



# 3DPRINTING EXERCISE

## Create Supports & Prepare for Print

Tutorial\_V11 - Updated: 3DXpert 16 Beta release

## Table of Contents

Part 1 – Creating Supports .....	3
Part 2– Prepare for Printing. ....	61
Part 3 – Manual Cones – 2 Points.....	79

In this exercise, we will use the various tools and options to prepare the part for printing by building a variety of metal supports.

## Part 1 – Creating Supports

3DXpert includes various types of supports that can be added to any area on the part. There are Cone, Solid, Wall, Lattice, Solid Wall, Power Support & Skirt supports. As you will see, some of them (like Wall supports) are built on the contour defining a region, so dedicated region creation tools are available too.

Before we begin, it is important to know the regions and supports used in this exercise are not meant to be a guide for the actual printing of this part. The goal here is to go through the various types of supports that are available in 3DXpert.

Our first step is to look for support regions along the part. This is done automatically by the system.

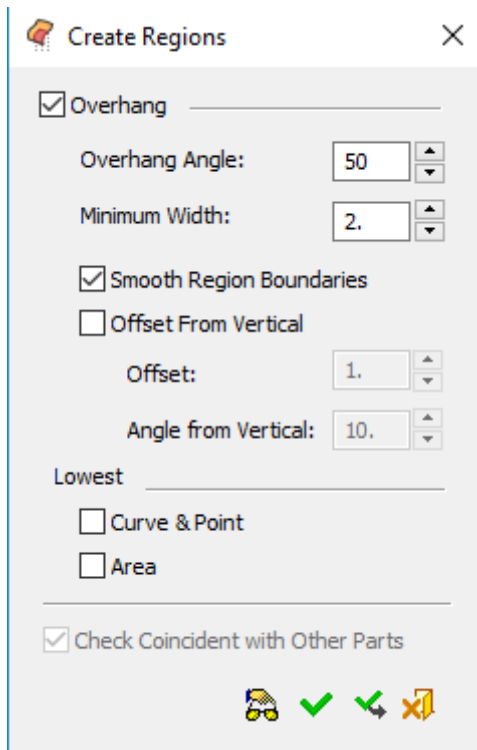
1. Unpack the file 3DXpert-Exercise1 Print Prep finish.ctf and load the project.
2. Click the Support Manager button from the toolbar.



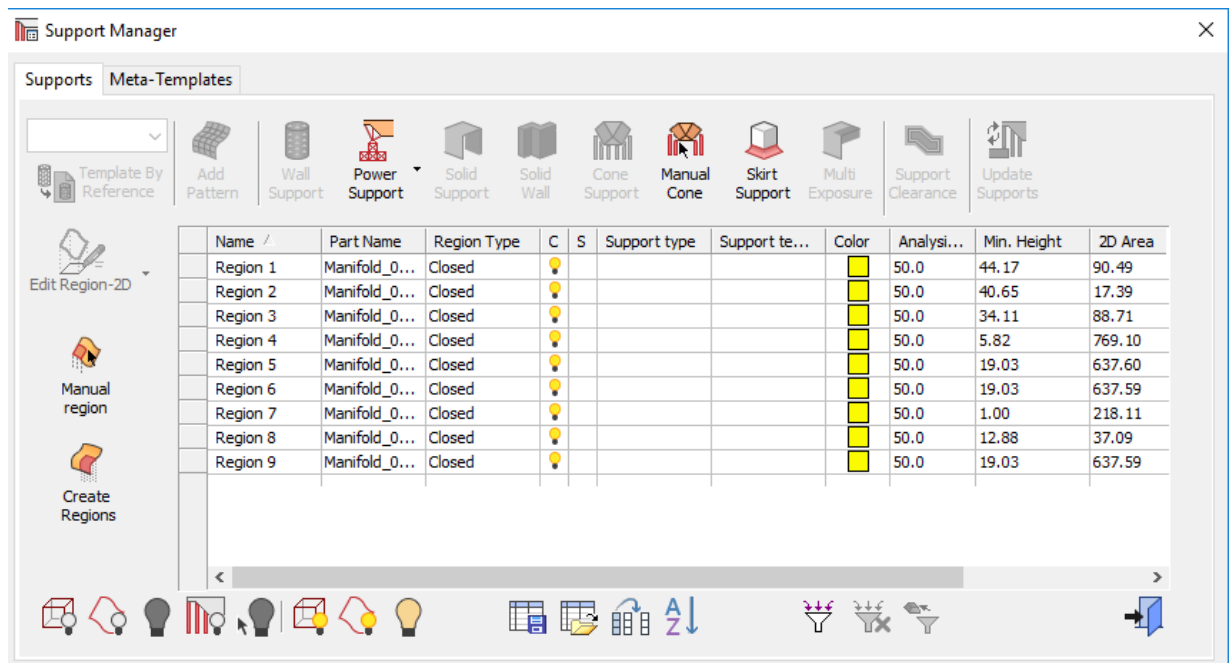
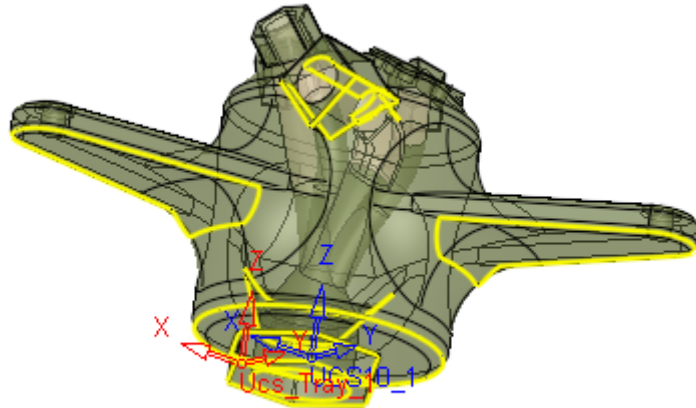
Since in this project there are no support regions defined yet, an additional window called Create Regions shows up.

(Create regions is also available through the Support Manager.)

3. Set the **overhang** value to **50**, as we have set at the beginning of the project. We can also set the Minimum Width for areas that are 'self-supporting'. This has no impact on the lowest areas of the part, where supports are mandatory, but can help to filter out higher areas whose width is less than the specified value and will not be recognized as a region.
4. Keep the option **Smooth Region** Boundaries checked.
5. Uncheck the two **Lowest** options (Curve & Point and Area).



- Clicking the Preview button displays regions without actually creating the regions. You can skip this stage (as we already know the part) and press the **OK** button. This creates the regions that need supports by creating their boundary curves.



NOTE: if you did NOT uncheck the Lowest options, the Support Manager includes additional regions. As we will not work on these in this exercise, you can either hide

them (select the rows and click Hide

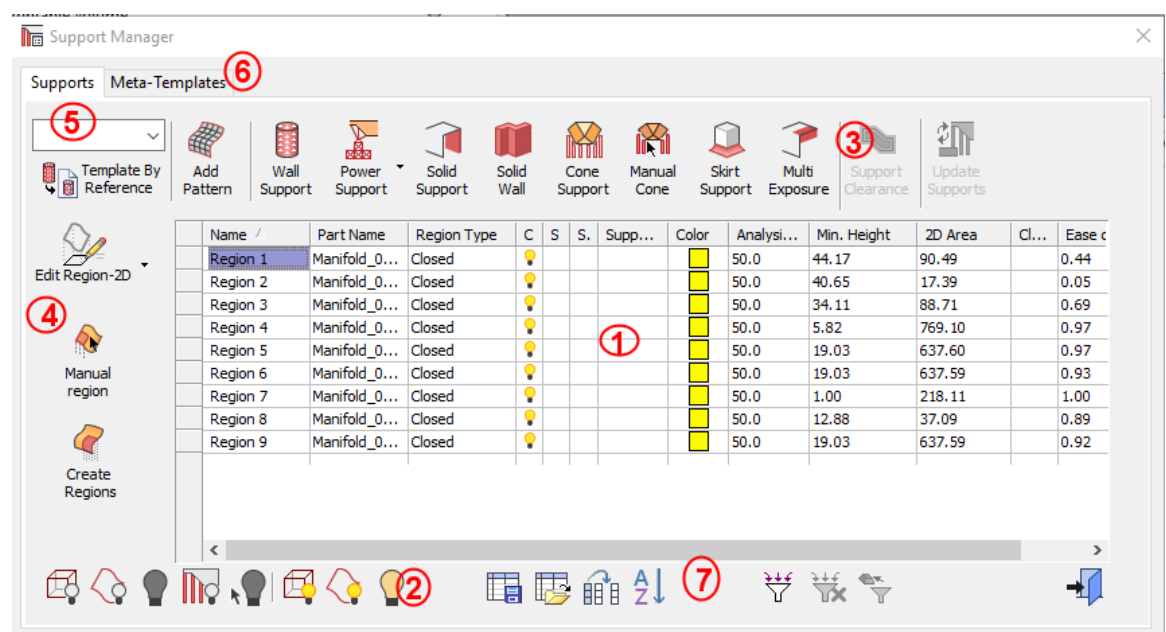



or right mouse click and delete them.

7. Now that we have regions defined, the Support Manager itself is filled with data. The Support Manager lists all the regions created on the part and includes all functionality needed to edit the regions and create supports on them. You can edit each of the support regions and create various kinds of supports.

The Support Manager dialog is composed of the following:



- (1) Table of regions created on this part. Each row stands for a different region with its parameters and supports (at this stage, no support is yet defined).
- (2) Visibility buttons for regions and supports.
- (3) Support creation tools (all are active in this image, as a region is selected).
- (4) Tools for creating and editing support regions.
- (5) Apply Template (will not be discussed in this exercise).
- (6) Meta Templates (will not be discussed in this exercise).
- (7) General dialog settings.



