, draw the boundary of the new face. For additional explanation please refer to the **Curve on Surface** command help.

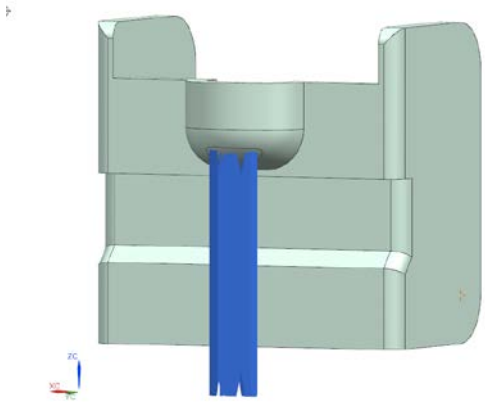


Create a face from the curve

1. In the **Modeling** application, execute the **Divide Face…** command.
2. Divide the face using the curve as the dividing object. For additional explanation please refer to the **Divide Face** command help.

Create block support type to the new face

1. In the Application tab, select the **Additive Manufacturing** application.
2. In the Additive Manufacturing viewer, select the part and execute the **Create Manual Support** command.
3. Select block support type and the newly created face and click ok.

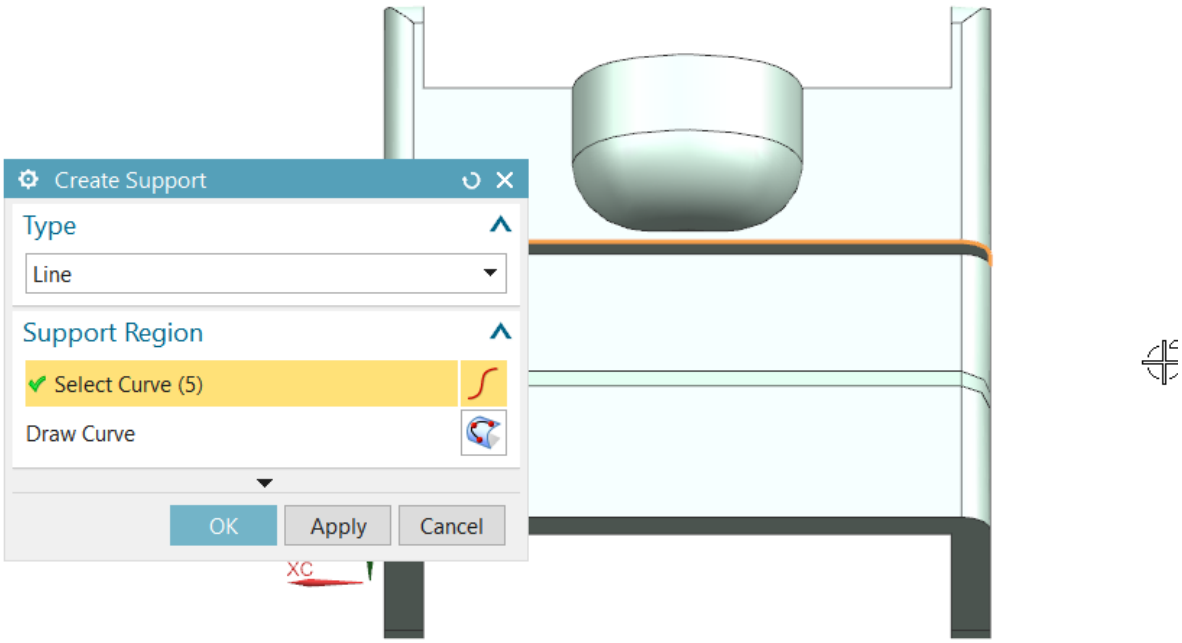


Create line support structure

In order to add line support structure to a curve in the part you can either select an existing curve on the part or sketch the curve to which you would like to add support.

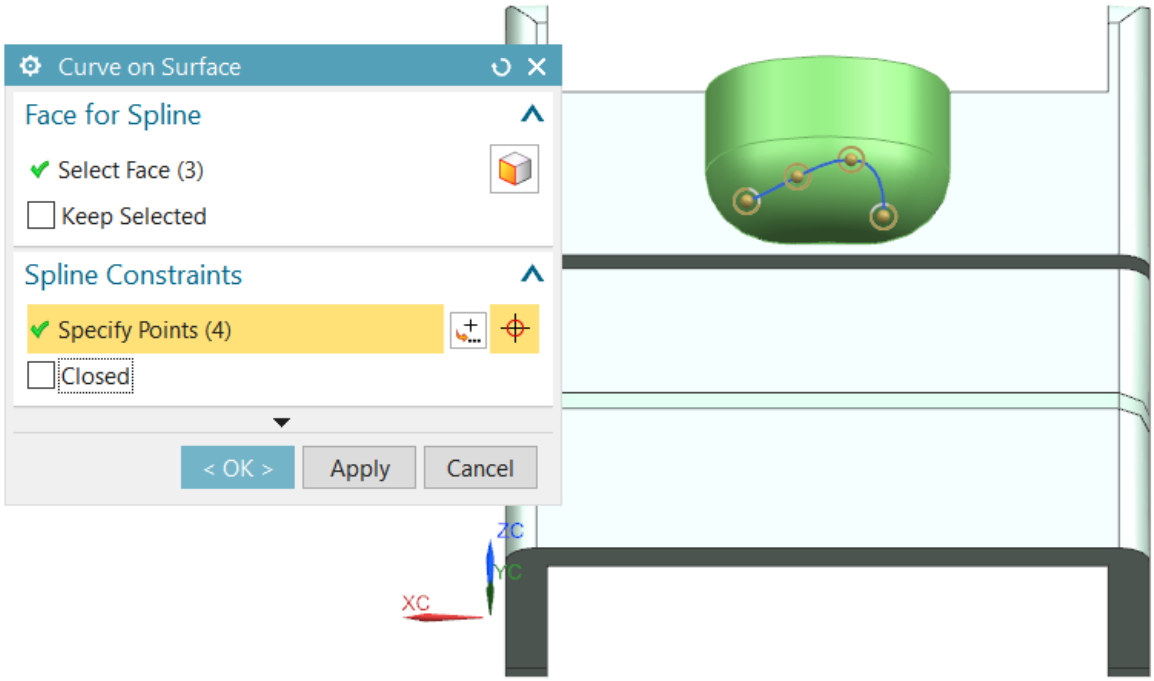
Add a line support structure to existing curve

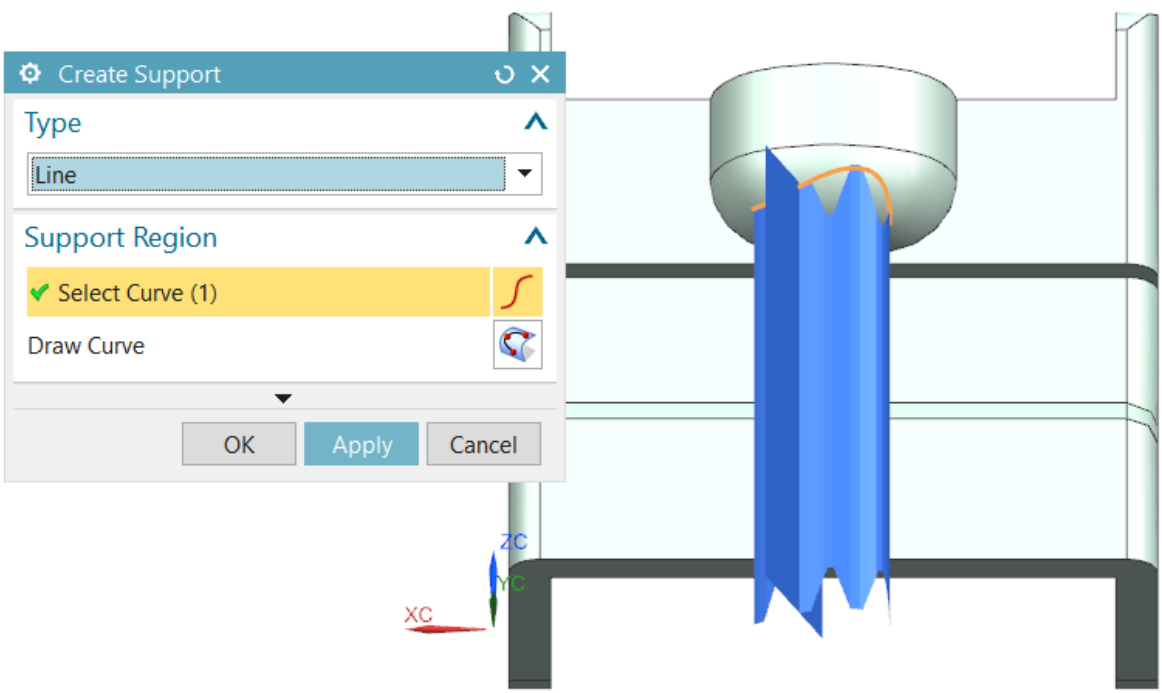
1. Open the **Create Manual Support** command on the selected part.
2. Select the **Line** support type.
3. In the Support region panel, select a curve.
4. Click **OK**.



