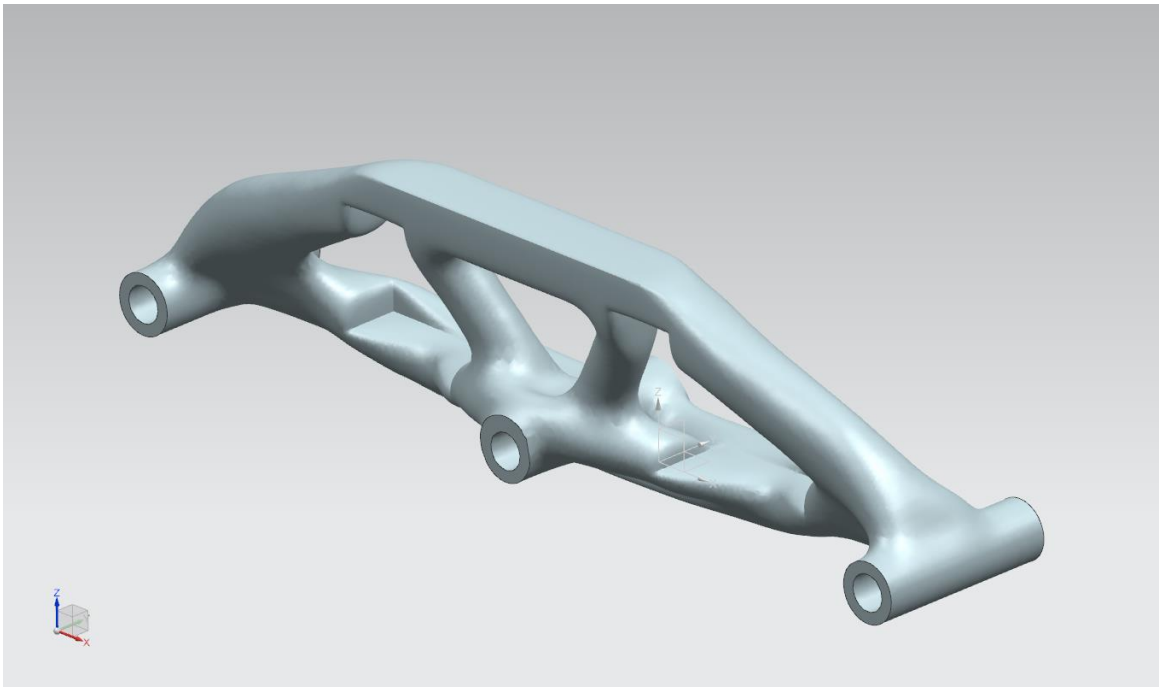
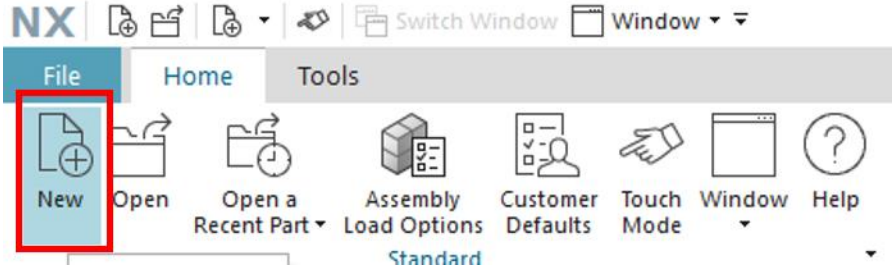
ADDITIVE MANUFACTURING

This tutorial shows how to open the AM template, select a printer, mount a part onto the AM build tray, subject the part to AM checkers, generate supports, modify support attributes and generate/prepare the part file for printing



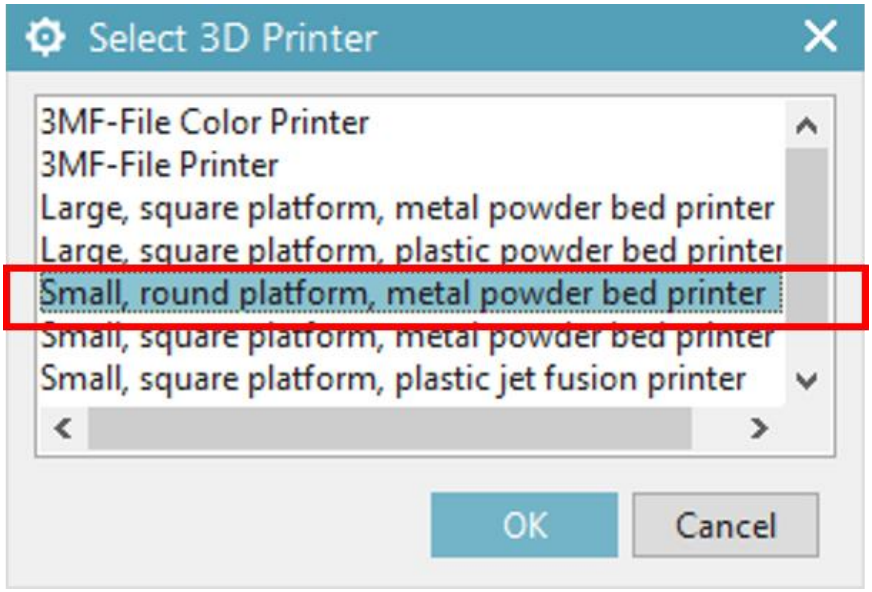
OPEN AM TEMPLATE

1. In NX, on the **Ribbon bar** in the **Home** tab, select **New**
2. In the **New** dialog box, select **Additive Manufacturing,** enter a name for the part file, (default additive_manufacturing1.prt) then click **OK**

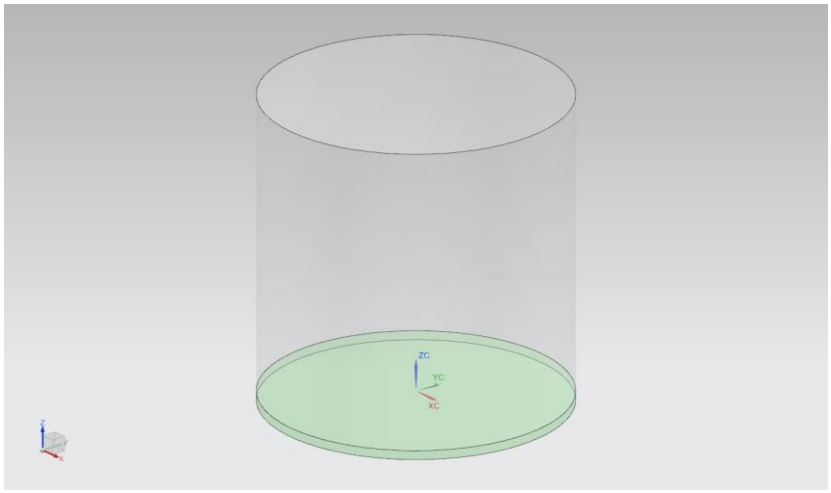


SELECT PRINTER

1. **Select 3D Printer** dialog box prompts to select a type of printer from the available list. Upon selection, click **OK**