8. Pick some regions on the screen and see that the relevant rows are highlighted. This works vice versa too.
9. You can hide and show each region (and later on its support) individually by clicking its bulb in the table or show/hide all by clicking the relevant button in the lower area of the dialog.
10. Press the Hide Region Contours  button and click the bulb to show one Region.

## Note on Removal Analysis

The last column in the table is called 'Ease of Removal'.

<div>  Support Clearance          Update Supports       </div>			
	2D Area	Cl...	Ease of Remo...
	90.49		0.44
	17.39		0.05
	88.71		0.69
	769.10		0.97
	637.60		0.97
	637.59		0.93
	218.11		1.00
	37.09		0.89
	637.59		0.92

For each region, a score between 1 and 0 is assigned by the system.

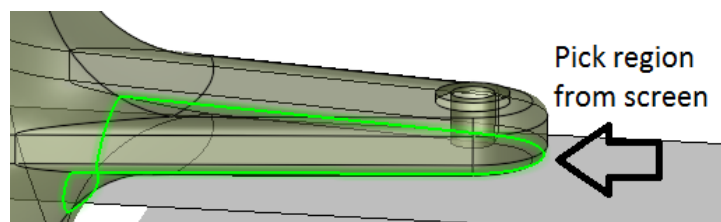
This score stands for how easy it will be to remove the supports created in this region by milling tools.

The value 1 means the supports created for this region should be very easy to remove.

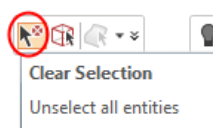
The value 0 means they will probably be very hard to remove.

Important: throughout this exercise, 'your' region number may not have the same region number shown in this exercise.

To follow the exercise, pick the matching region using the cursor directly from the screen:



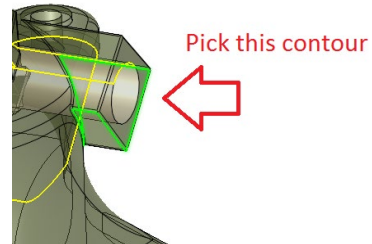
If you wish to reset the selection (i.e., unselect the region), you can always press the Clear Selection button from the floating toolbar.



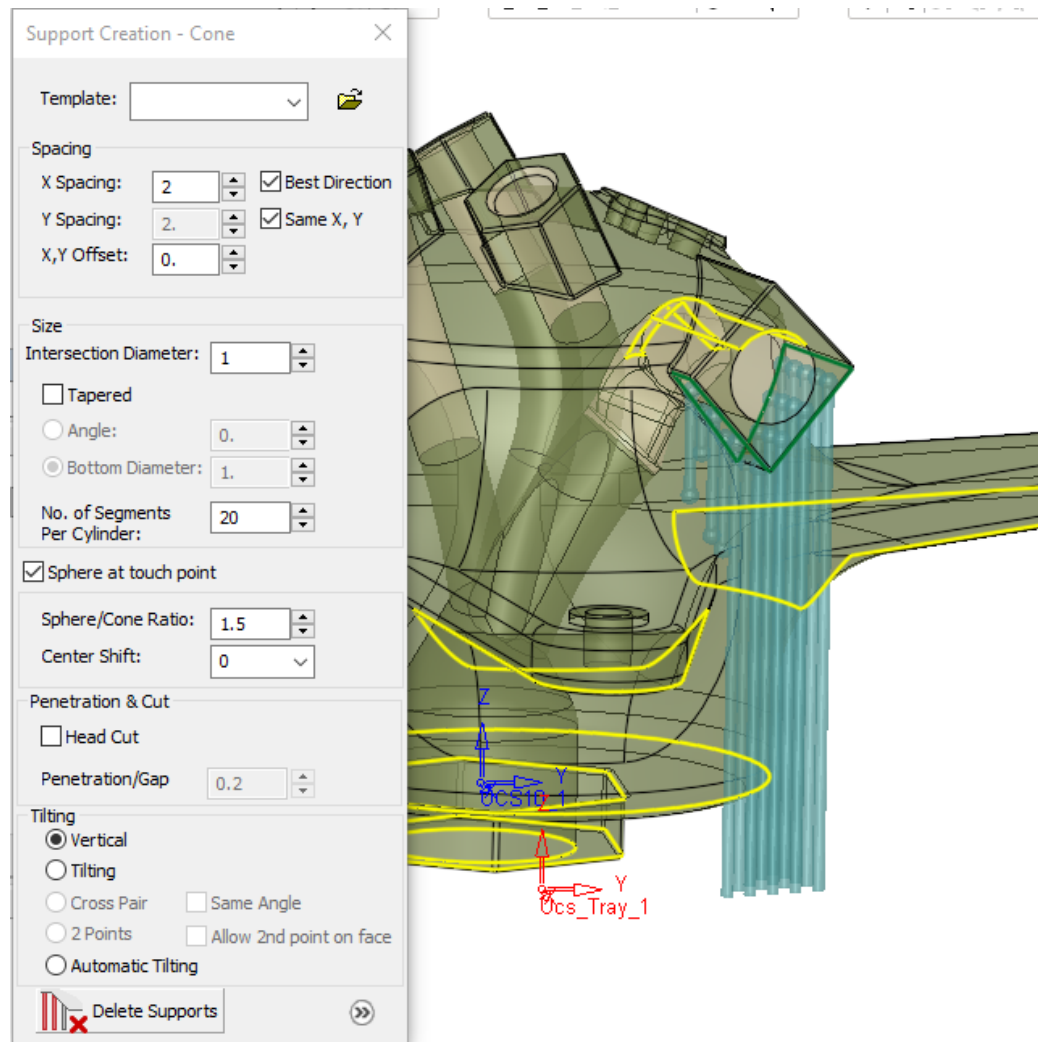
On this part, nine support regions were detected. Let's now add various types of supports to each of these regions.

11. Select the region shown in the image:

Click the Cone Support button.

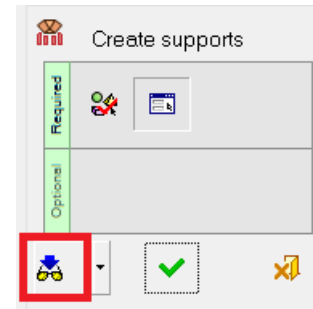


The system creates several cones based on the default parameters and the cone support creation dialog shows up. You can now change the number of cone supports, their size and orientation.

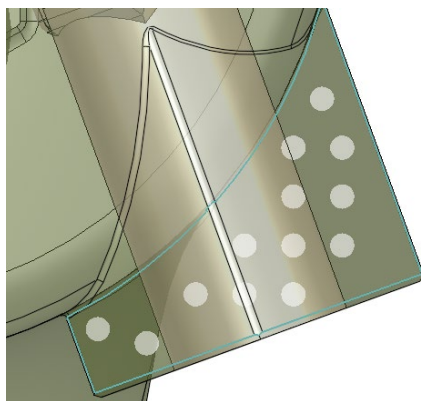
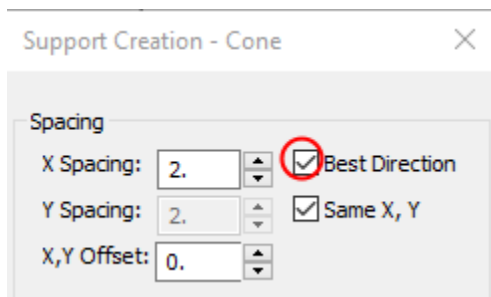




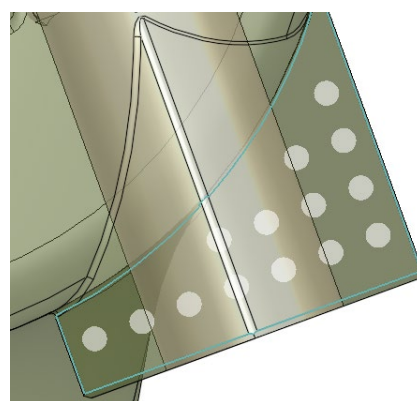
12. Change the Spacing parameters and the Size as you like and watch them dynamically change on the display. If needed, click the Preview button.



13. Use Spacing to add 'empty rows' in each direction.
14. If **Best Direction** is checked **ON**, the X & Y directions are placed along the minimum bounding box of the projected contour.