Sketch a curve

1. Open the **Create Manual Support** command on the selected part.
2. Select the **Line** support type.
3. In the Support region panel, click the **Draw Curve** command.
4. Using the **Curve on Surface** command, draw the required curve. For additional explanation please refer to the **Curve on Surface** command help.
5. Select the newly created curve in the select curve field.
6. Click **OK**.


Curve on Surface
Face for Spline
Select Face (3)
Keep Selected
Spline Constraints
Specify Points (4)
Closed
< OK >
Apply
Cancel
ZC
YC
XC


Create Support
Type
Line
Support Region
Select Curve (1)
Draw Curve
OK
Apply
Cancel
ZC
YC
XC

Edit Support Structure

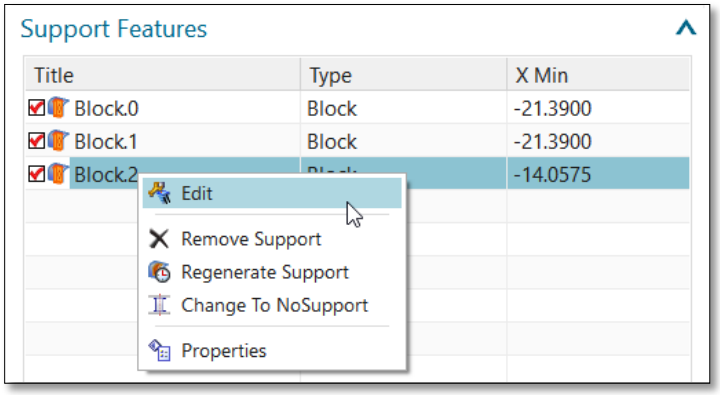
Use the **Edit Support** command to graphically manipulate the support angle or to change the support feature type to No Support.

Edit angled support properties

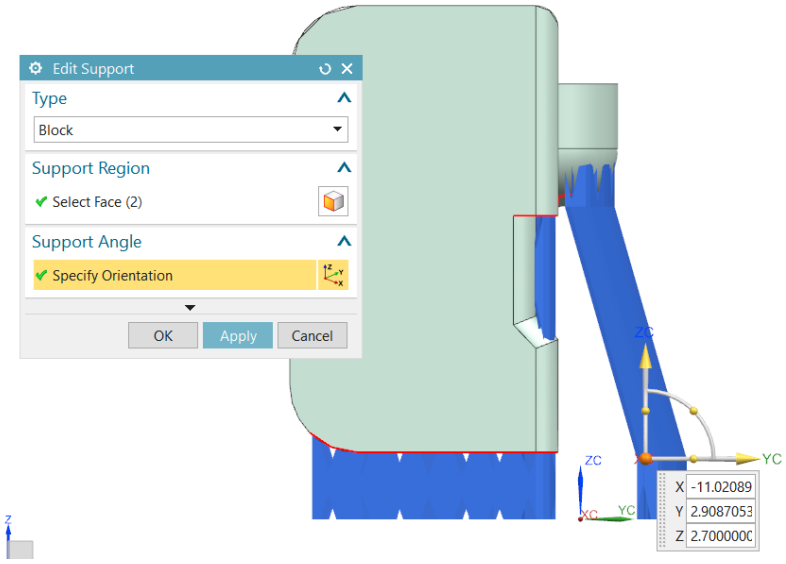
Angled supports allows you to avoid contacting unnecessary surfaces on the part and reduce the amount of post processing work by angling out the support structure from the vertical plane.

To change the angle of the support:

1. Select the relevant support structure in the Additive Manufacturing viewer.
2. Select the **Edit** command in the support structure right click menu.



3. Drag the placement manipulator along the X and Y axes to change the Angled support Shift X, Shift Y properties. Please refer to the Support Feature properties for further information.
4. Drag the placement manipulator along the Z axes to change the Angled support Bottom Straight Length property. Please refer to the Support Feature properties for further information.

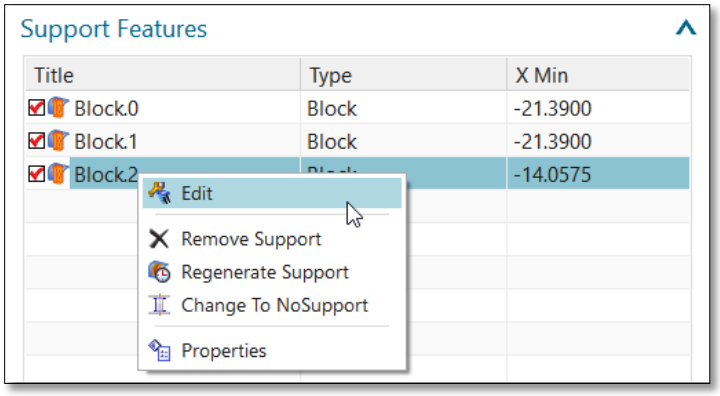


No Support type

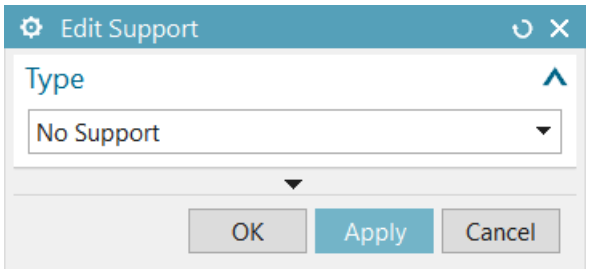
Use the **Edit** command to remove the support structure from a certain region without deleting the region.

To set the support feature type to no support:

1. Select the relevant support structure in the Additive Manufacturing viewer.
2. Select the **Edit** command in the support structure right click menu.



3. Change the type to **No Support.**

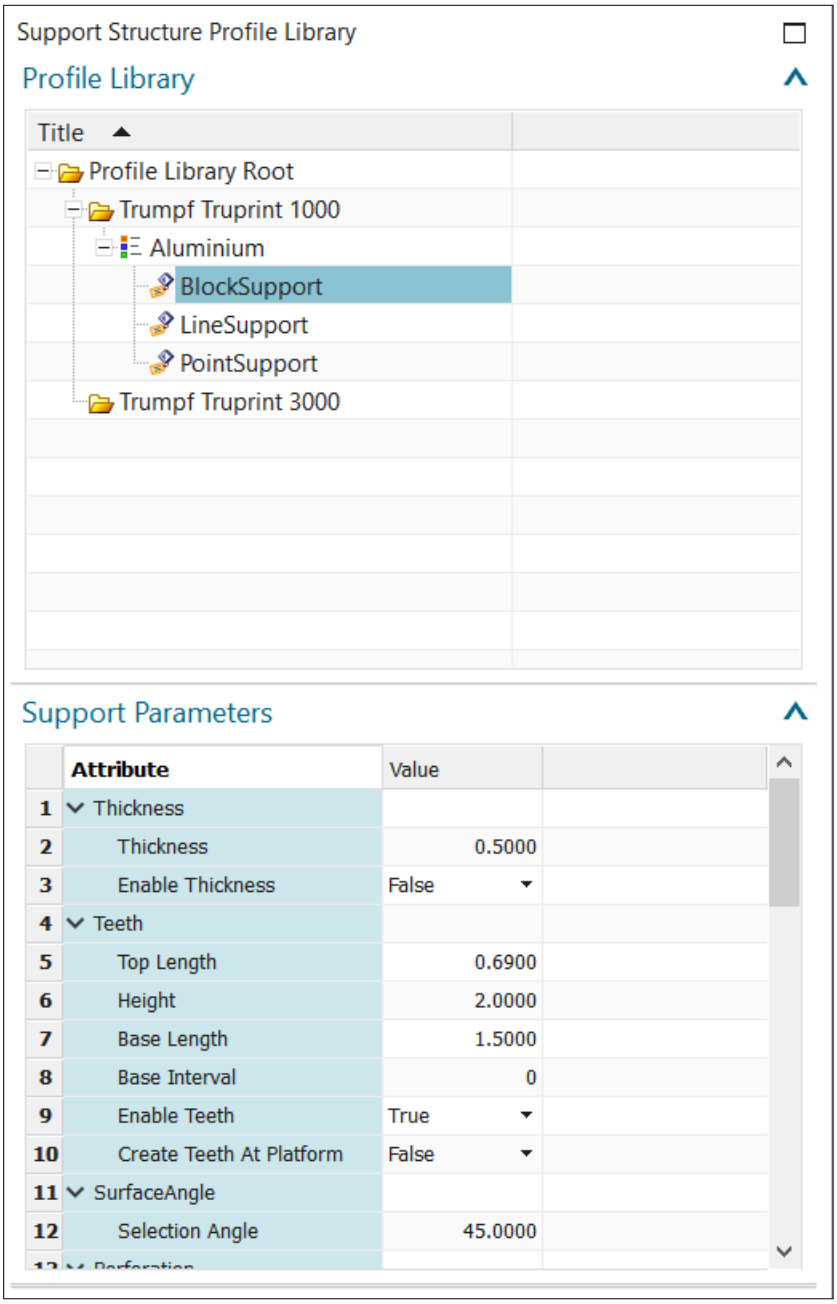


4. Click **OK**.

Where do I find it?

Application	**Additive Manufacturing**
Prerequisite	**One support features selected**
Additive Manufacturing Navigator	**Right click menu on a support feature**

Support Structure Profile Library



Use the **Support Structure Profile library viewer** to define your own default set of support structure parameters per machine, material, part type, etc.