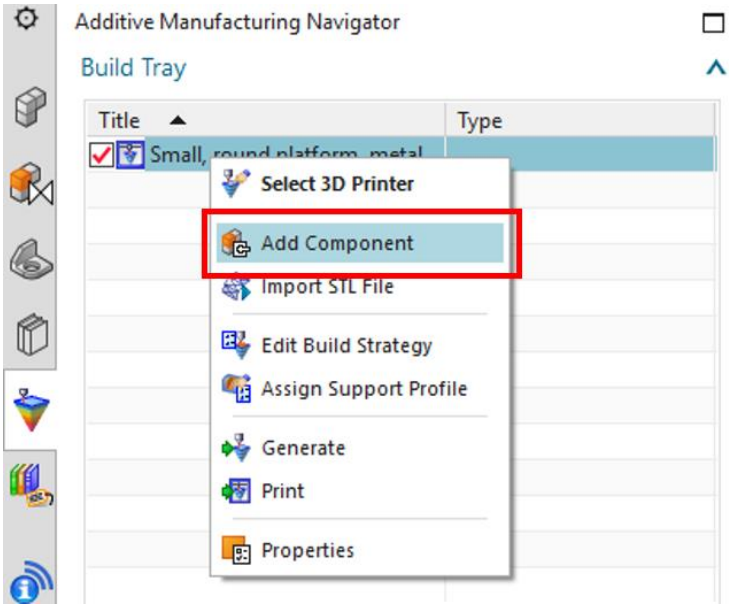


Notice the selected build tray (printer) appears in the graphics window



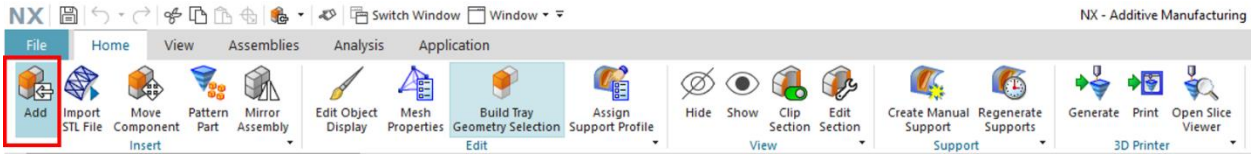
ADD COMPONENT

1. To add a component to the build tray, in the **Additive Manufacturing Navigator** menu, right click on the printer in the **Build Tray** group, from the drop-down menu, select **Add Component**



OR

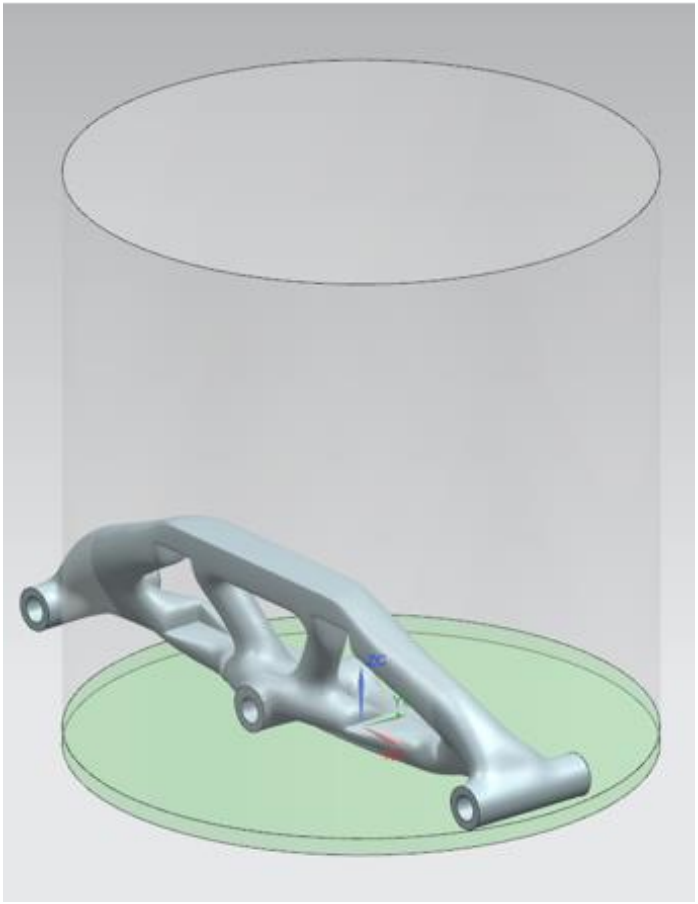
In the Ribbon bar, under the **Home** tab, click on **Add**



2. In the **Add Component** dialog box, click on **Open**  icon to browse for the part "**Simple_Cantilever_result**" in the stored folder
   Notice the selected part appears on the build tray in the graphics window
   NOTE: Multiple parts can be selected
3. Enter the desired number of copies of the selected part (or parts) under the **Count** list. Enter **Count = 1**
4. Specify desired **Location** of the part
   Component Anchor = Absolute
   Assembly Location = Absolute – Work Part
   Cycle Orientation = WCS 
5. Specify the **Placement** of the part
   Placement = Move

Click **OK**

Notice the component mounted on the build tray in the graphics window



DESIGN VALIDATION FOR ADDITIVE MANUFACTURING

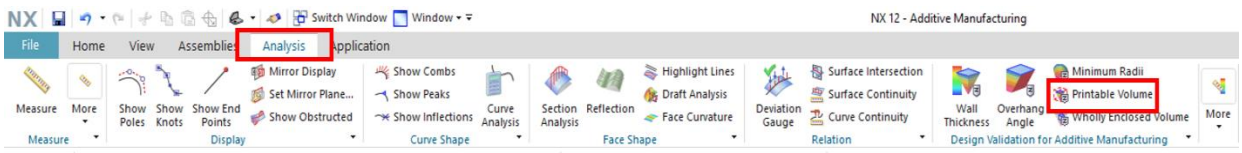
Part (or parts) mounted on the build tray must satisfy some AM checkers-

PRINTABLE VOLUME

The **Printable Volume** checker helps determine the size of build tray required for a part positioned in a specified orientation.

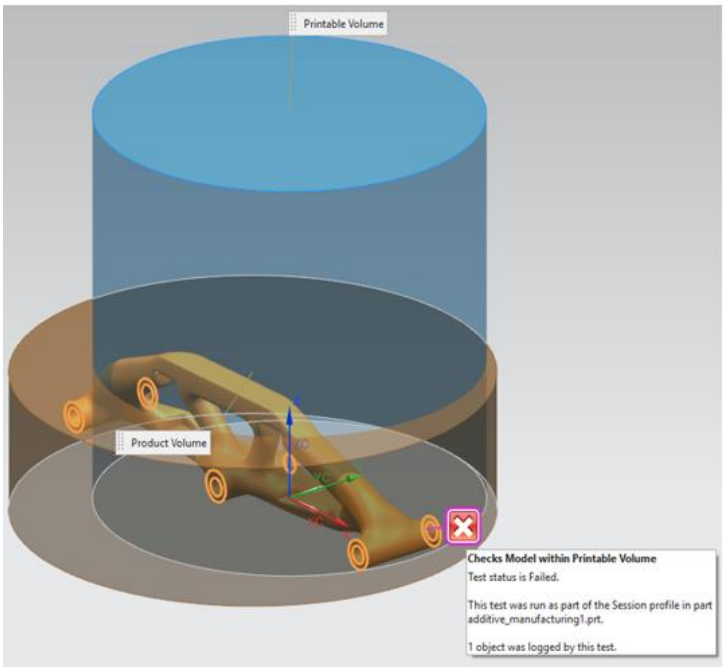
The test results in- passed,  failed  or passed with a warning 