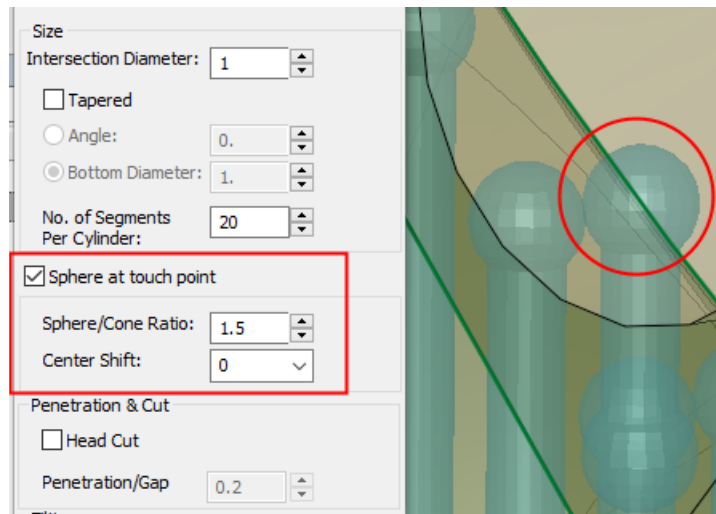


Best Direction = OFF



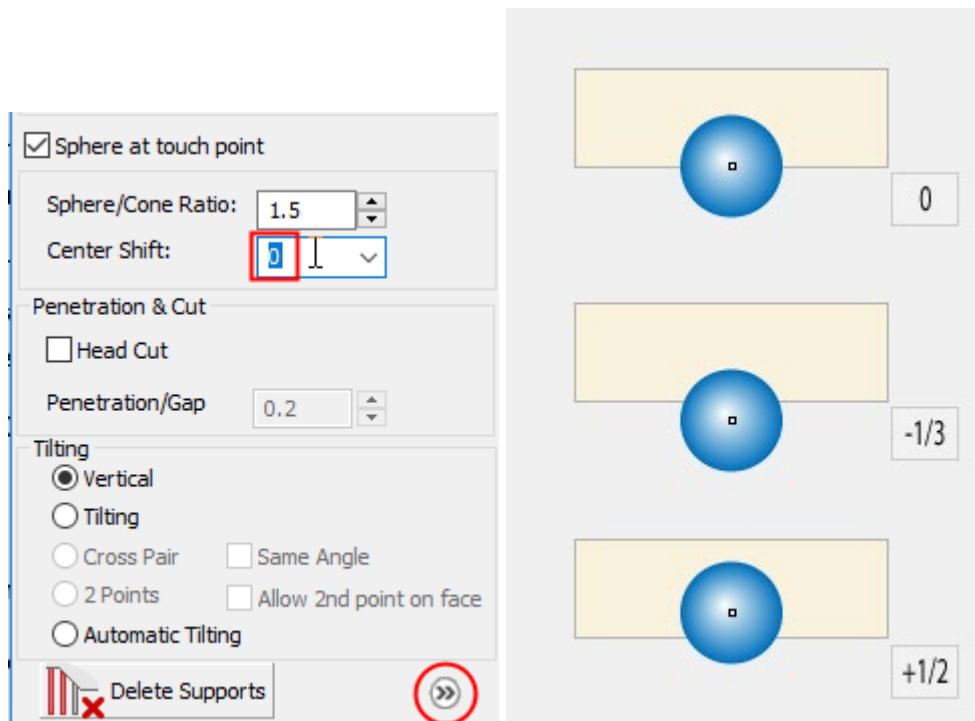
Best Direction = ON

You can also add a taper so the cone will be thicker at the bottom or add spheres at the connecting points with the material (image below).

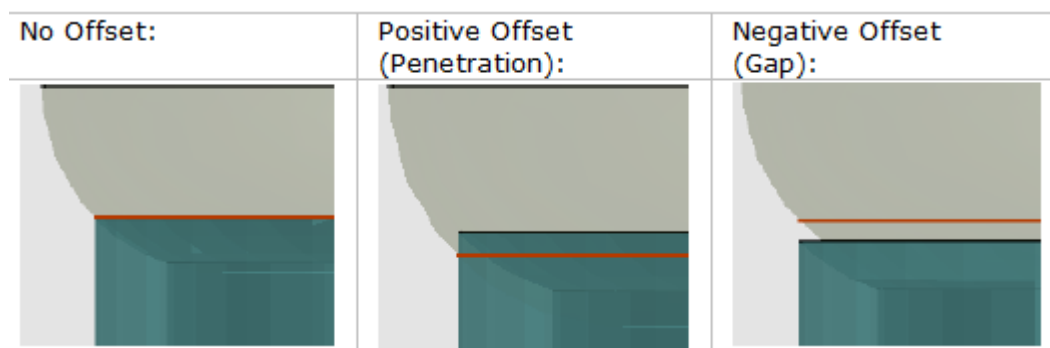
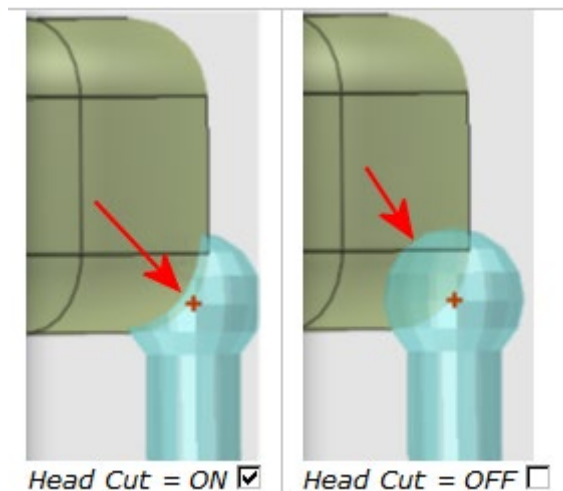


As supports are integrated into the part, the Center Shift sets how deep these connectors will go inside the material. A zero shift value means that the center of the sphere will be placed exactly on its touch point with the geometry. A + 1/2 value means that the center will move 1/2 of the RADIUS up, and so on.

Click the Center Shift field and expand the dialog to display the image explaining this parameter.



The Penetration/Gap value sets a distance between the cone and the part: a positive value means that the support will enter the part by that distance (Penetration) while a negative value means that the support will be outside the part (Gap).

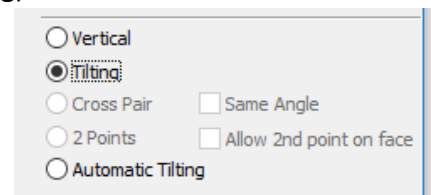


This will be visible only after you press the OK button and the supports are created (what you see now is their preview).

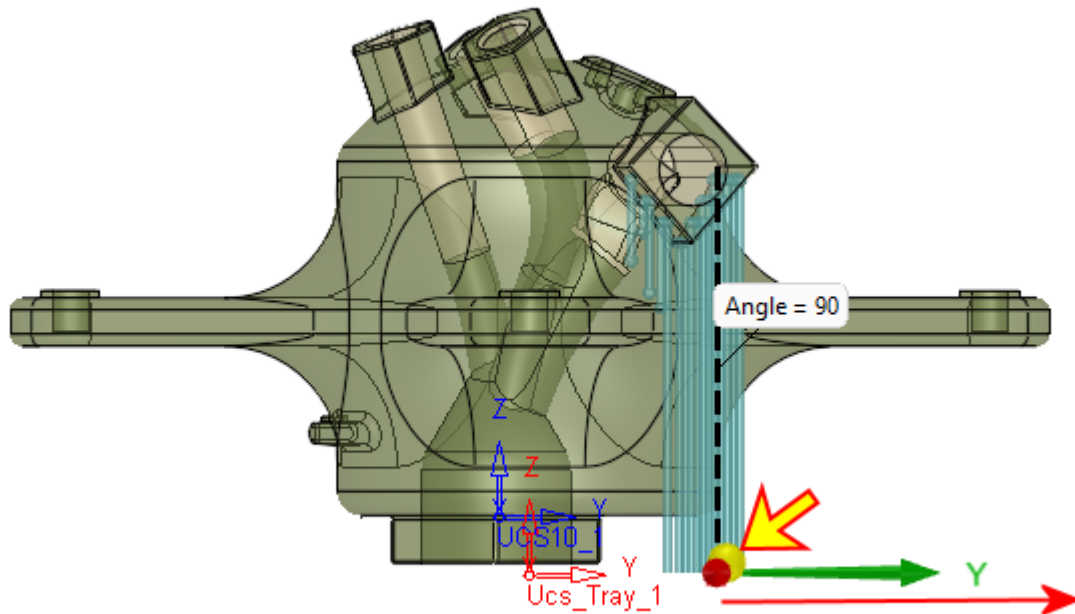
Right now, the cone supports are vertical and so are located too close to the vertical walls of the part. You can also add tilting to the supports.

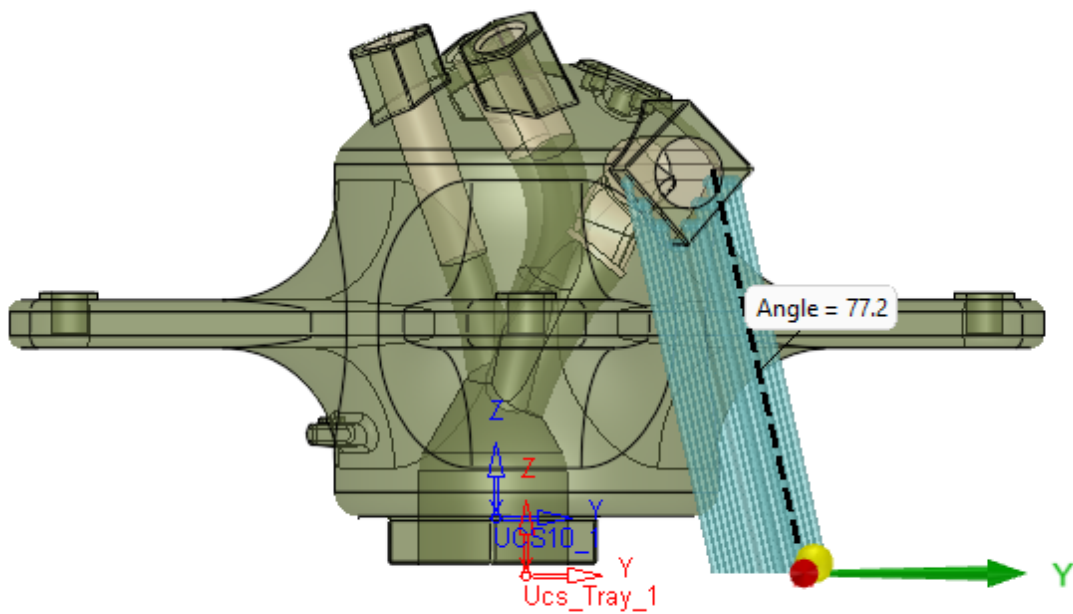
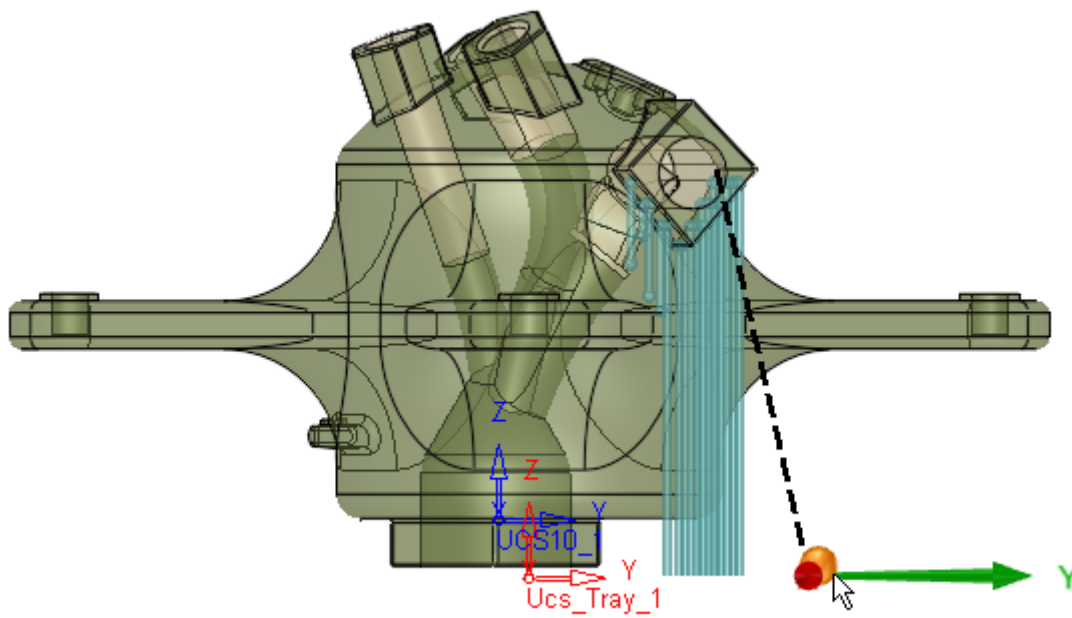
15. Click the **Tilting** radio button (the second option in the dialog):