The support structure profiles can be stored and reused from one build to another.

Once you specify a support structure profile as the default profile for the build, the default parameters will be used when a user creates the support structures either automatically or manually.

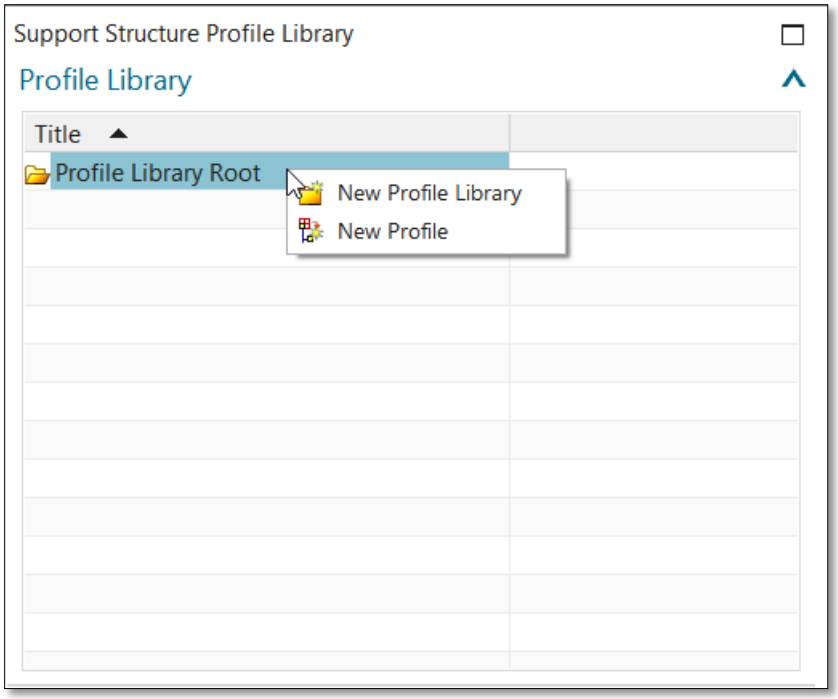
You can do the following with this command:

1. Create a profile library
2. Create a profile
3. Rename a profile library
4. Rename a profile
5. Delete a profile library
6. Delete a profile
7. Set the default support parameters values per support structure type
8. Set a default profile per a build.

By default the Profile library is read only and cannot be updated. In order to create a profile in the profile library, remove the ready only flag from the am.xml file under the $UGII_BASE_DIR\ mach\auxiliary\mfgam\ directory.

Create a profile library

In order to create a profile library, click on the **New Profile Library** command in the Profile library root or Profile library right-click menu.

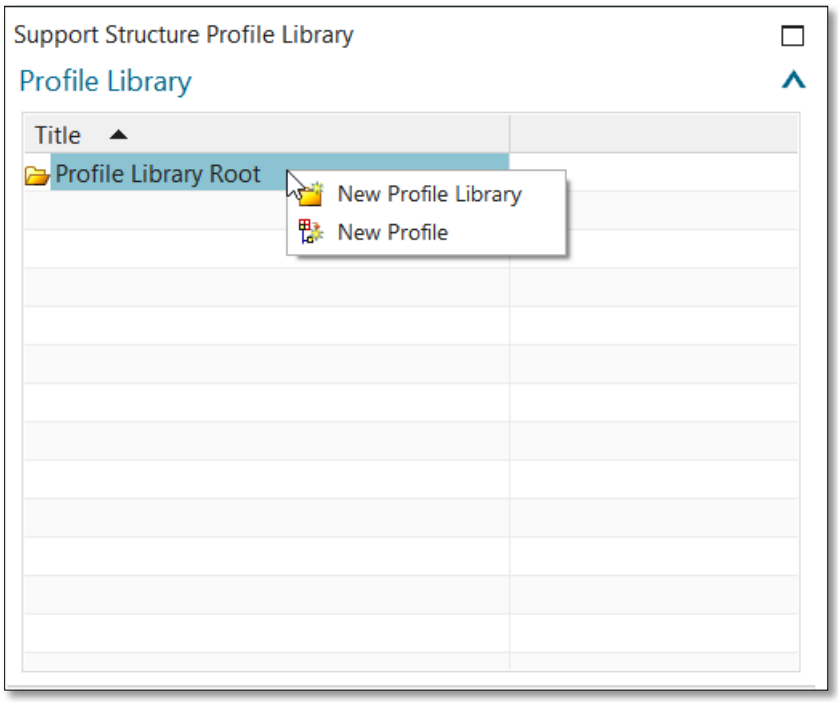


Rename a profile library

In order to rename a profile library, select the profile library and press F2.

Create a profile

In order to create a profile, click on the **New Profile** command in the Profile library root or Profile library right-click menu.

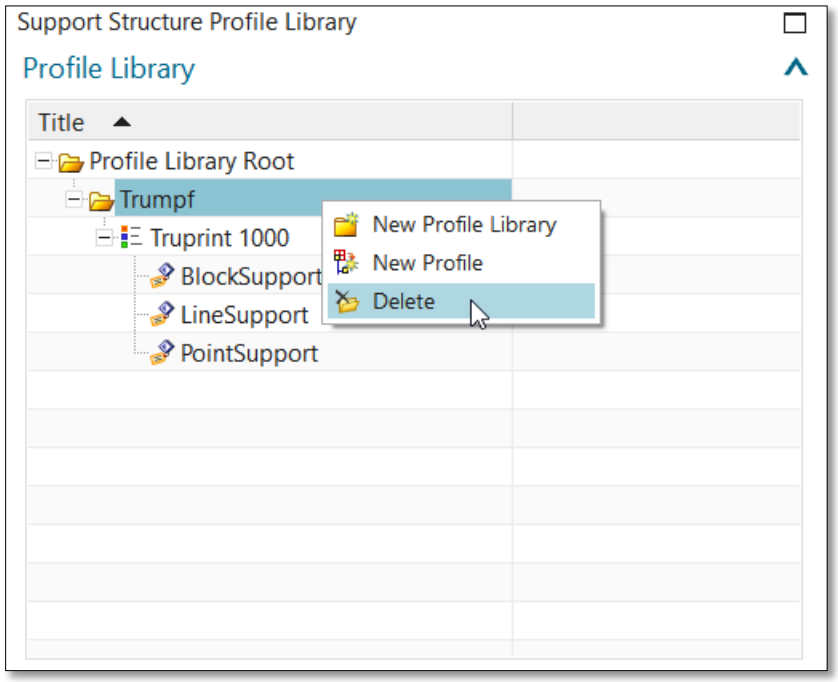


Rename a profile

In order to rename a profile, select the profile and press F2.

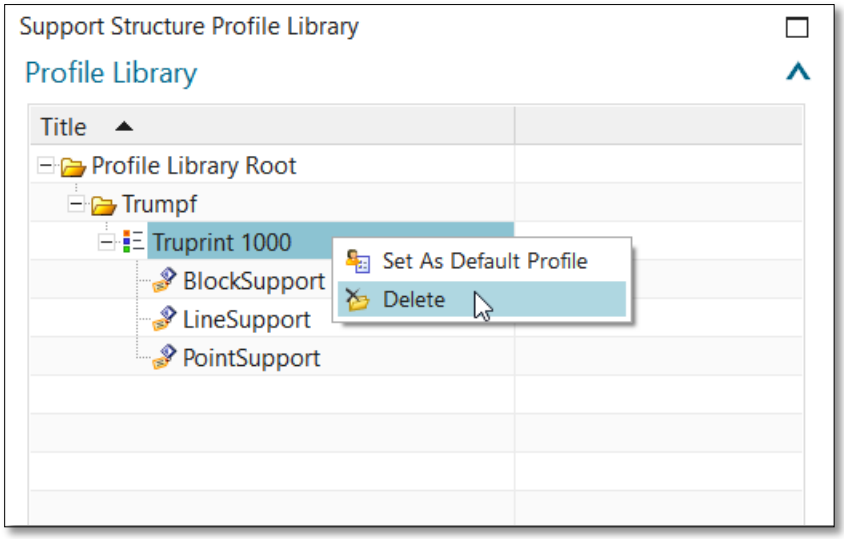
Delete a profile library

In order to delete a profile library, click on the **Delete** command in the Profile library right-click menu.