1. On the Ribbon bar, in the **Analysis** tab, click on **Printable Volume**



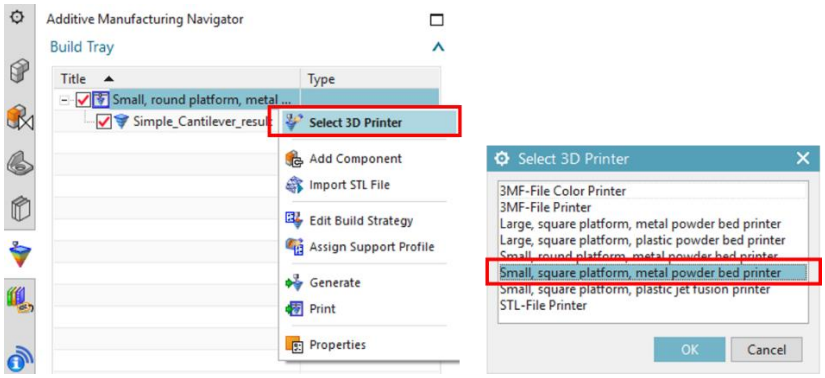
2. In the **Check Printable Volume** dialog box, **Select Body** from the graphics window
3. In the **Volume Display** group, select **Show = Both** to visualize both Printable Volume and Product Volume
4. Hover mouse over the  symbol either in the graphics window or in the **HD3D Tools** navigator on the **Resource bar** to see information about test results

   Notice in the graphics window, the result of Printable volume test

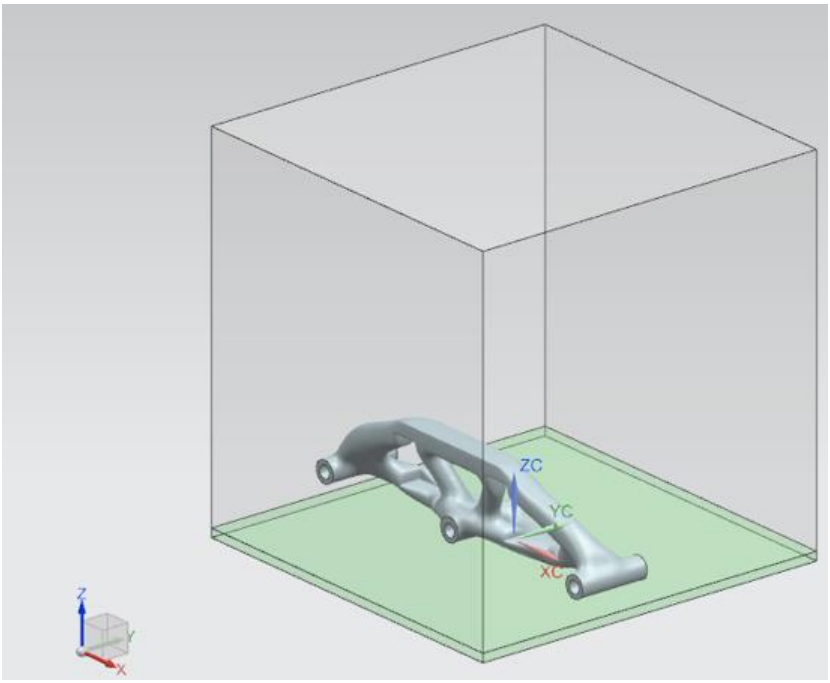


5. As the part dimensions exceed the build tray dimensions, the test has failed. Therefore, either the part can be repositioned to fit into the printer or a different printer can be selected

   NOTE: Re-positioning the part may compromise with optimum build orientation; the build height and hence the build time and consequently the build cost may increase. Therefore, in this case, assuming that the default orientation is the optimum orientation, we select a different printer
6. In the **Additive Manufacturing Navigator** menu, in the **Build Tray** group, right click on the printer, then click on **Select 3D Printer.** In the **Select 3D Printer** dialog box, choose a printer from the list

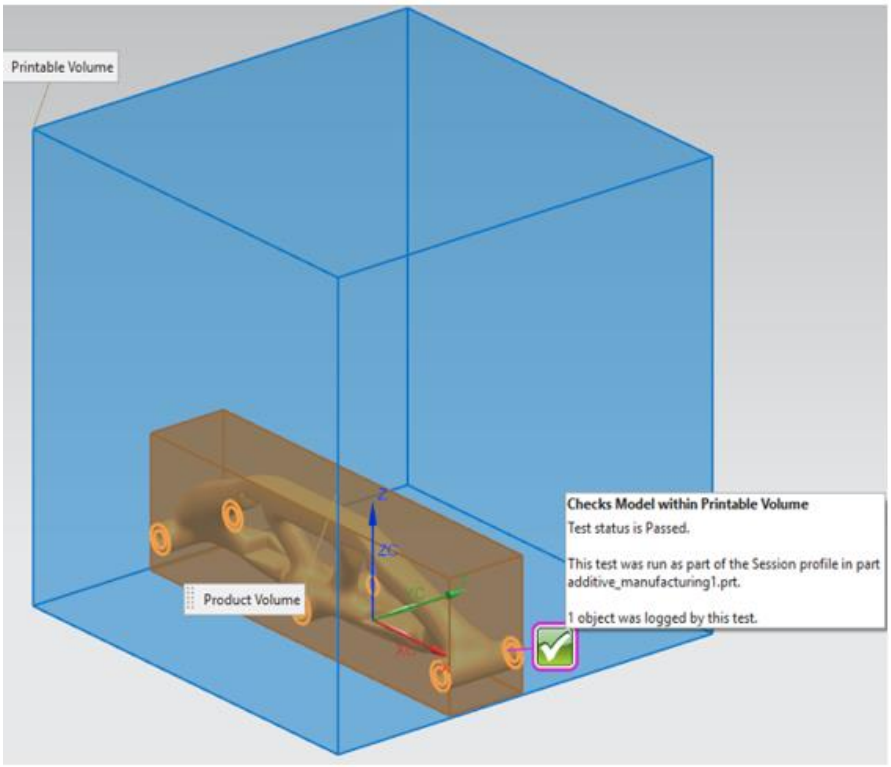


Notice the part mounted on a different printer



7. Check once again for **Printable Volume**
8. Hover mouse over the  symbol either in the graphics window or in the **HD3D Tools** navigator on the **Resource bar** to see information about test results

   Notice the part has now passed the **Printable Volume** checker.