16. A dashed black line will show alongside the cone supports.

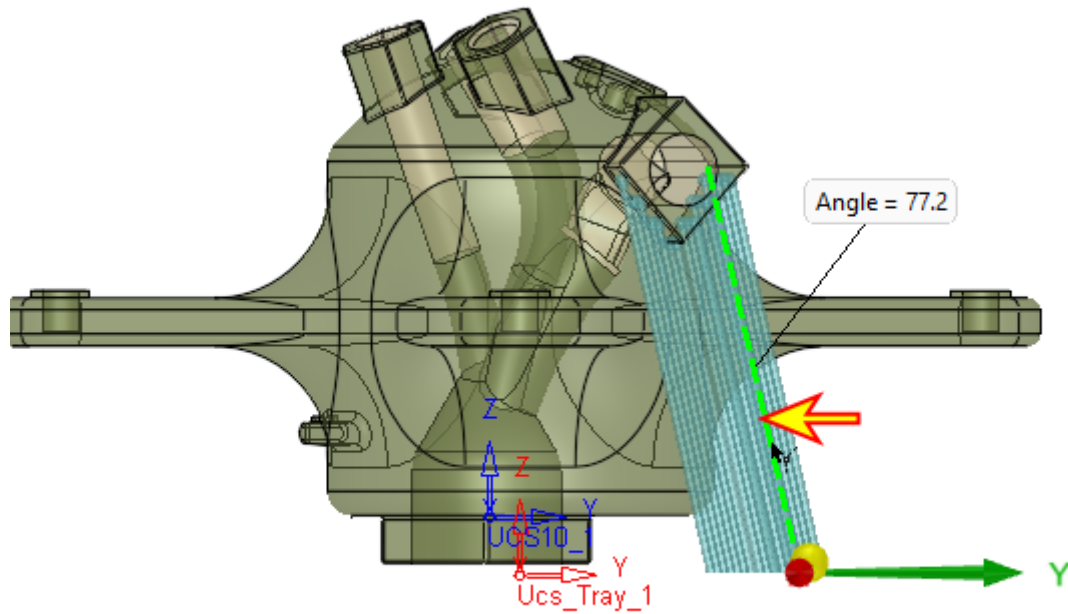


17. A Triad is displayed, together with a dashed black reference line along the supports. To tilt the support, pick the **yellow** Triad ball and drag it. As you release the mouse, the cone supports will move accordingly. An **Angle** box displays the current tilt angle of the support.

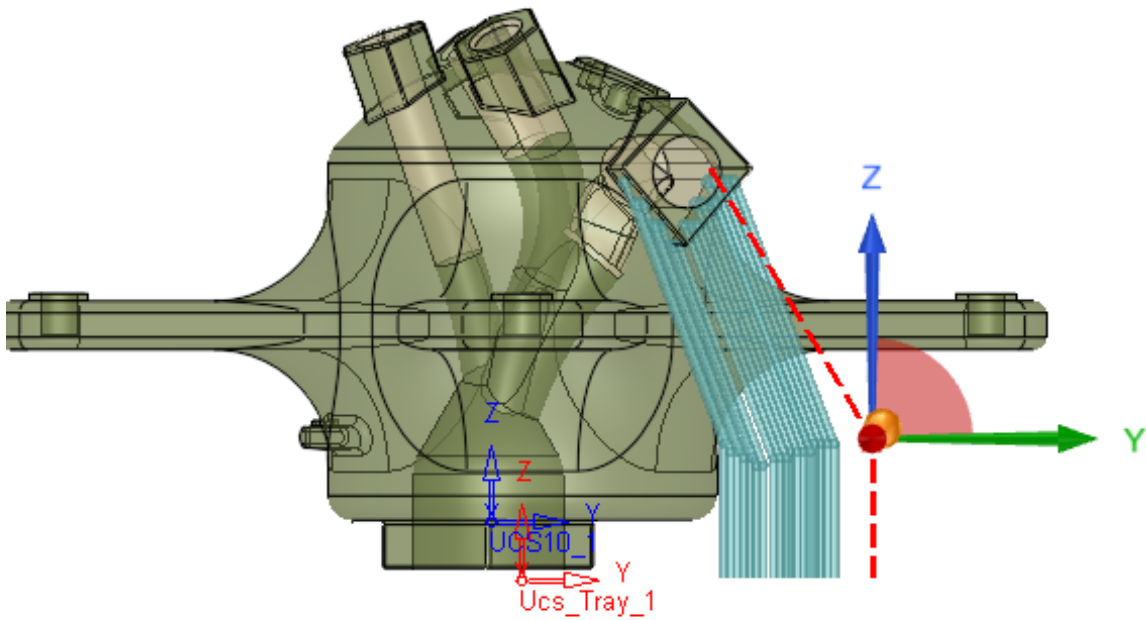




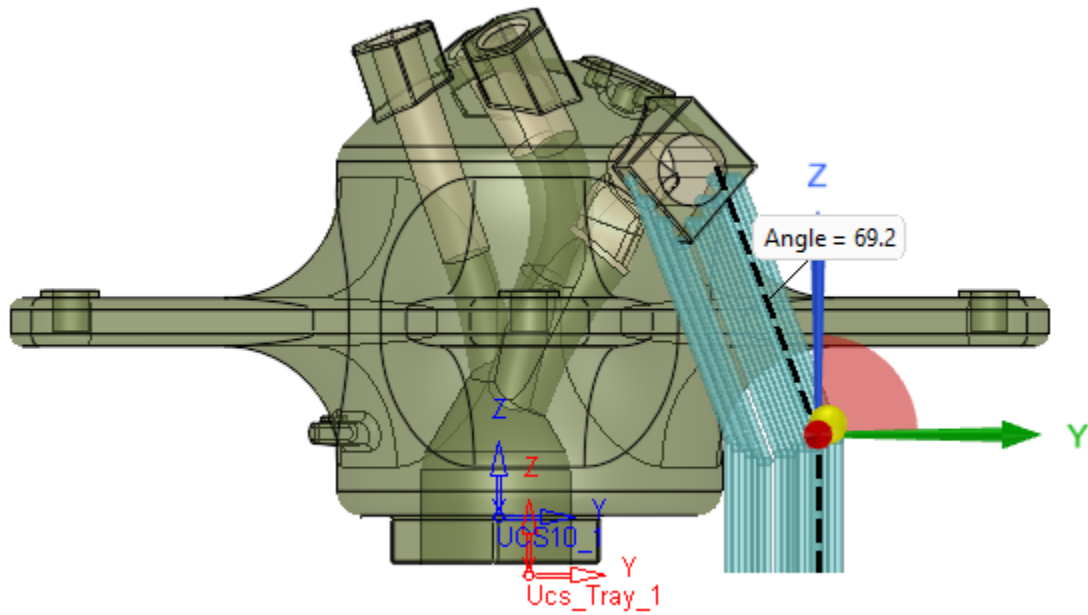
18. Now click somewhere along the line and drag it – it will add a breakpoint , which you can dynamically set:  
 Adding a breakpoint to the tilt



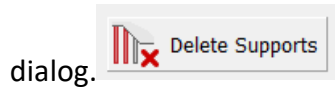
Note: If the line changes its color from black to red, this means that the tilt angle is too big and therefore these supports cannot hold themselves.