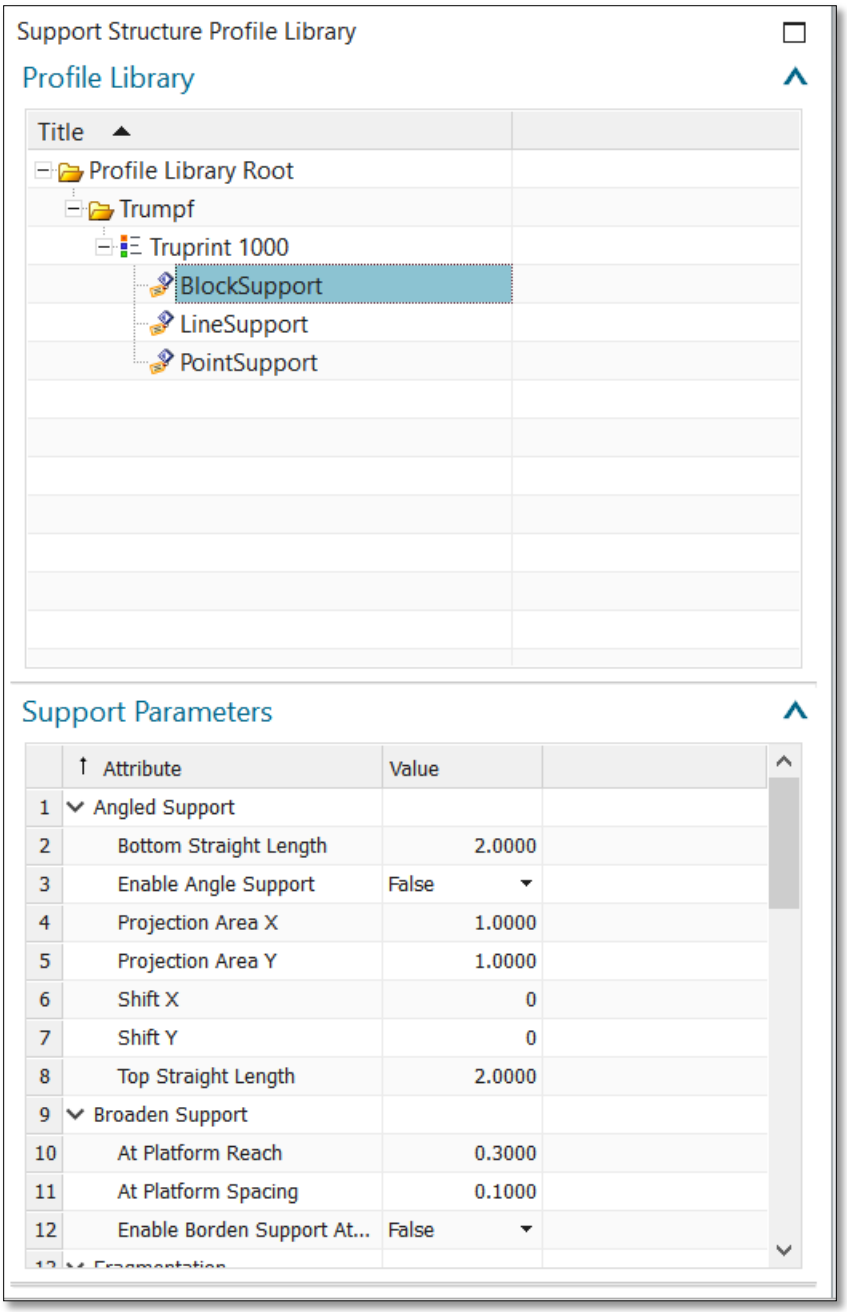
Delete a profile

In order to delete a profile, click on the **Delete** command in the Profile right-click menu.



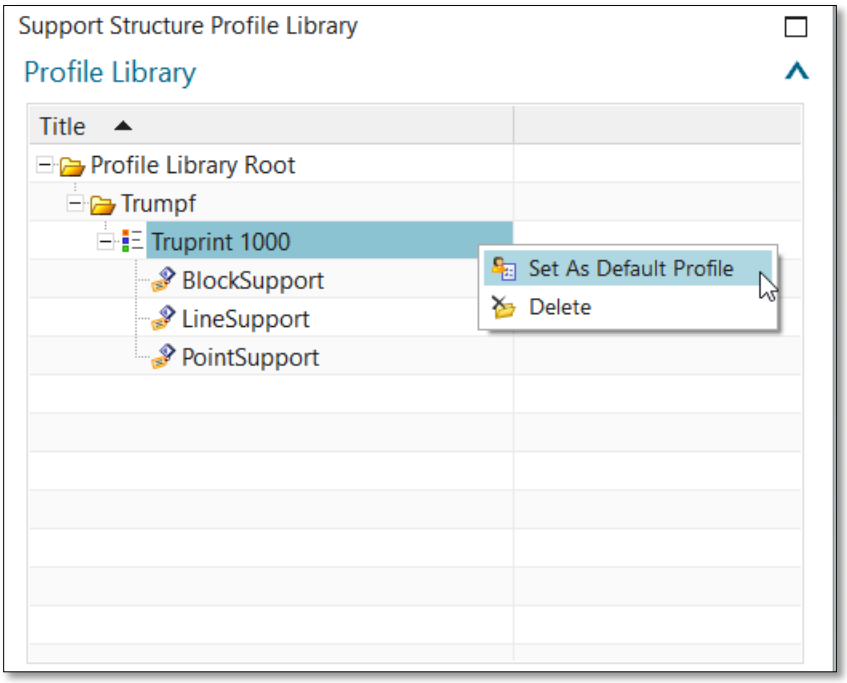
Set the default support parameters values per support structure type

In order to set the default support structure parameters for a profile, select the required support type and set the default parameters value in the support parameters panel. For further information about the support structure parameters please refer to **Support Feature Properties.**



Set a default profile per a build

In order to define the default profile for the build that will be used when create support structure, use the **Set as default profile** command on the Profile right-click menu.

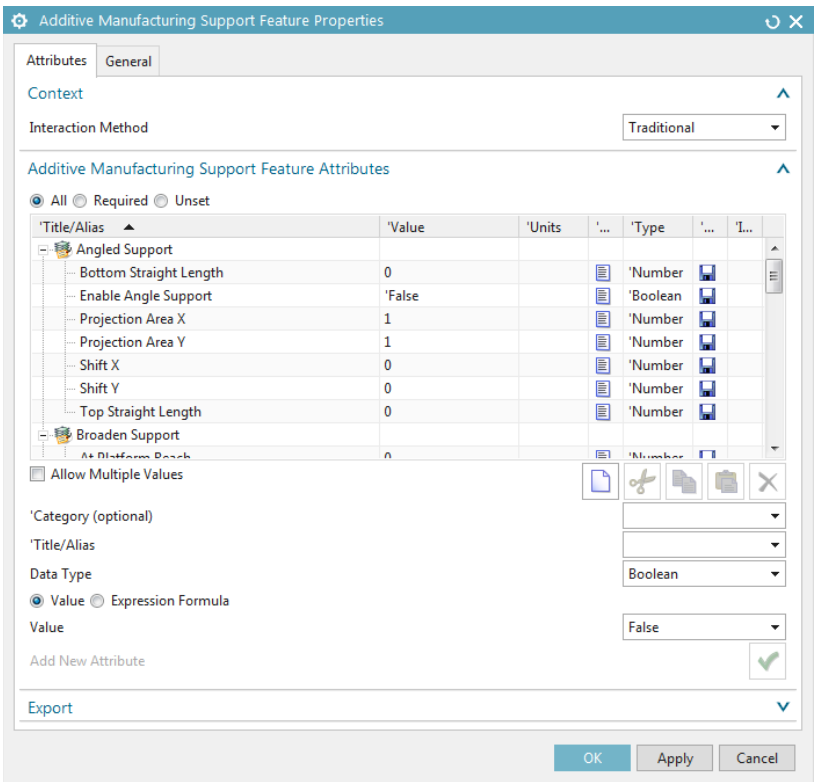


Where do I find it?

Application	**Additive Manufacturing**
Prerequisite	**Additive Manufacturing part file**
Support structure profile library Navigator	**Support structure profile library Navigator**

Support Feature Properties

User the **Support Feature Properties** command to edit the support feature properties.



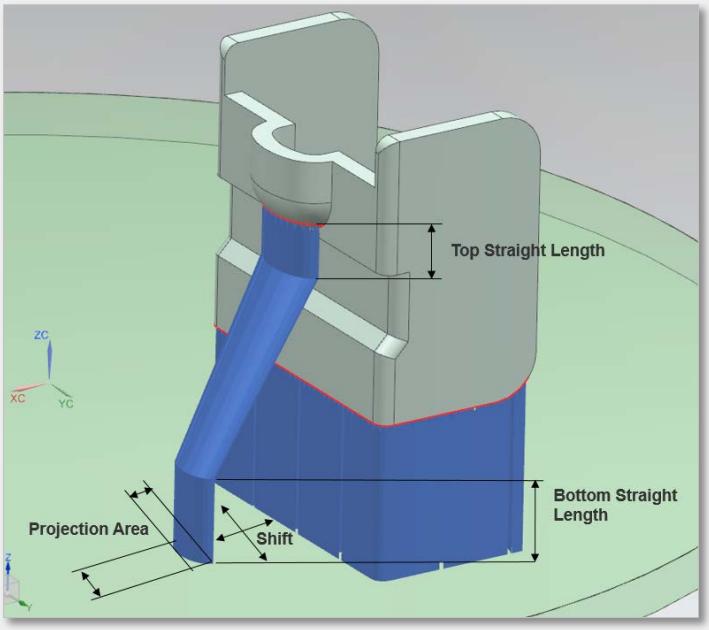
By right click on a support feature and execute the **Properties** command, the user can modify the supports by editing its properties and execute the **Regenerate Support** command.

In NX11.0.1 we introduced Support Structures of type block, line and point.