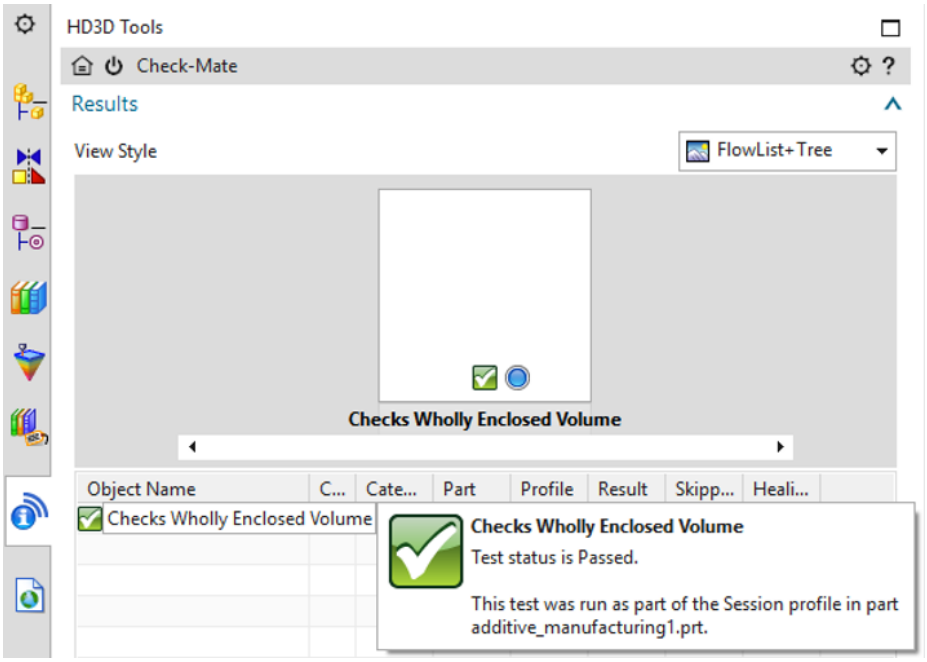
9. Close the **Check Printable Volume** dialog box

WHOLLY ENCLOSED VOLUME

The **Wholly Enclosed Volume** checker checks for support requiring areas on the part that are entirely enclosed by other regions in a solid body. This test results in either passed ✅ - No enclosed volumes or failed ❌ - Atleast one enclosed volume

1. On the Ribbon bar, in the **Analysis** tab, click on **Wholly Enclosed Volume**
2. In the **Check Wholly Enclosed Volume** dialog box, select body from the graphics window
3. In the Resource bar, hover mouse over the ✅ symbol to check result in the **HD3D Tools** navigator, under **Results** tab

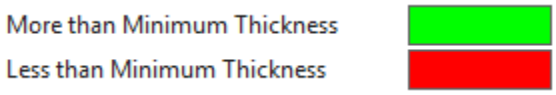
Notice the result status for **Wholly Enclosed Volume** is passed



4. Close the **Check Wholly Enclosed Volume** dialog box

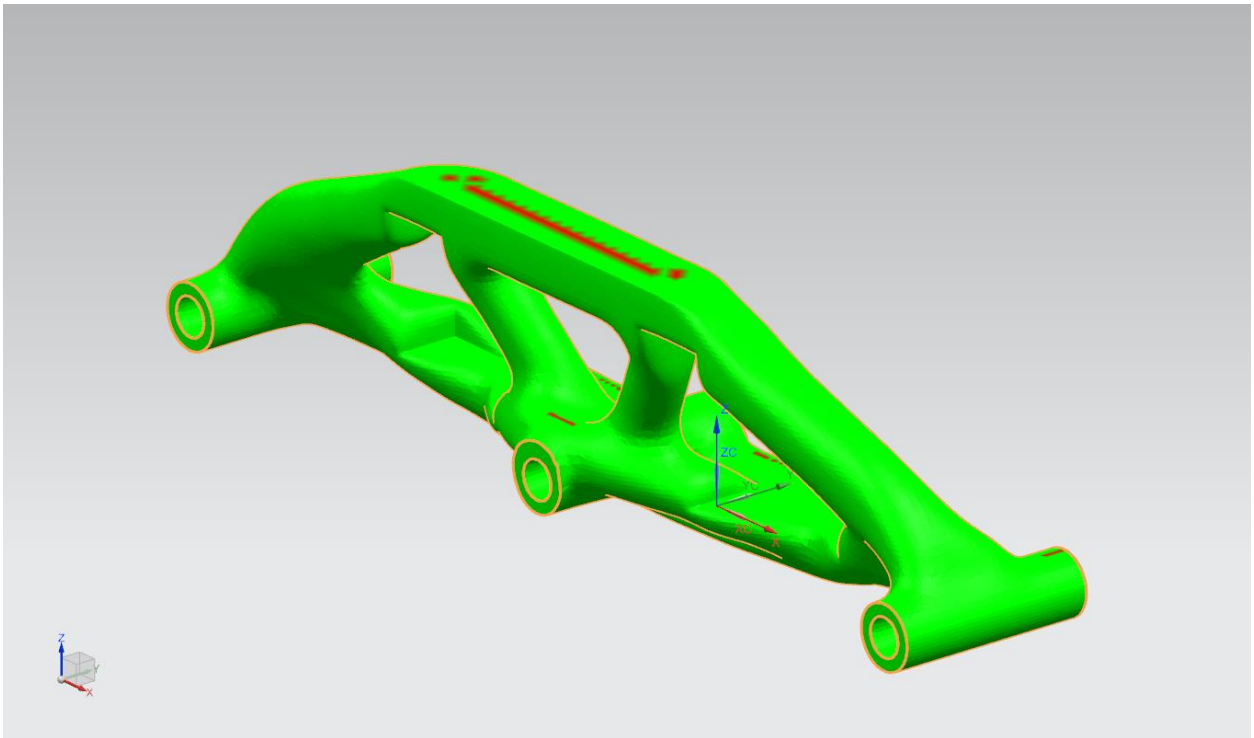
WALL THICKNESS

The **Wall Thickness** checker checks the minimum wall thickness of the model for Additive Manufacturing. The test results are displayed graphically into highlighted areas on the part that are more than and less than the Minimum Thickness



1. On the Ribbon bar, in the **Analysis** tab, click on **Wall Thickness**
2. In the **Check Minimum Wall Thickness** dialog box, select the body from the graphics window
3. **Notice** in the **Thickness** group, the default value for **Minimum Thickness** is 0.4. Keep the **Show Only Less than Minimum Thickness** box unchecked
4. In the **Process Results** group, click **Calculate Thickness**  to display results

Notice the areas less and more than the minimum thickness highlighted on the part