The tilt angle is also shown on screen (making sure that the Tilt of the supports is within the limits)



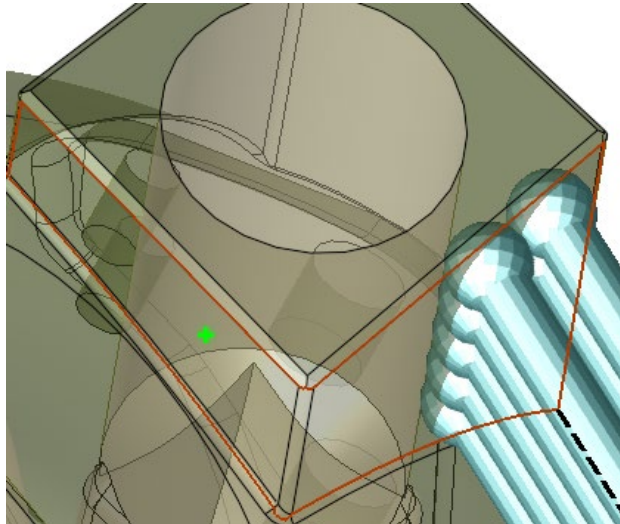
19. To remove individual cone supports, press the Delete Supports button from the



dialog.

20. Pick the supports to remove (note that when picked, they turn red) and confirm by pressing the middle mouse button. To pick several cone supports together, you can also select them by box and press the middle mouse button.

21. You can also add individual supports interactively. Hover with the mouse on the part where you wish to add cones - the green preview point is shown for better location - and click the left mouse button to add the cone at that point.



As mentioned before, while you are in the Support Creation tool, if you wish to remove a Cone just added reselect it.

22. When you finish creating Cone supports, right mouse click to show the Feature Guide and press **OK**.

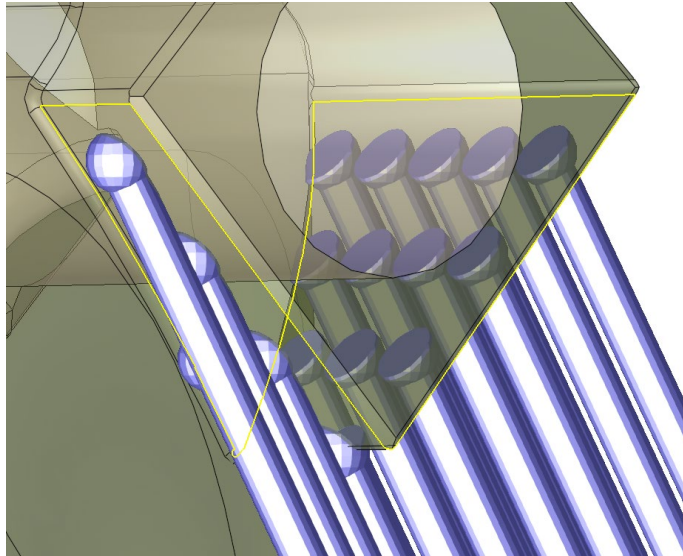
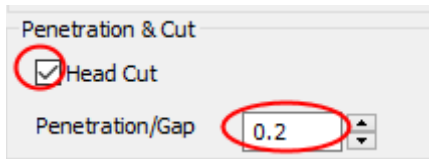


Once you invoke any function, the Feature Guide is also available on the left side of the screen.

See the result:



The cone supports are cut with the part. If you have defined a Penetration/Gap value, they will be positioned accordingly, at an offset from the part's faces.



The Support Manager is back on screen and notice that it shows that Cone supports were created on this region:

Region 4	Closed			Manifold_05...		50.0	40.03	10.33		0.00
Region 5	Closed			Manifold_05...		50.0	34.11	88.64	<input type="checkbox"/>	0.69
Region 6	Closed			Manifold_05...		50.0	19.03	637.61		0.94

(Note: the number of cone supports may vary according to the values you have entered).

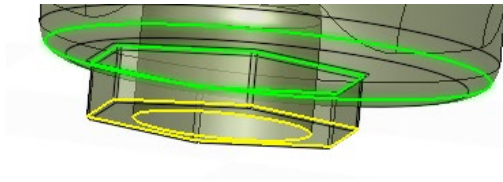
The Region is automatically hidden (bulb is off).

23. Hide the Cone supports by clicking the bulb in this Region's 'S' column.

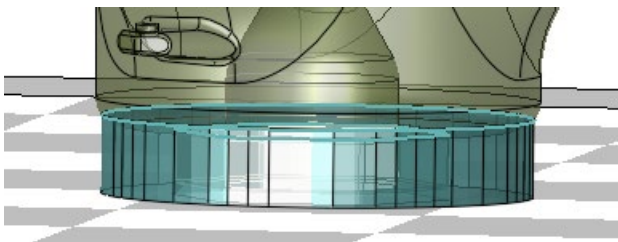




24. Pick the Region shown here in green and press the Solid Support button .



25. The Solid support is a simple Solid block, which is extruded down to the tray.



As you can see, this is a fairly large support (relatively, of course). We can divide the support into smaller supports to save material and time. This operation of dividing one large support into an array of smaller supports is called Fragmentation.

26. In the Solid Support Dialog, click the Fragmentation box.

**Solid Support** ✕

☒ **Fragmentation**

X Spacing:     Angle:

Y Spacing:     ☒ Same X, Y

Width:

Ignore fragments less than:  mm<sup>2</sup>

---

☐ **Skirt**

☐ External    Radius:

☐ External + Internal

Penetration / Gap:

**Tilting**

☒ Vertical

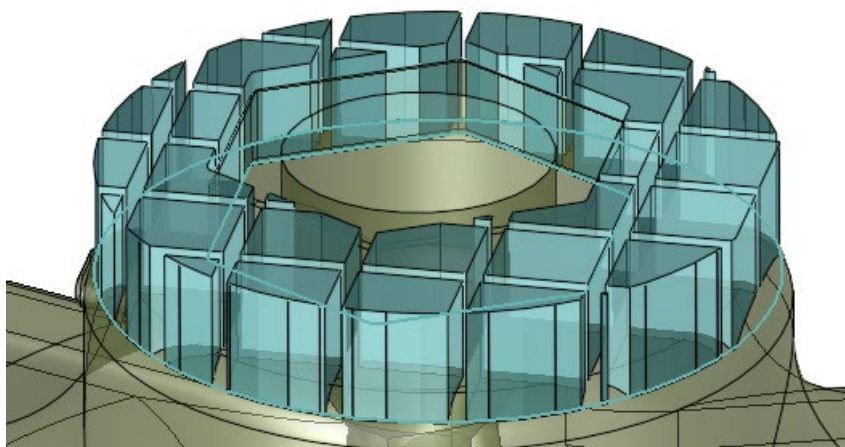
☐ Tilt & Scale

☐ Radial Tilting    Scale:

☐ Automatic Tilting

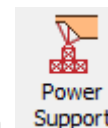
Fragmentation divides a large support (in this case a Solid support) into smaller portions. The Spacing values set the distance to the next fragmented piece on each direction, while the width sets the distance between the pieces.

27. Set for example, Spacing=7, Angle = 30, Width =1.

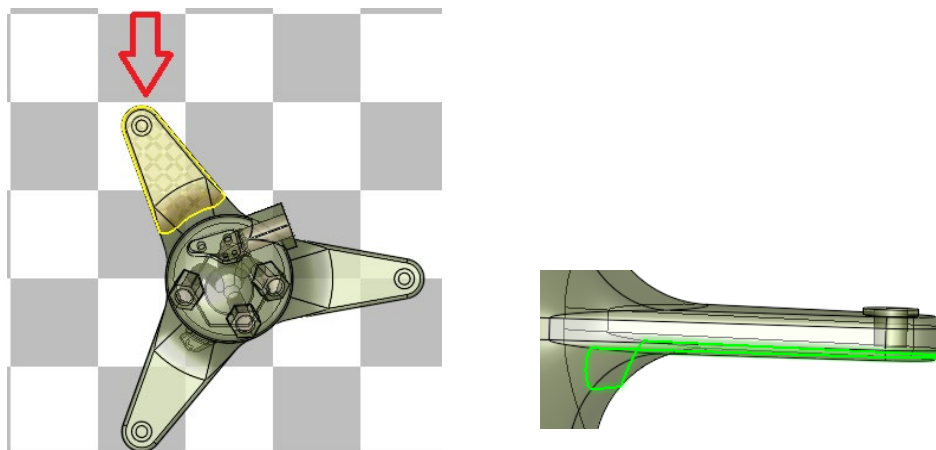


28. Click **OK** on the Solid Support Feature Guide.

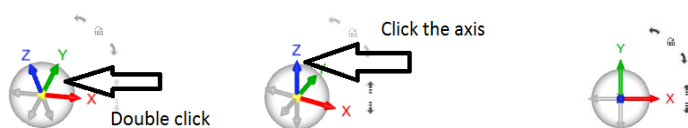
Region 6	Closed			Manifold_05...		50.0	19.03	637.61		0.94
Region 7	Closed			Solid		50.0	5.82	767.70	<input type="checkbox"/>	0.93
Region 8	Closed			Manifold_05...		50.0	19.03	637.61		0.97



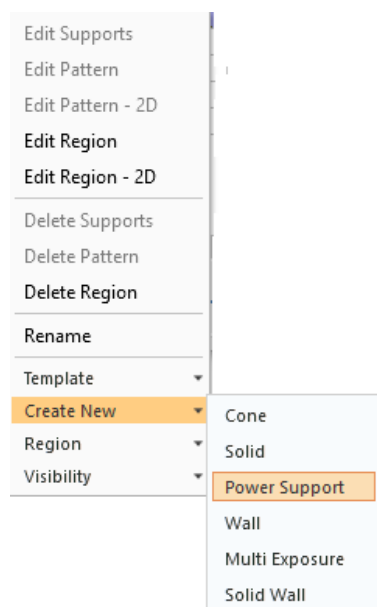
29. Select the region shown here and then press the Power Support button .



30. Select this region. To quickly find this orientation double click the bubble for ISO view and then click the Z axis for TOP view:



It is possible also to access the various Support functions by right mouse clicking the relevant Region in the Support Manager dialog and selecting from the sub menus:



The Power support may form lattice-like or tree-like structures.