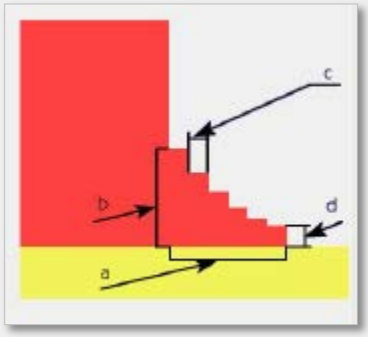
Common properties

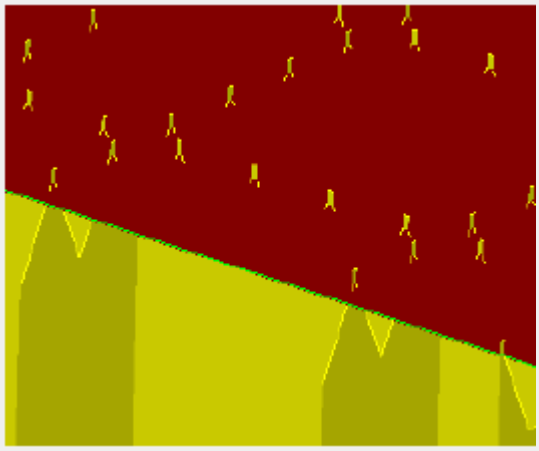
Angled Support

Angled supports allows you to avoid contacting unnecessary surfaces on the part and reduce the amount of post processing work by angling out the support structure from the vertical plane.



Bottom Straight Length	Add a straight support section at the bottom of the support structure in a given length.
Enable Angle Support	By enabling the angle support, you can create a support structure with a certain angle from the vertical plane.
Projection Area X	Extend or shrink the bottom of the support structure by the user defined factor figure multiply the top support length along the X axes. Negative values are possible if you want to make the bottom connection a smaller surface than the top connection.
Projection Area Y	Extend or shrink the bottom of the support structure by the user defined factor figure multiply the top support length along the Y axes. Negative values are possible if you want to make the bottom connection a smaller surface than the top connection.
Shift X	Shift the bottom of the support structure from the vertical plan along the X axes in a given distance.
Shift Y	Shift the bottom of the support structure from the vertical plan along the Y axes in a given distance.
Top Straight Length	Add a straight support section between the part and the support structure in a given length.

Broaden Support	
Extend the area of the support at the bottom of the support structure. 	
At Platform Reach (a)	Width of the additional support area.
At Platform Spacing (c)	Distance between each additional support segment.
Enable Broaden support At Platform	By enabling the broaden support, the base of the support structure will be extended in a stepwise manner.
Lowest Line	
When placing a support under a curved surface, it can happen that the lowest line (= the imaginary line connecting the lowest points of the surface) is not supported correctly. This can cause problems when building the part with some RP-techniques. When lowest line is checked, an extra line of support will be placed so that this lowest line is correctly supported.	
Enable Draw lowest line	By enabling the “Draw lowest line”, an extra line of support will be added to the external surface of the part.
Offset	
Lower Z offset	All the supports can have a certain offset into the part in order to ensure a better contact between part and support. Positive values ensure the support penetrates the part. Distance between the support and the lower surface. Required when the support rest on the part itself and not on the build tray.
No Support offset	Defines a distance from a vertical wall, where no supports are required. 

Upper Z offset	All the supports can have a certain offset into the part in order to ensure a better contact between part and support. Positive values ensure the support penetrates the part. Distance between the support and the upper surface. 

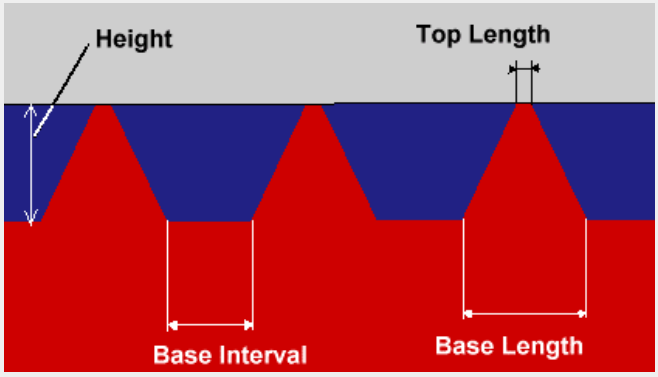
Surface angle

The Surface Angle defines which surfaces of the printed part require supports. The Surface Angle is the angle between the horizontal XY plane and the surface. Any surface that its angle to the horizontal XY plane is bigger than the selection angle is considered as self-supporting and does not require support structure.

Selection angle	The Surface Angle defines which surfaces of the printed part require supports. The Surface Angle is the angle between the horizontal XY plane and the surface. Any surface that its angle to the horizontal XY plane is bigger than the selection angle is considered as self-supporting and does not require support structure.

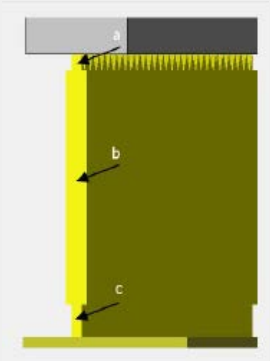
Teeth

Teeth parameter allows you to create a teeth profile at the top and the bottom of the support structure in to facilitate its removal.



Base Interval	Distance between two consecutives teeth.
Base Length	Length of the teeth base at the contact point with the support

Create Teeth at platform	By enabling create teeth at platform, teeth profile will be created at the bottom of the support in the conjunction with the platform.
Enable Teeth	By enabling teeth, teeth profile will be created at the top of the support in the conjunction the part. Lower teeth will be created only if the support is trimming on another part.
Height	Height of the teeth.
Top Length	Length of the teeth at the contact point with the part.
Thickness	

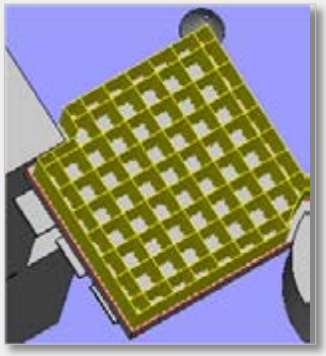


Enable Thickness	By enabling Thickness, the user can set the thickness to the non-solid support type.
Thickness	Set the thickness of the support walls and teeth.

Block support properties

Hatching

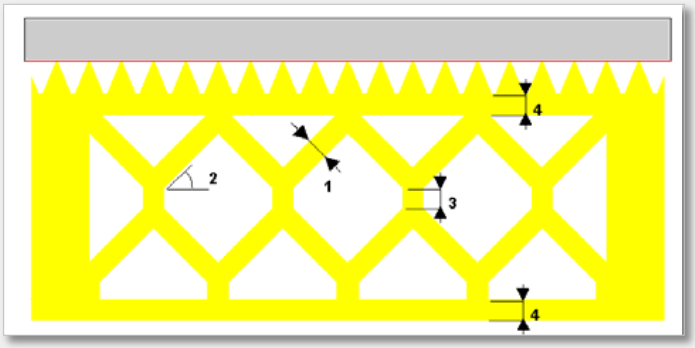
Block Supports are made with a grid of X and Y lines. The distance between the lines is defined by the hatching parameter.



X Hatching	Distance between the lines along the X axes in block support.
Y Hatching	Distance between the lines along the Y axes in block support.

Perforation

Define whether the support structure will be solid or perforated. Currently only Diamond shape perforation is supported. Perforation reduces the amount of material that is used for the support structure and enables easy removal of the block support.



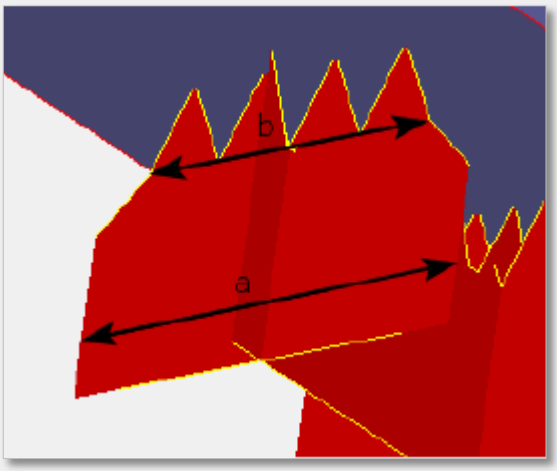
Angle (2)	Angle of the diamond-shaped perforation.
Bean thickness (1)	Thickness of the perforation wall.
Perforation	By enabling perforation the support structure will be perforated.
Solid height (4)	Height of the solid structure that connect the perforation support to the platform and to the upper surface.
Height (3)	The height of the vertical segment at the diamond-shaped perforation.

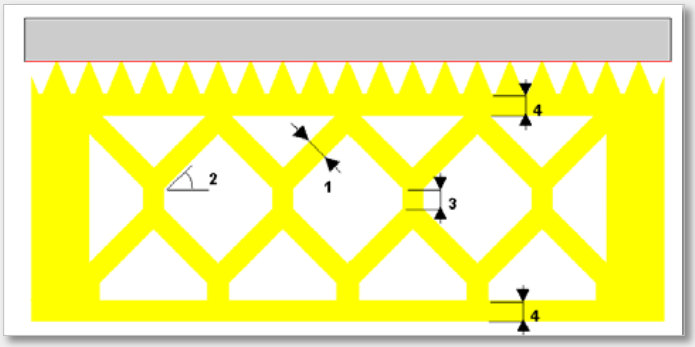
Fragmentation

Fragmentation separates the supports to several blocks. Fragmentation enables easy removal of the block support.