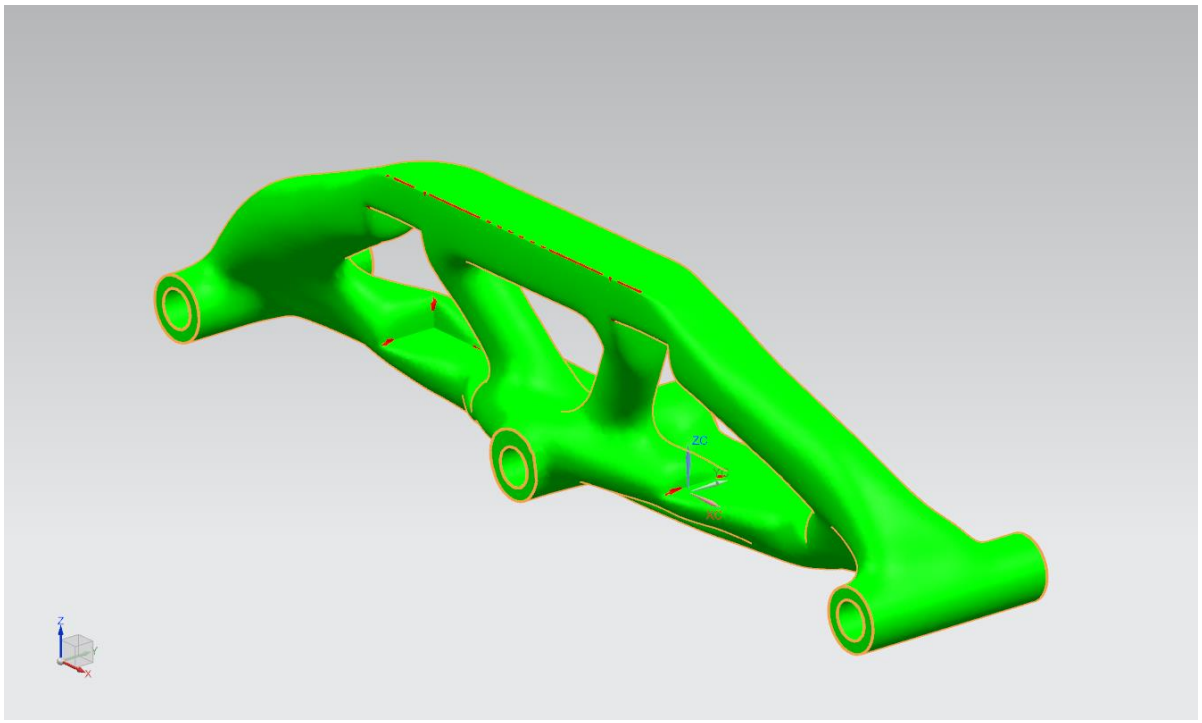
5. Close the **Check Minimum Wall Thickness** dialog box

MINIMUM RADII

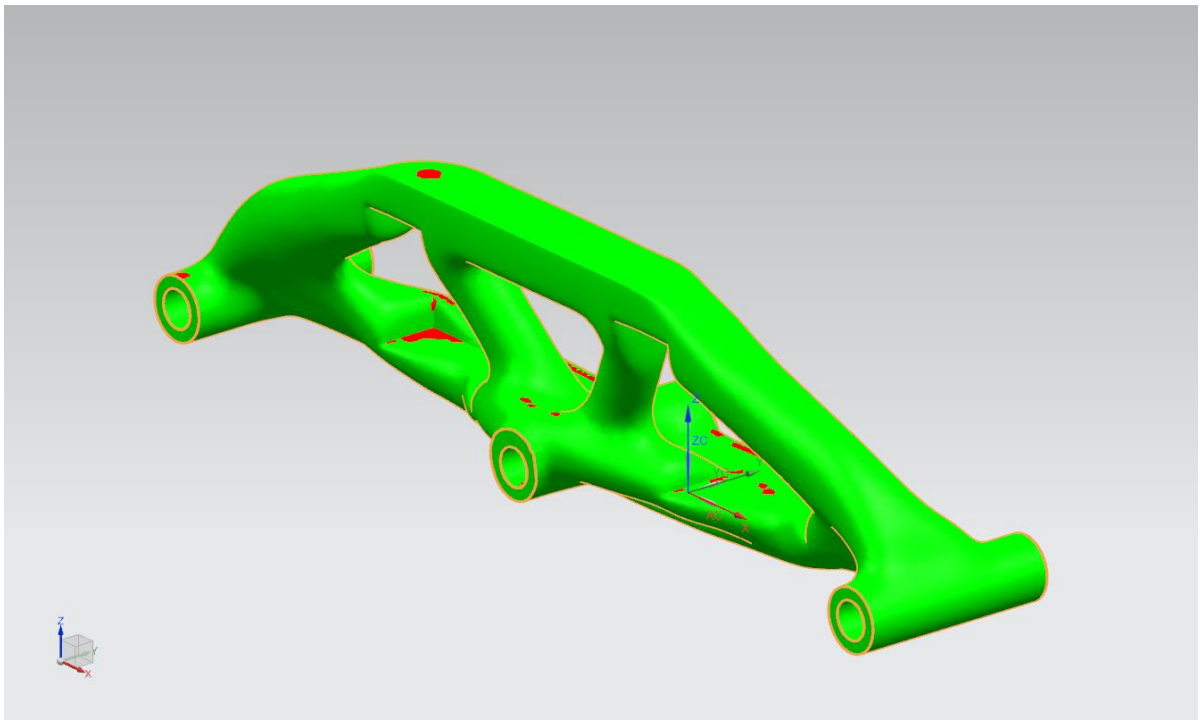
The **Minimum Radii** checker checks the minimum radii of the model for Additive Manufacturing. The test results are displayed graphically into highlighted areas on the part that are more than and less than the Minimum Radius

1. On the Ribbon bar, in the **Analysis** tab, click on **Minimum Radii**
2. In **Check Minimum Radii** dialog box, select the body from the graphics window
3. **Notice** in the **Radius of Curvature** group, the default value for **Minimum Radius** is 0.5. Keep the **Exclude Convex Radii** and **Show Only Less than Minimum Radius** boxes unchecked
4. Select from drop-down menu whether the results are to be displayed in **3D Minimum** mode or **Sectional** mode

Notice the results in **3D Minimum** mode



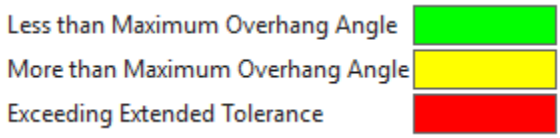
Notice the results in **Sectional** mode