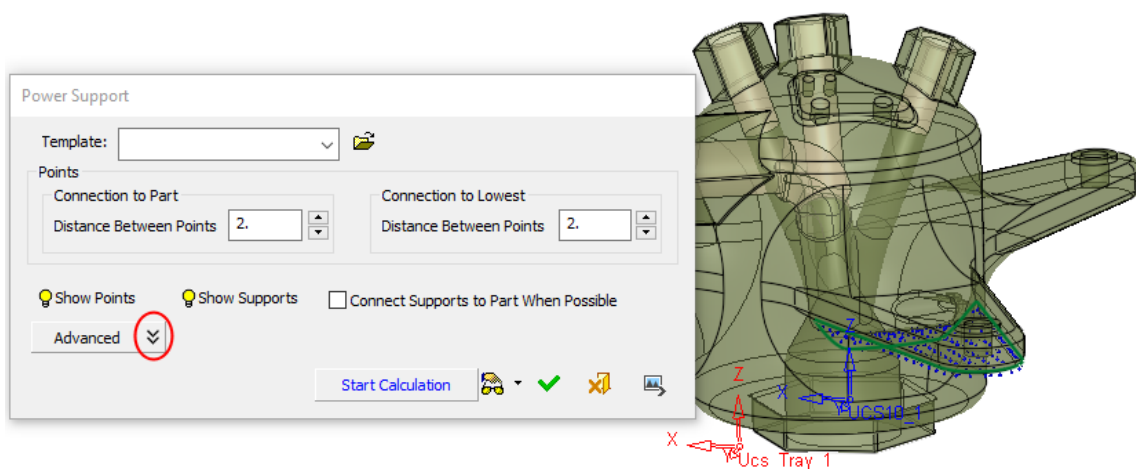
These supports are particularly useful for:

- Heat sinks (removing heat from the printed body) in materials that do not form great stresses (such as Aluminum) or parts that do not form great stresses (such as thin or small parts).
- The prevention of fusion overflow on overhanging areas (also known as dross).

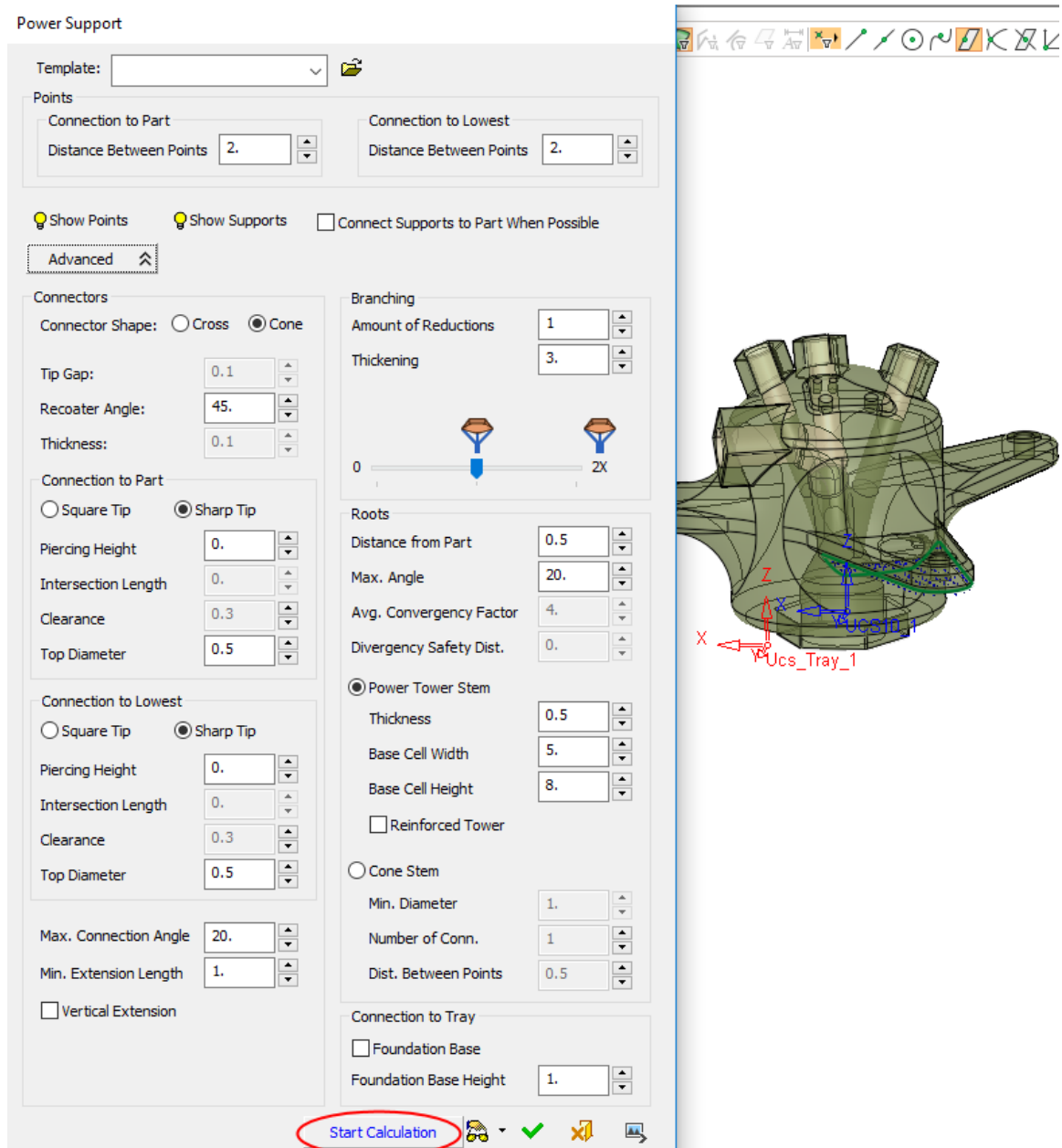
They can be used in other materials and parts alongside more sturdy supports that will manage stresses, while the Power Supports manage heat sink and eliminate dross formation

Note that there is a separate exercise dedicated to the Power Support functionality. In this exercise, simply accept the default parameters and click OK.

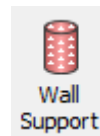
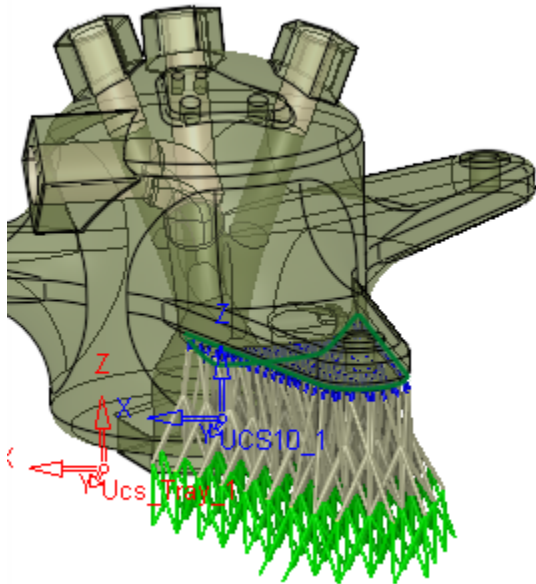
31. Expand the dialog.



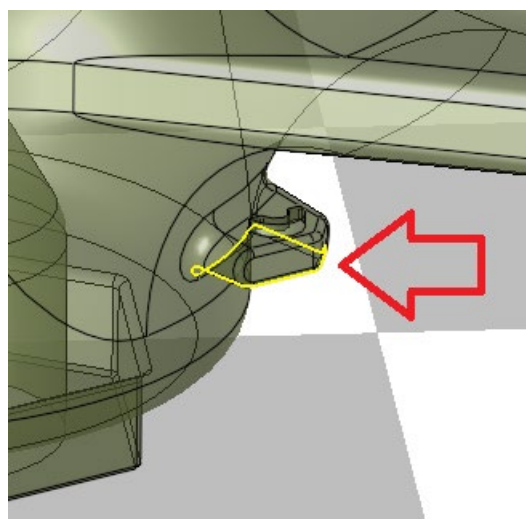
32. Press the Start Calculation button to preview the result.



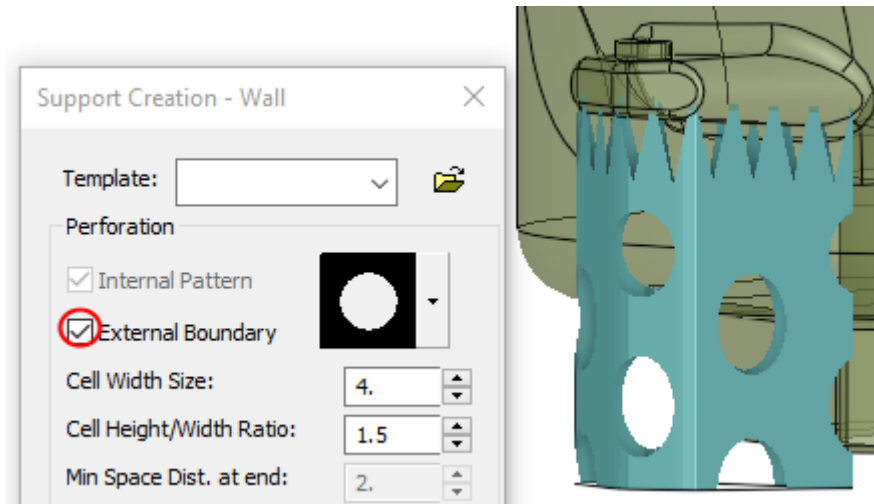
33. Select **OK**.