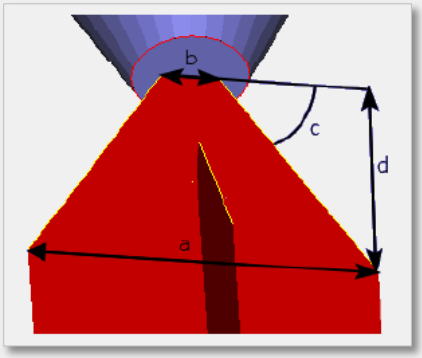
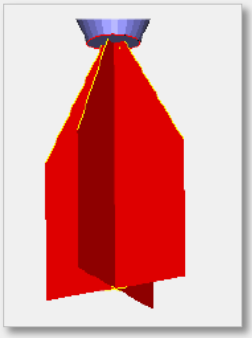
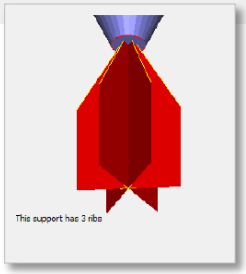
Fragmentate Borders Interval	Length of the border support between each gap.
Fragmentate borders	By enabling Fragmentate borders, gaps will be generated in the border of the support structure.
Interval	Length of the support (along X and Y axes) between each gap

Line support properties

Cross Line	
Cross line interval	Set the length of the crossing lines. 
Maximum Contact Length (b)	Maximum contact Length limits the length of the contact between the support and the surface.
Minimum Rib Length (a)	The Minimum Rib Length determines the length of the cross line. Larger ribs are more stable but may be harder to remove.

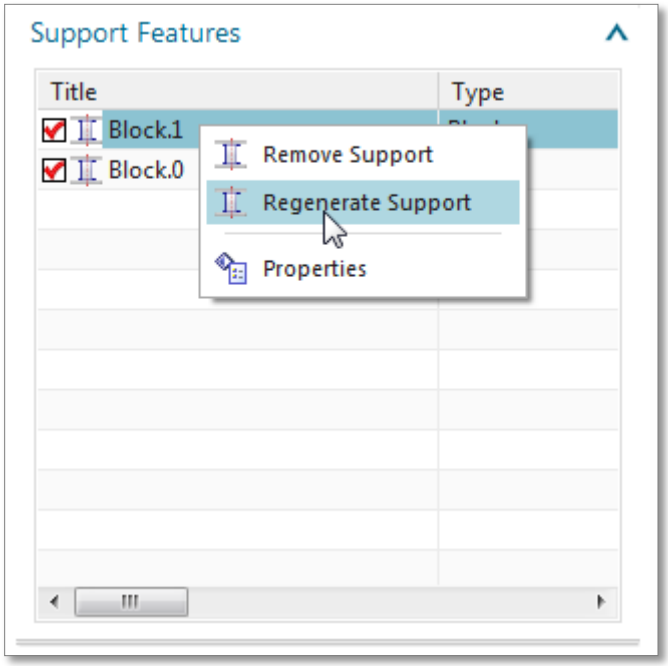
Perforation	
Define whether the support structure will be solid or perforated. Currently only Diamond shape perforation is supported. 	
Angle (2)	Angle of the diamond-shaped perforation
Bean thickness (1)	Thickness of the perforation wall
Perforation	By enabling perforation the support structure will be perforated
Solid height (4)	Height of the solid structure that connect the perforation support to the platform and to the upper surface.
Height (3)	The height of the vertical segment at the diamond-shaped perforation.

Point support properties

Contact Length	
	
Maximum Contact Length (b)	Define whether to support the complete surface or only a certain Contact Length.
Minimum Rib Length (a)	Set the length of the ribs. You need a minimal length in order to have enough stability and to prevent the support from falling through the platform grid.
Vertical Distance (d)	Set the distance from the support to the part.
Reinforcement	
Reinforce a support by adding an extra contour. 	
Enable Reinforcement	By enabling Reinforcement, set an extra contour to the support.
Rib	
Number of Ribs	Determine the number of the ribs. 

Regenerate Support

Use the **Regenerate Support** command to update the selected support feature after making changes to the part or to the support feature's properties.



In order to use the **Regenerate Support** command:

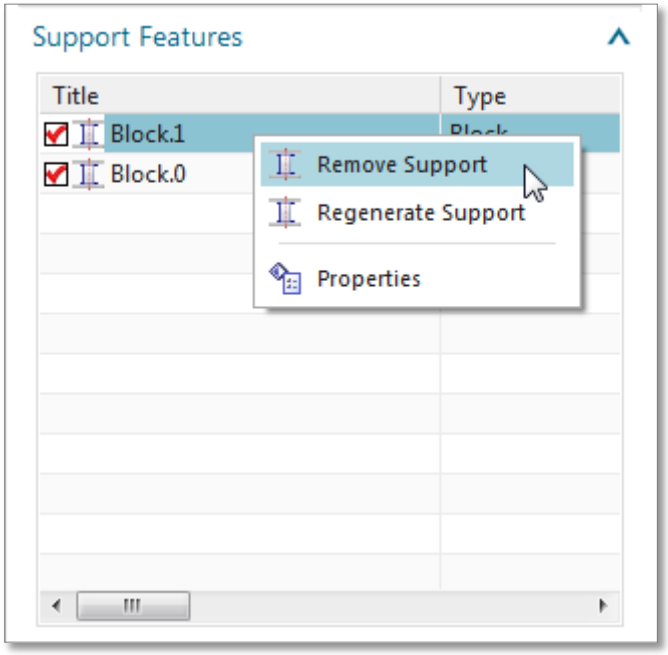
1. Select one or more support features at the Support Feature panel or in the graphic viewer. A new Type Filter support was added to simplify the selection of the support in the graphic viewer. For more information refer to the **Support Type Filter**.
2. Using the right-click menu, select the **Regenerate Support** command.

Where do I find it?

Application	Additive Manufacturing
Prerequisite	One or more support features selected
Additive Manufacturing Navigator	Right click menu on a support feature

Remove Support

Use **Remove Support** to delete one or more support features.



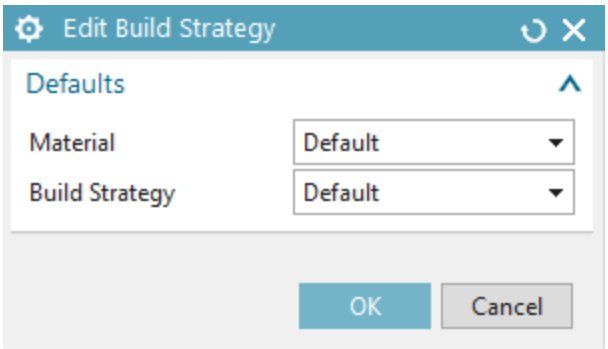
In order to use the **Remove Support** command:

1. Select one or more support features at the Support Feature panel or in the graphic viewer. To simplify the selection of the support in the graphic a new Type Filter support was added. For more information refer to the **Support Type Filter**.
2. Using the Right Click menu select the **Remove Support** command.

Where do I find it?

Application	Additive Manufacturing
Prerequisite	One or more selected Support Features
Additive Manufacturing Navigator	Right click on the selected support features

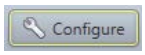
Edit Build Strategy

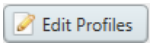


Use **Edit Build Strategy** to assign different build strategy to the entire build tray, or for a specific component.

Converting your digital CAD model into a physical part requires a translation to a layer-by-layer representation that can be interpreted and built by the machine. This translation, build strategy, directly affects the behavior of the machine (e.g. build speed, material consumption) and the properties of the physical parts (e.g. strength, weight).

In order to create, edit, or view build strategies:

1. Launch the Build Processor Manager application.
2. Select the target machine.
3. Execute the **Configure** command. 

4. In the Configure Printer dialog click on the **Edit Profiles** command. 

Where do I find it?

Application	Additive Manufacturing
Prerequisite	Select the build tray or one or more parts.
Additive Manufacturing Navigator	Right click menu on the build tray or on the selected parts

Generate

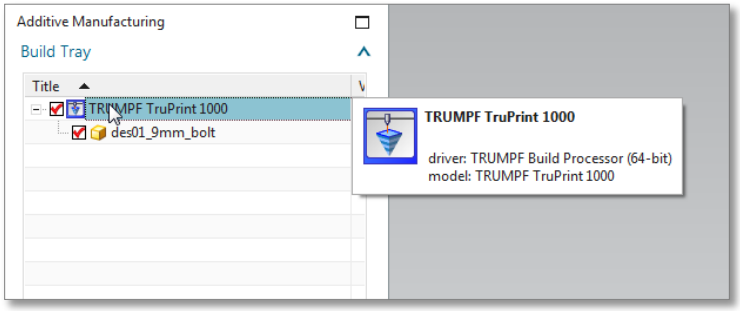
Use **Generate** to start the 3D printer build processor and to translate the assembly and its support structures to the machine file format.

The default output folder is the parts location.