5. Close the **Check Minimum Radii** dialog box

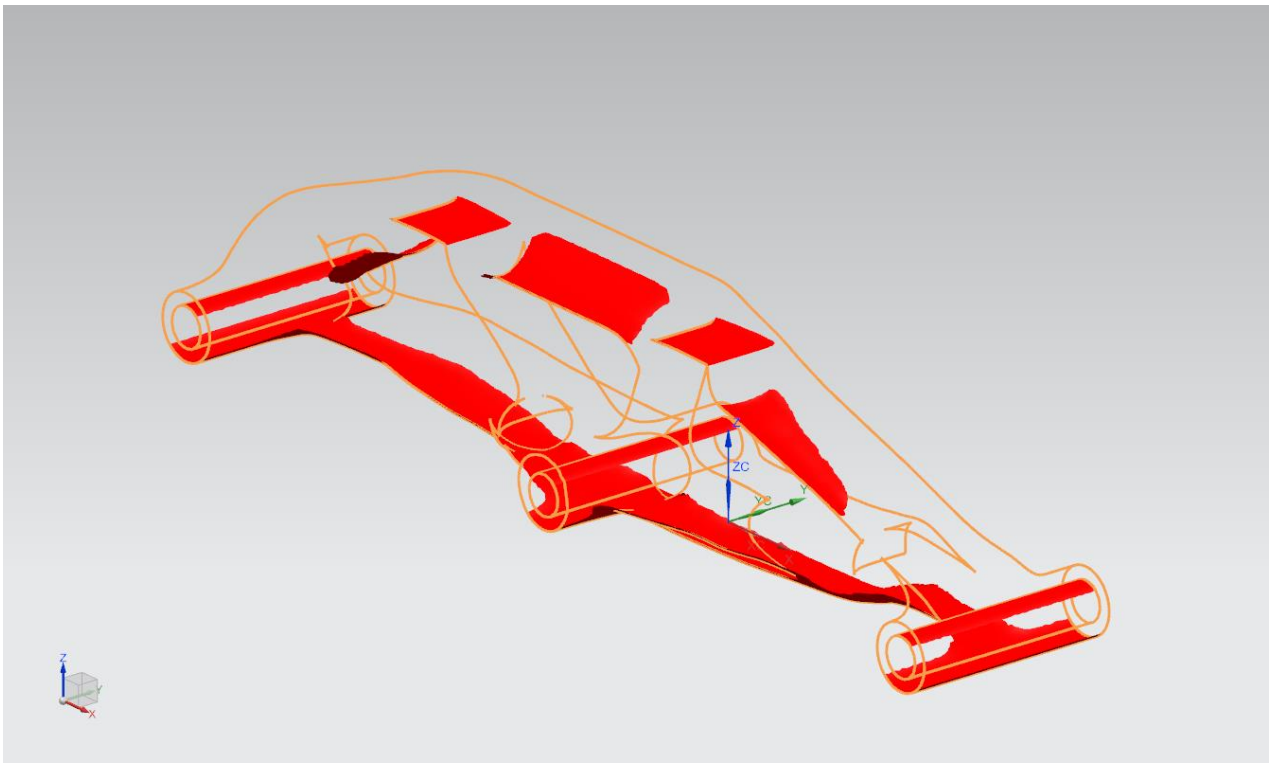
OVERHANG ANGLE

The **Overhang Angle** checker calculates and displays the area requiring supports on a part positioned in a specified orientation. The test results are displayed graphically into highlighted areas on the part less than and greater than the Maximum Overhang angle including Extended Angular Tolerance



1. On the Ribbon bar, in the **Analysis** tab, click on **Overhang Angle**
2. In **Check Maximum Overhang Angle** dialog box, select the body from the graphics window
3. In the **Angle** group, **notice** that the **Maximum Overhang Angle** is 45 degrees. Check the **Show Only Exceeding Overhang Angles** box

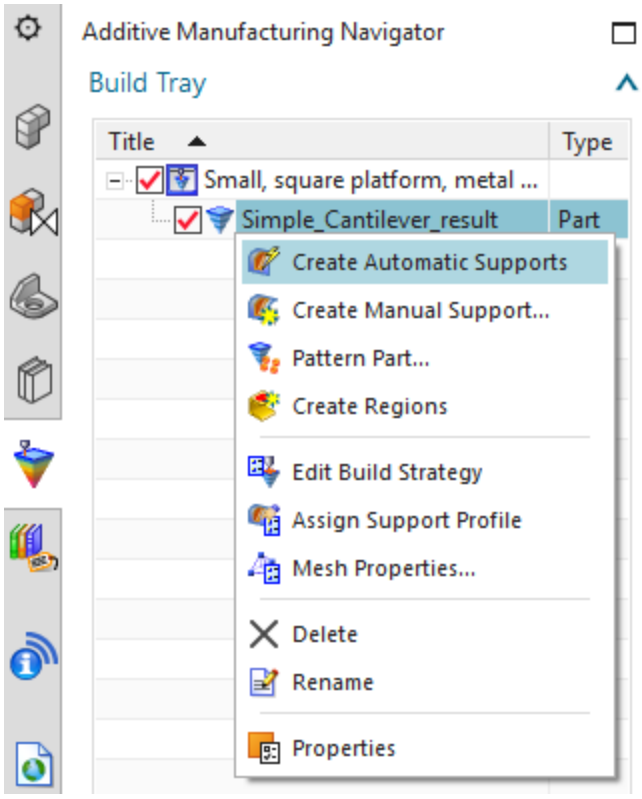
Notice in the **Overhang Area** group, the area requiring supports is displayed as a numeric value 2360.376959 mm^2 and the areas are highlighted on the part in the graphics window



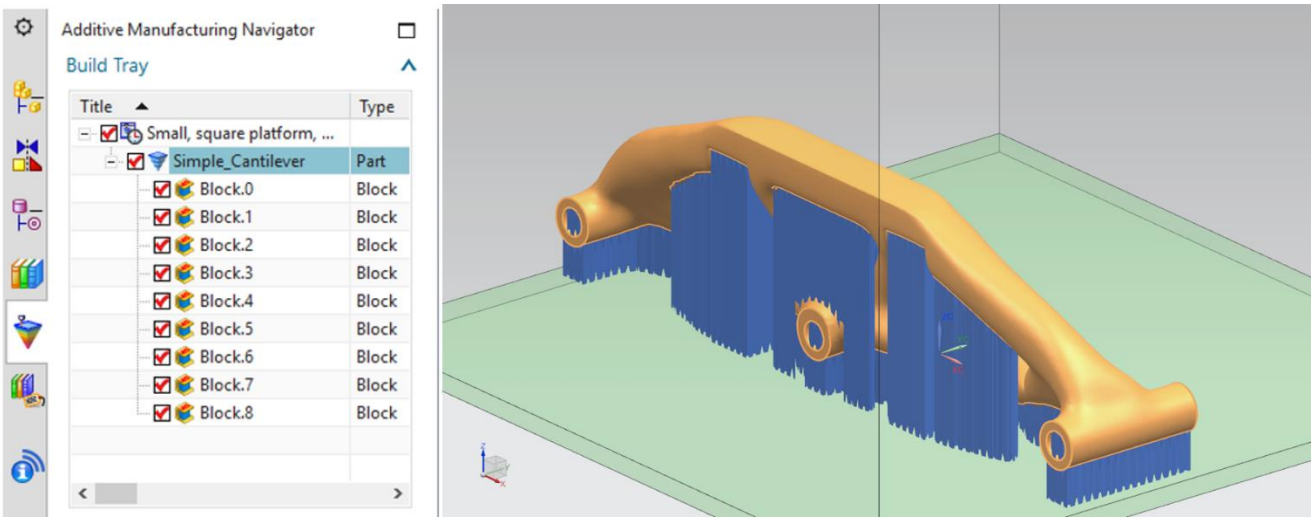
4. Close the **Check Maximum Overhang Angle** dialog box

CREATE AUTOMATIC SUPPORTS

1. For automatic support generation, in the **Additive Manufacturing Navigator** menu, right click on the part in the **Build Tray** group, from the drop-down menu, select **Create Automatic Supports**