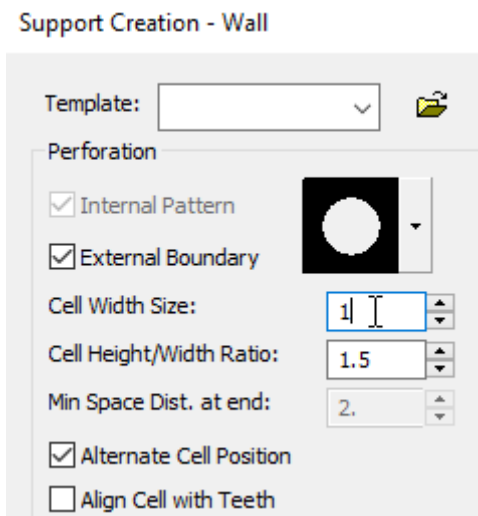
34. Pick the region shown in the image below and press the Wall Support button



Notice that the wall support runs along the boundary of the contour and reaches the tray. You can add a perforation to the wall. The default option is 'Perforation' – External Boundary is checked.

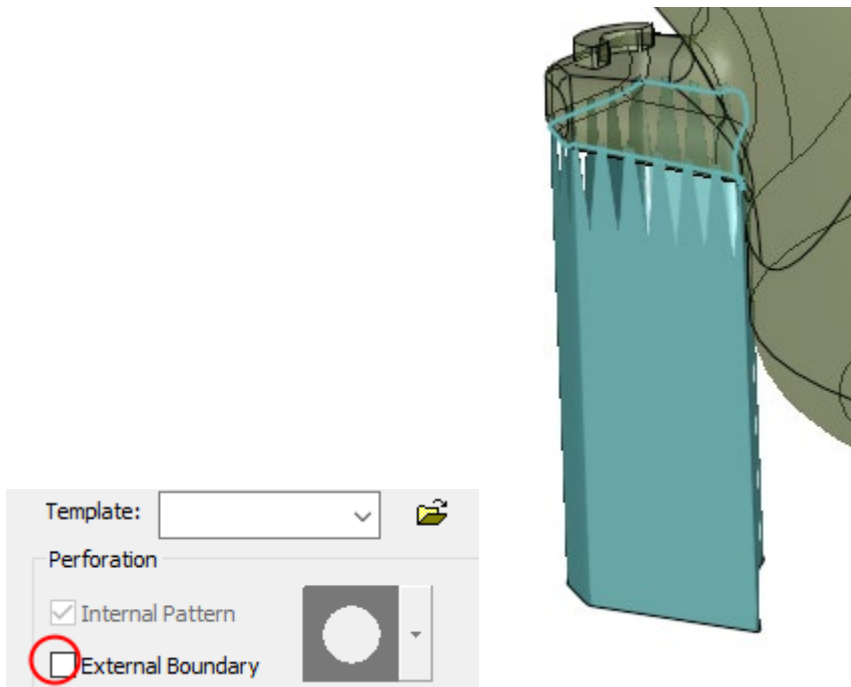


Set the Cell Width.



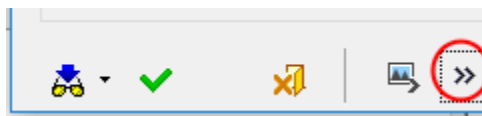
If you want to remove the perforation, uncheck the External Boundary option.





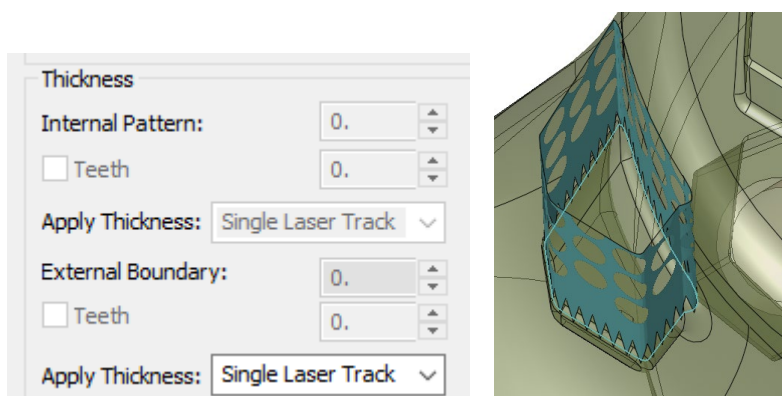
The Wall Support creation dialog includes numerous Parameters.

35. Press the **Display Advanced Setting** to view other parameters.



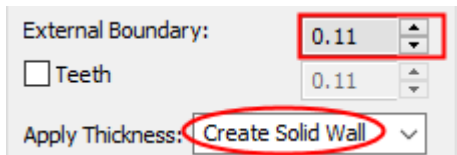
The 'Thickness' section of the dialog controls the width of the walls, as you can control the width of both the Internal Pattern and the External Boundary.

Right now, the options for Internal Pattern are not active, as the wall support we create only has an external boundary (we will discuss Patterns later on in this exercise).



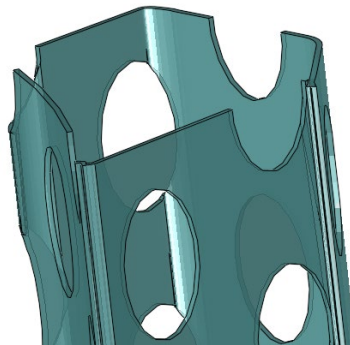
The default thickness for the external boundary is set as zero (Single Laser Track). This practically means that the width of the actual support will be dictated by the 'width' of a single laser beam.

36. Set the Apply Thickness option to **Create Solid Wall**. Accept the default minimum width **0.11**. .

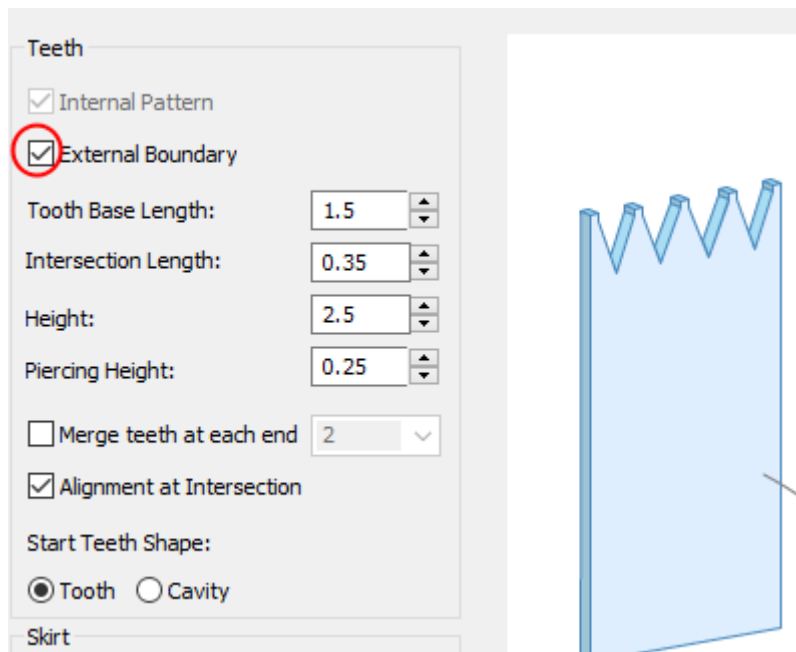


This option creates an actual 3D solid wall that will be sliced accordingly. In this case, the outer wall support may be technologically different from the inner.

When this option is selected, this means that an external boundary wall thickness value greater than zero has been defined. This value is set by the **Thickness > External Boundary** parameter.



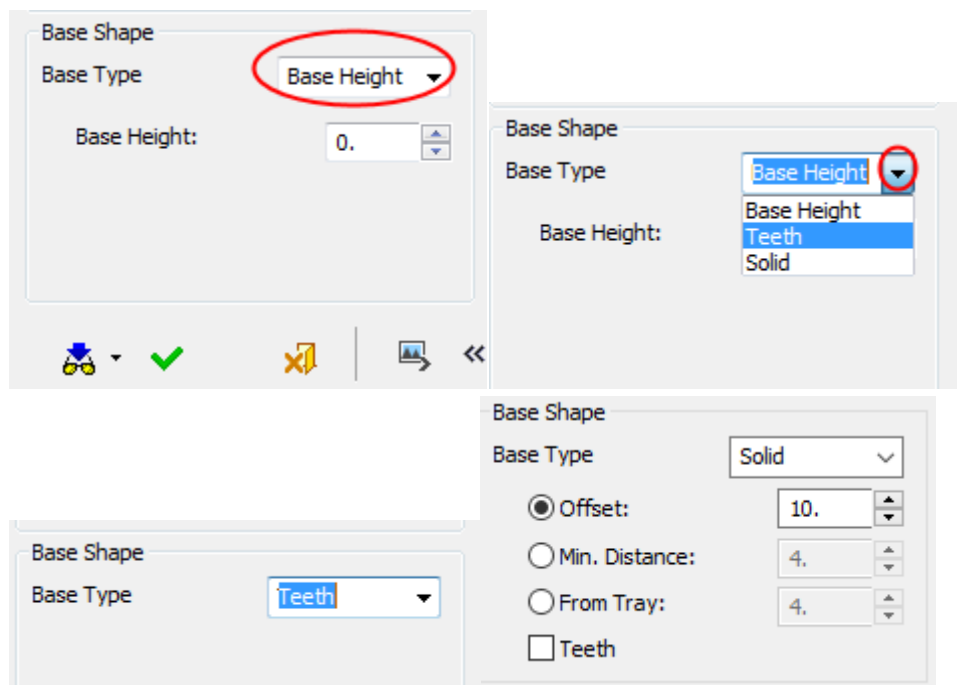
If the Teeth option is checked, then you can set the teeth thickness parameters.