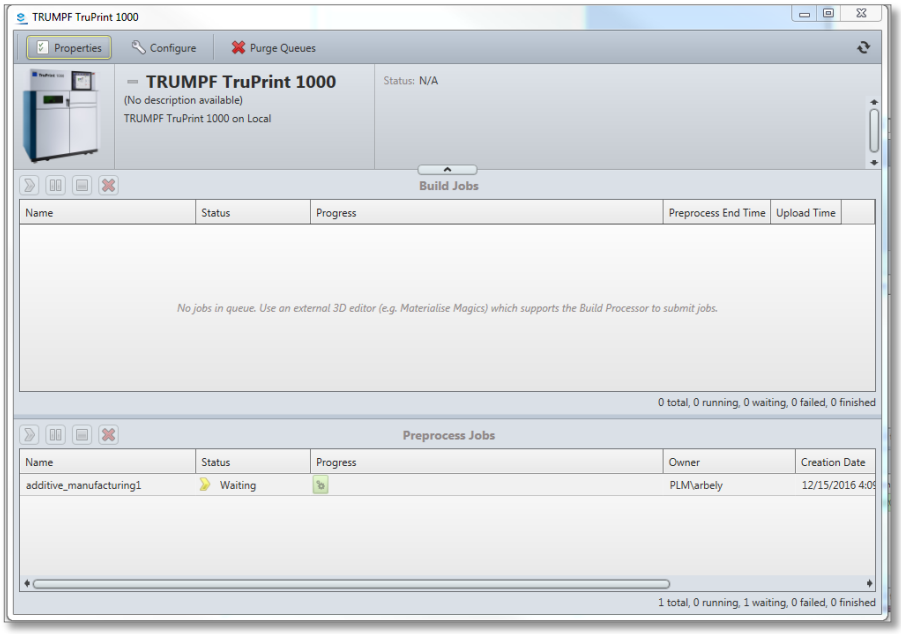
In order to generate the machine's output file, right-click on the build tray and select the **Generate** command.

The **Generate** process can be a time consuming operation. You can view its progress:

- By hovering over the build tray object in the Additive Manufacturing viewer, a tooltip with the machine information and the job status is displayed.



- In the **Build Processor Manager** application, at the machine's queue.

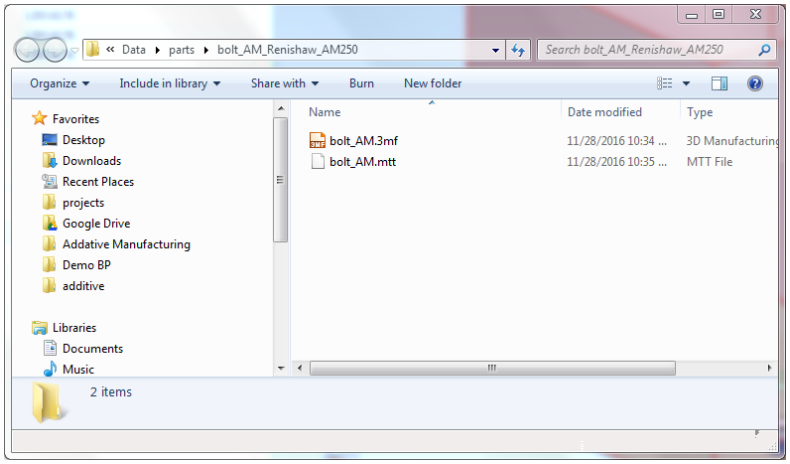


Where do I find it?

Application	**Additive Manufacturing**
Additive Manufacturing Navigator	**Right click menu on the build tray**

Explore Output Directory

Use the **Explore Output Directory** command to inspect the output files generated by the **Generate** command**.** The command opens the windows explorer at the last **Generate** command output folder.

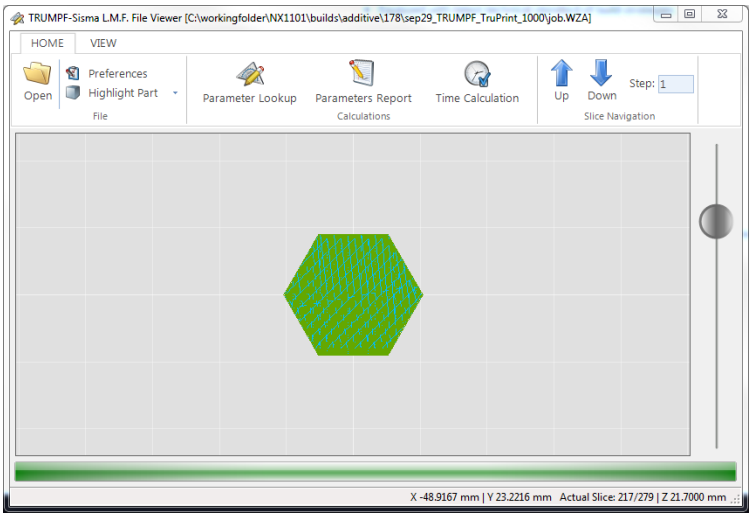


To run the **Explore Output Directory** command, right-click on the build tray object.

Where do I find it?

Application	**Additive Manufacturing**
Prerequisite	**The user generated the machines output files.**
Additive Manufacturing Navigator	**Right click menu on the build tray object.**

View Slices

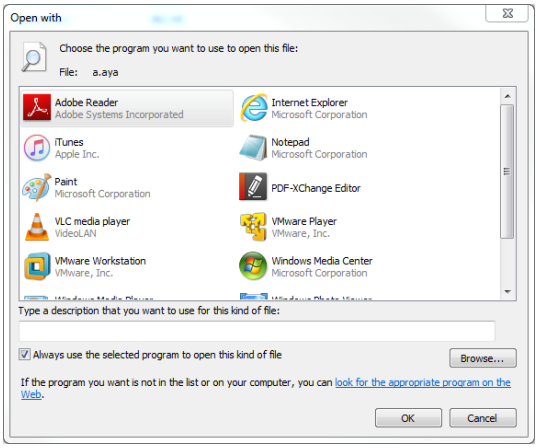


If supported, use the **View Slices** command to view slices and hatches with the slice viewer.

Different printers require dedicated slice viewer software. The slice viewer should be supplied by Materialise with the dedicated build processor, if such a viewer exists for the build processor.

In order to install the slice viewer please follow the following steps:

1. Generate the Machines' output file.
2. Run the **View Slices** command.
3. On first Load: In the **Open with** dialog box, click the **Browse** command, select the dedicated machine exe file that you received from Materialise, check **Always use the selected program to open this kind of file**, and click **OK**.



Where do I find it?

Application	**Additive Manufacturing**
Prerequisite	**The machine output file were generated**
Additive Manufacturing Navigator	**View Slices**

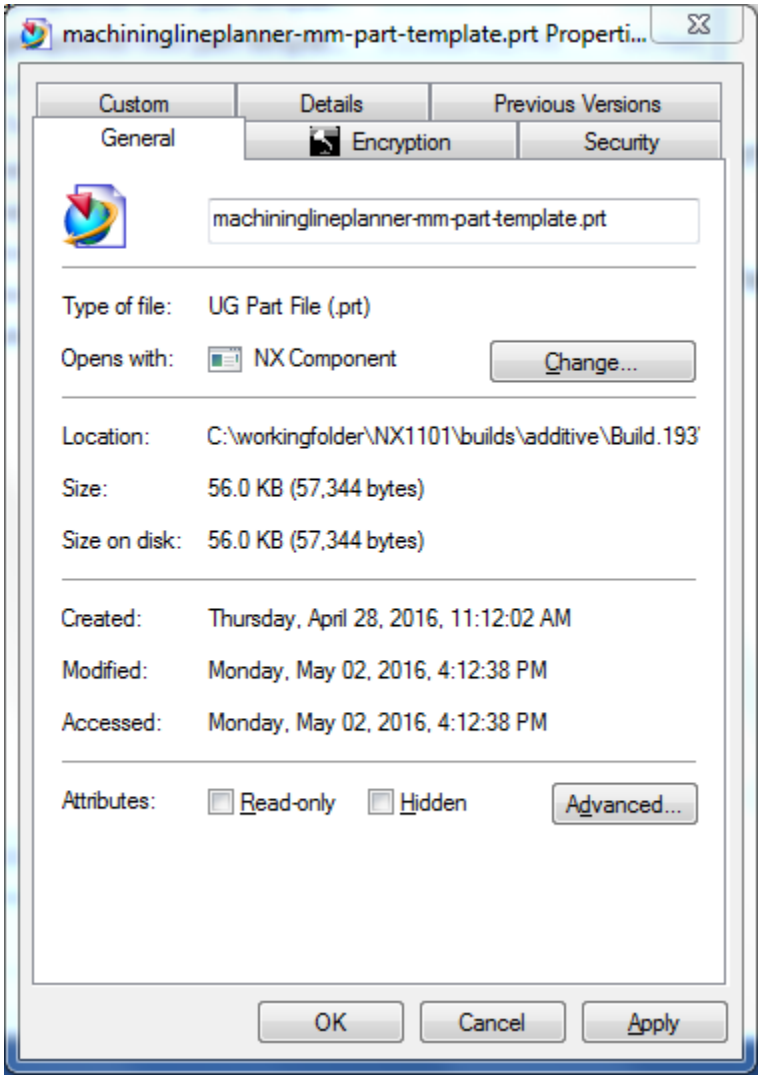
How to edit a template with specific 3D Printer information

When you create a new Additive Manufacturing part file using the out of the box vendor templates, you must first set the target printer in order to start pre-processing the build.