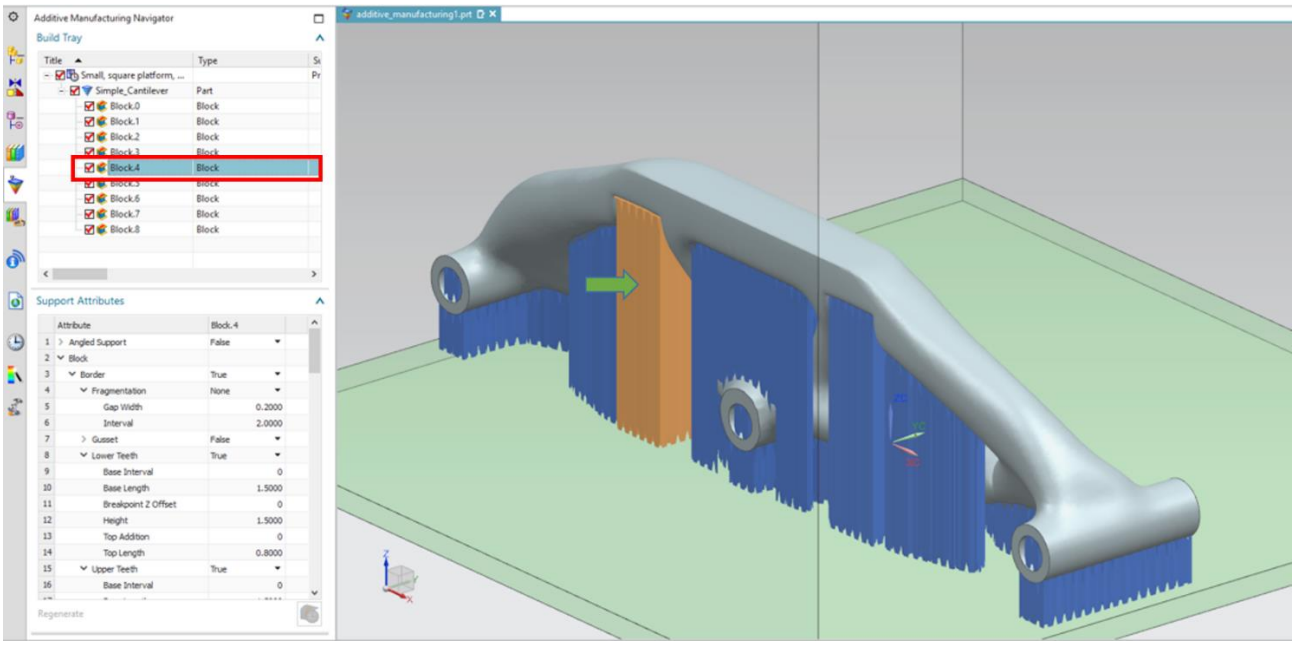
Notice the block supports generated in the **Additive Manufacturing Navigator** menu and on the part in the graphics window



SUPPORT ATTRIBUTES

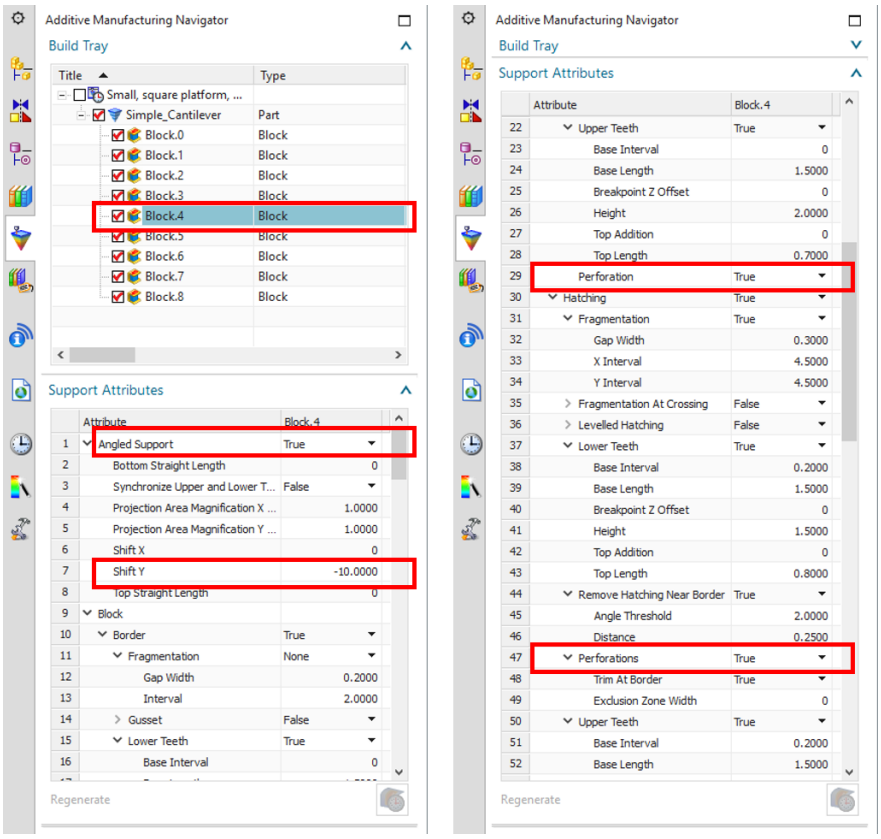
1. To edit support attributes, select a support block on the part in the graphics window. **Notice** the selected block-name highlighted in the **Additive Manufacturing Navigator** menu, in the **Build Tray** group
   NOTE: The generated Block name of selected support block could change the next time automatic supports are generated on the same part

Notice the selected support block in the graphics window



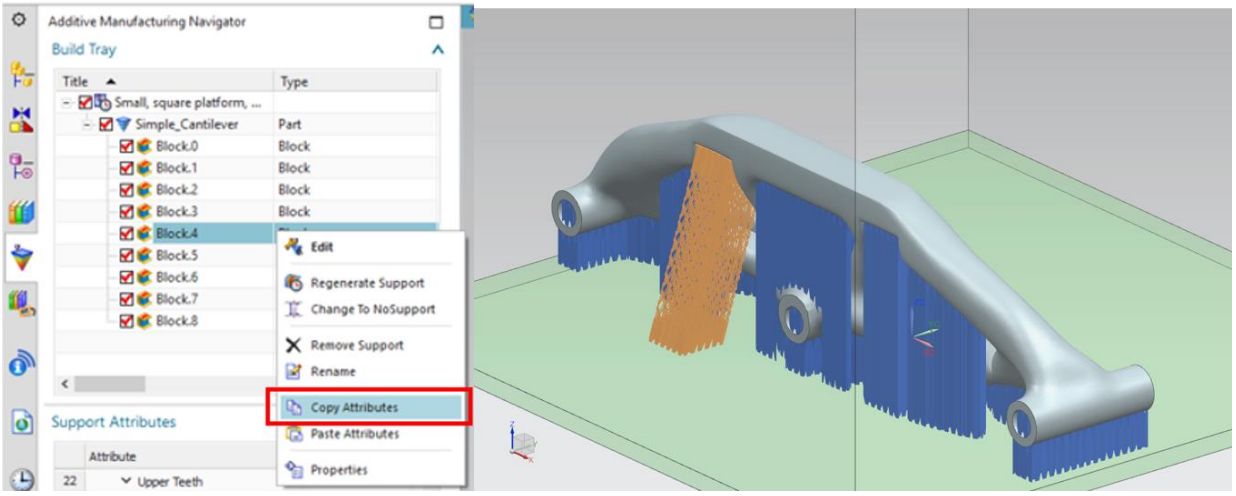
2. In the **Support Attributes** group, change the support attribute values as shown for the selected support block
   Angled Support = True
   Shift Y = -10.0000
   Perforations = True
   NOTE: Set the **Perforations** 'True' twice- for perforations both inside the support block volume and for perforations on the block border

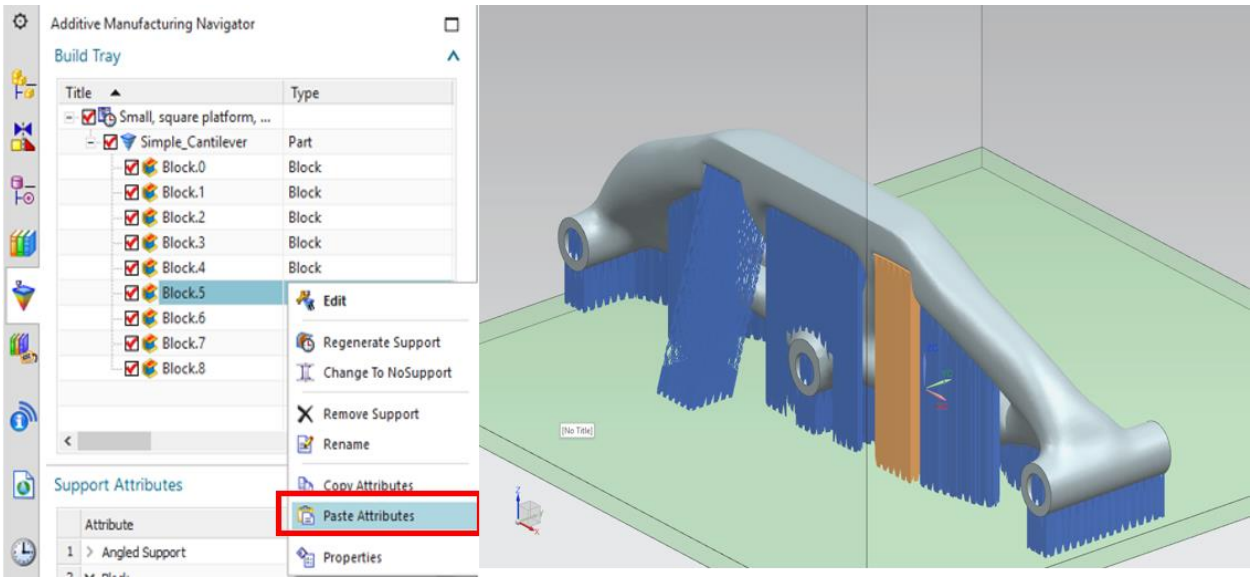
Notice the support attribute values in the graphics in the **Additive Manufacturing Navigator**



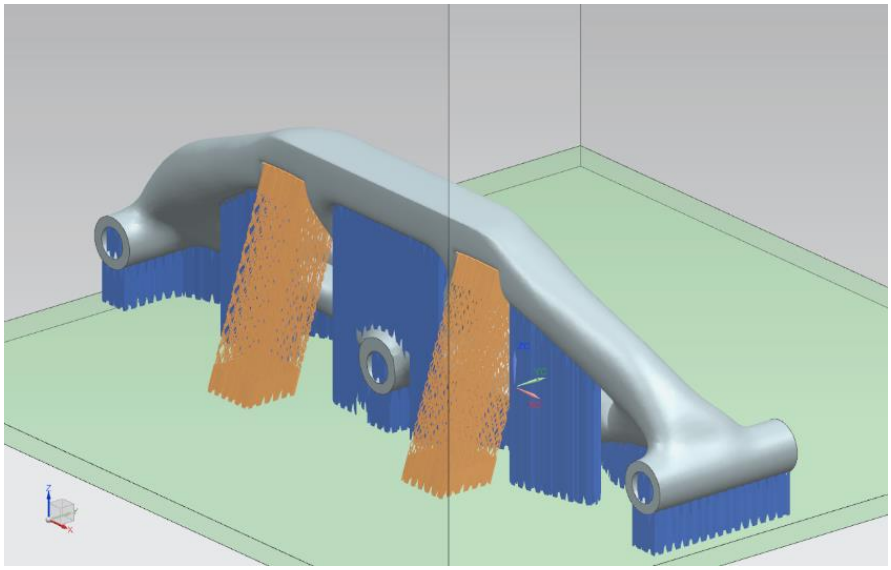
3. After changing the values, click on **Regenerate**  to regenerate support
4. To copy attributes of one support block onto another, in the **Build Tray** group-
    - Right-click on the support block and select **Copy Attributes**
    - Now, in the desired support block, right-click and select **Paste Attributes**

    NOTE: Attributes can be pasted into multiple support blocks (multi-select using **Shift** button)





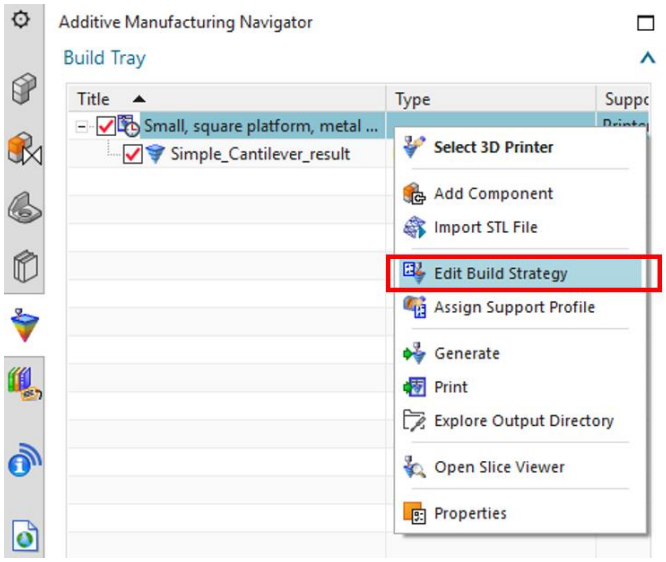
Notice the change in support blocks



ASSIGN BUILD STRATEGY

The printer needs to be assigned a build strategy; like setting certain machine parameters, material, slice profile, etc. Build strategies can be configured in the Build Processor manager of **Materialize**. For this tutorial, we use a demo build processor with a default build strategy

1. To assign a build strategy, in the **Additive Manufacturing** navigator, in the **Build Tray** group, right-click on the printer. From the list, select **Edit Build Strategy**

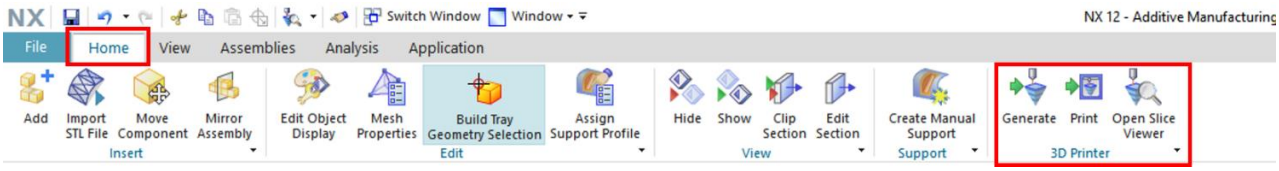


2. In the **Edit Build Strategy** dialog box, set each column to **Default**

3D PRINTER

Now that the support structures are generated, build strategy is defined, the part is ready to be sent for printing. On the Ribbon bar, in the **Home** tab, under **3D Printer** group, click on each icon-

1. **GENERATE:** To send the part file for pre-processing printer specific files, click on **Generate**
2. **PRINT:** To queue the job file for building into the printer, click on **Print**
3. **OPEN SLICE VIEWER:** To view all the slices and hatch pattern on each layer of the printer job file, click on **Open Slice Viewer**



Notice the hatch pattern on the slice