In order to avoid selecting the 3D printer each time you create a new part file, you can add the 3D printer information to the template. Adding the 3D printer information will still allow you to change the target printer.

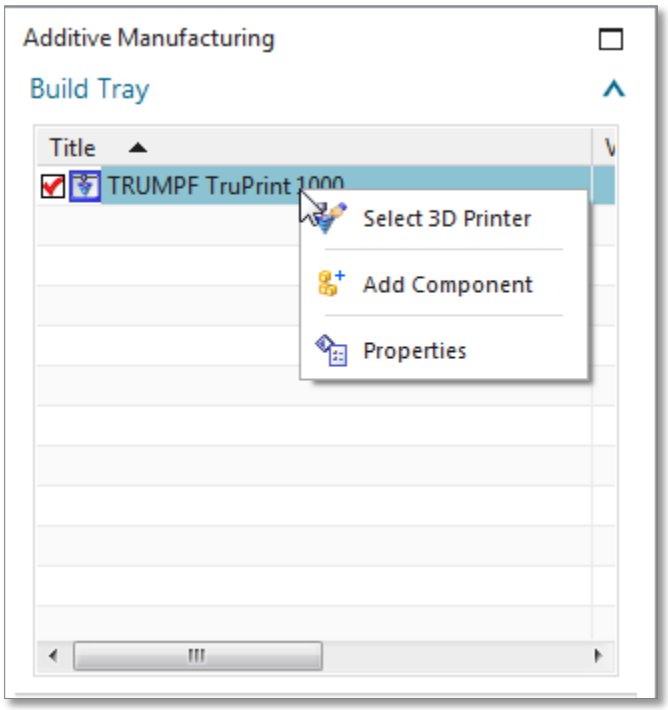
You can set the printer definition in the template following these steps:

1. In the $UGII_BASE_DIR\mach\templates folder remove the read-only flag from the template that you would like to modify.



2. Open the selected template in NX using **Open…** command.

3. Select the required printer using the **Select 3D Printer** command.



4. Save the template using the **Save** command.
5. Close all parts.

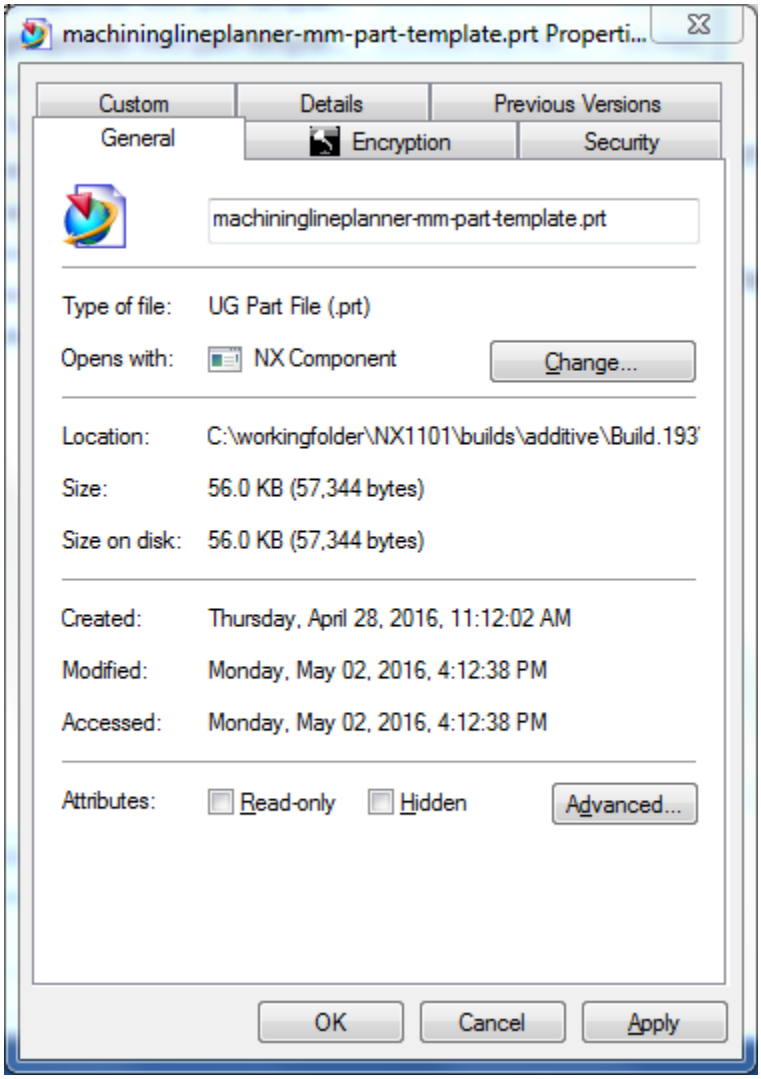
How to add no build zone to the template

To visualize the No Build Zones in addition to the build tray geometry, you can modify the machine's build tray geometry in the template.

Please note that No Build Zone geometry can be used for visualization and for collision detection. However, the **Automatic Nesting** command currently ignores it.

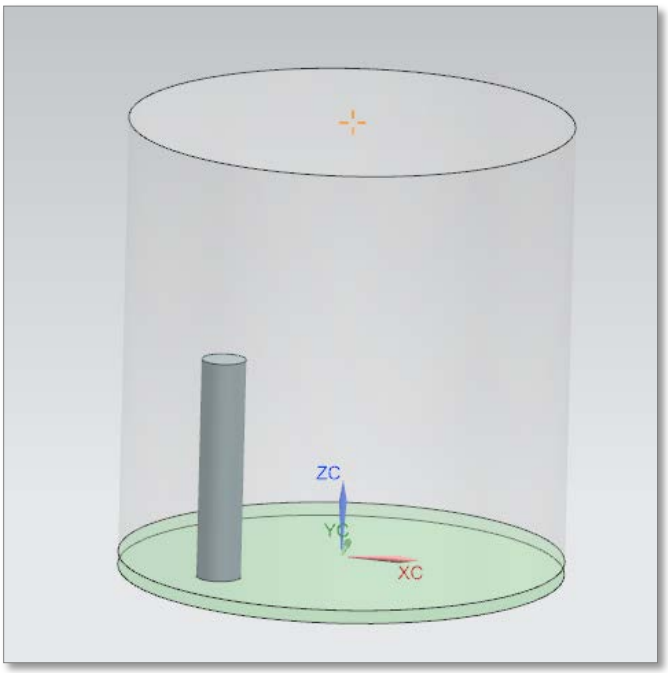
In order to add No Build Zones to the template please follow these steps:

1. In the $UGII_BASE_DIR\mach\templates folder, remove the read-only flag from the template that you would like to modify.



2. Update the template with the specific 3D printer definition. For more information please refer to **How to edit a template with specific 3D Printer information**.

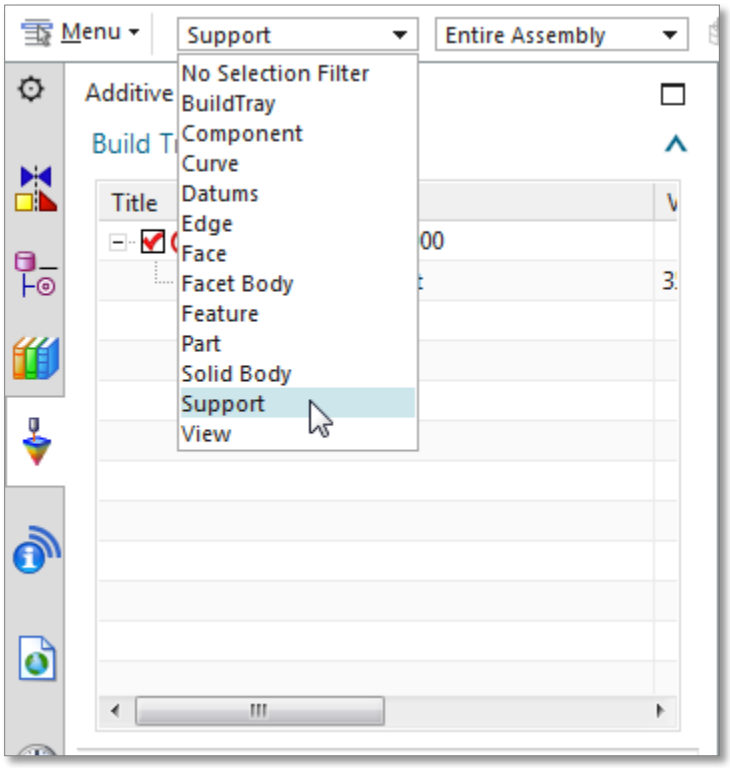
3. Open the selected template in NX using the **Open…** command.
4. Start the **Modeling** application.
5. Modify the build tray geometry.



6. Save the template using the **Save** command.
7. Close all parts.

Support Type Filter

To simplify the support feature selection, a new type **Support** was added to the **Type Filter** command.



Set the **Support** type filter to directly select the supports in the graphic.

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