Note that the teeth are thinner than the wall support. In this case, the Solid Wall option is used.

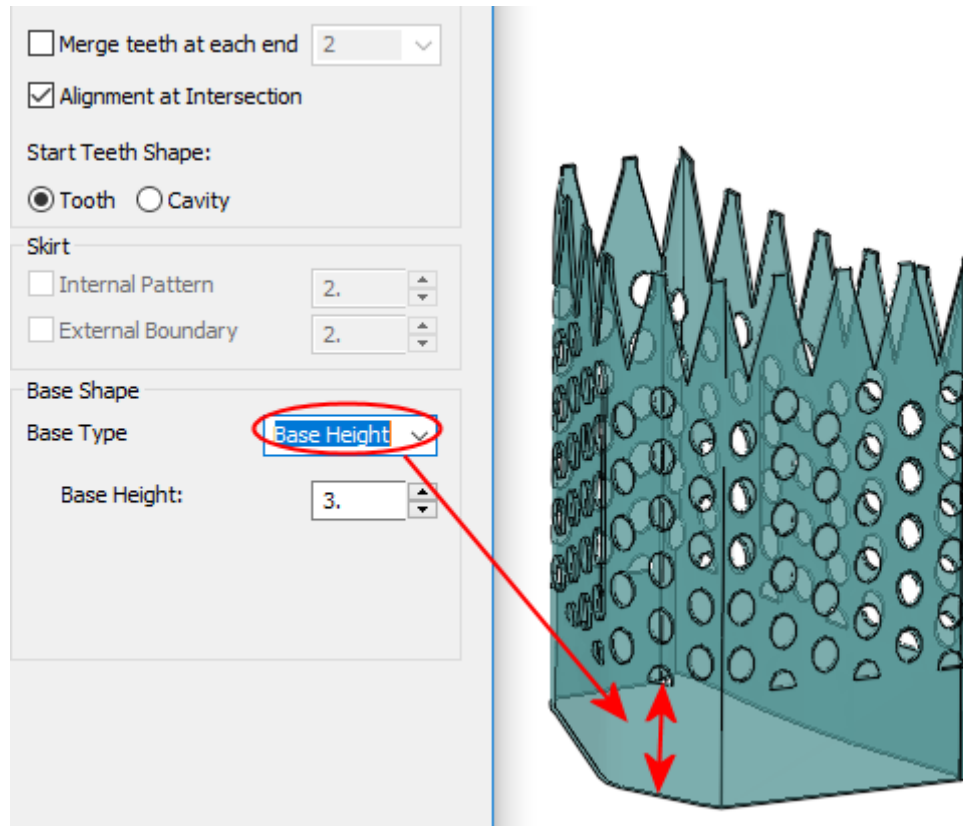
This makes removal of the support easier.

Other Wall Support features are the base shape and the teeth.

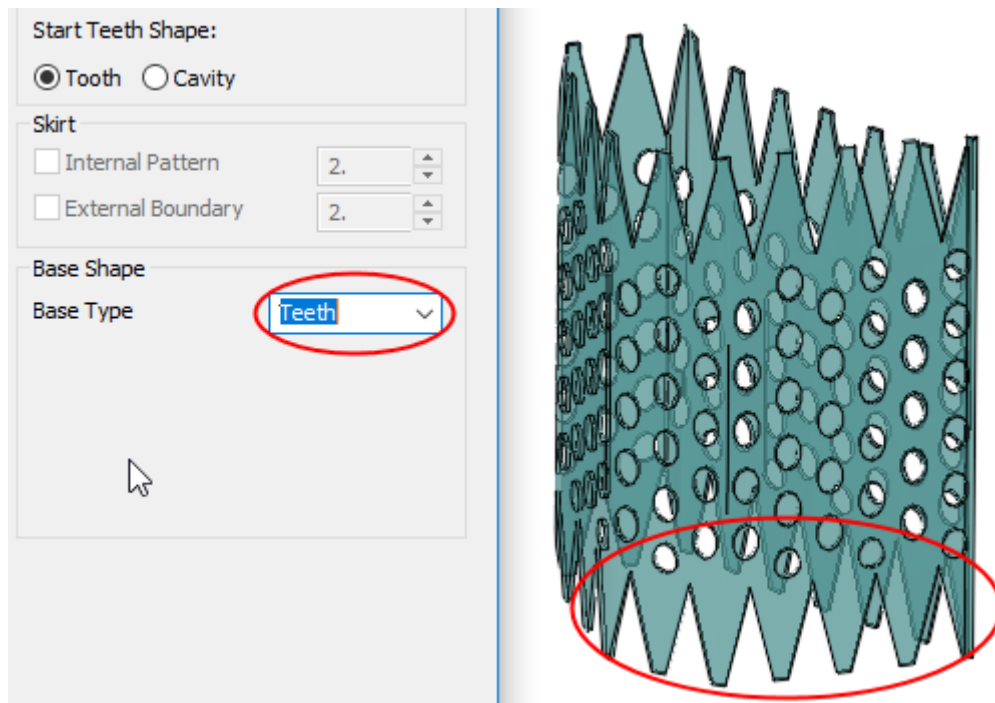


The following images describe the various options of Base Shape.

Base Height - start of texture height

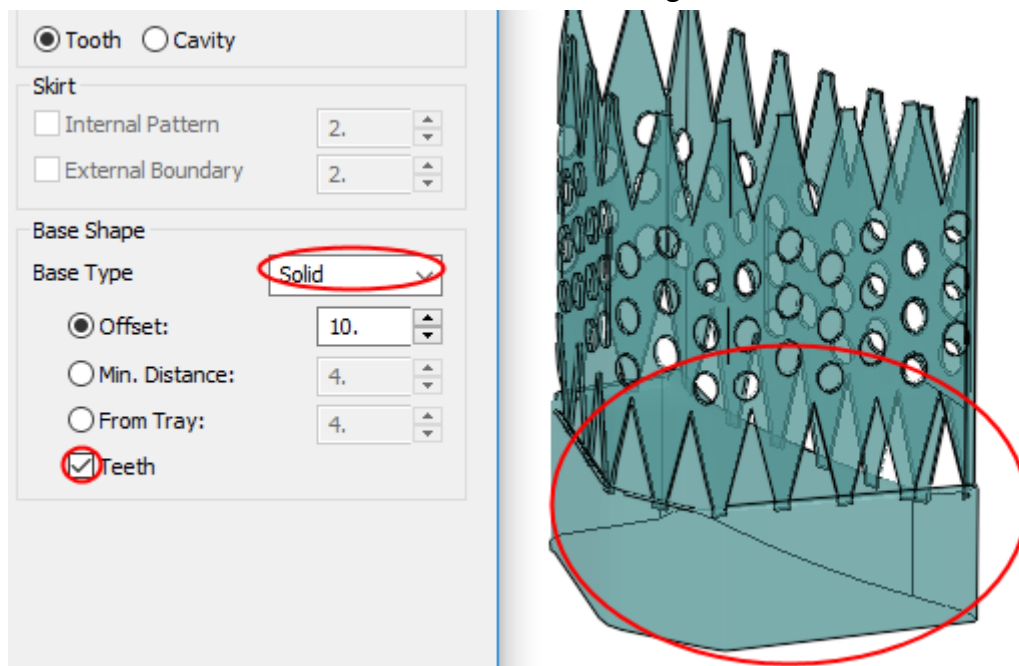


Teeth - add teeth also to base



Solid - add a solid support at the bottom.

This makes the wall more stable and durable during a roller movement.



Let's look now at the teeth creation parameters.

Teeth

☐ Internal Pattern  
☒ External Boundary

Tooth Base Length:

Intersection Length:

Height:

Piercing Height:

☐ Merge teeth at each end

☒ Alignment at Intersection

Start Teeth Shape:  
☒ Tooth ☐ Cavity

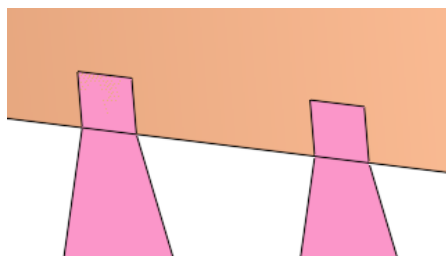
The teeth are trapezoidal teeth.

This enables control of the length of each tooth as it penetrates the part.

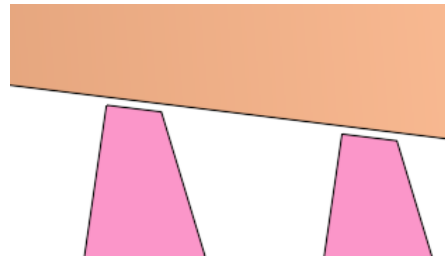
You can control the following teeth parameters:

- Base Length of the tooth (length where the tooth starts).
- Intersection Length (length of the tooth section as it penetrates the part).
- Height – Teeth Height (up to the tooth's virtual sharp vertex).
- Piercing height (the maximum height of the tooth inside the part).

The Piercing Height is practically an offset, so that the tooth will either penetrate deeper into the part (positive offset) or end outside the part (negative offset).



Positive Offset



Negative Offset

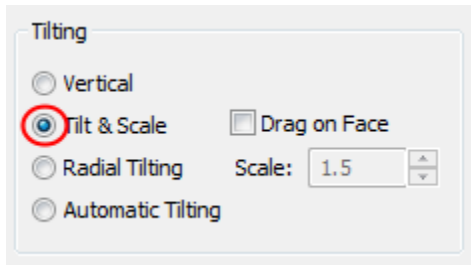
This region is a closed region. It is composed of a closed contour.

In open regions, you can also merge several teeth at the edges of the contour, to add strength (edges are more susceptible to stress).

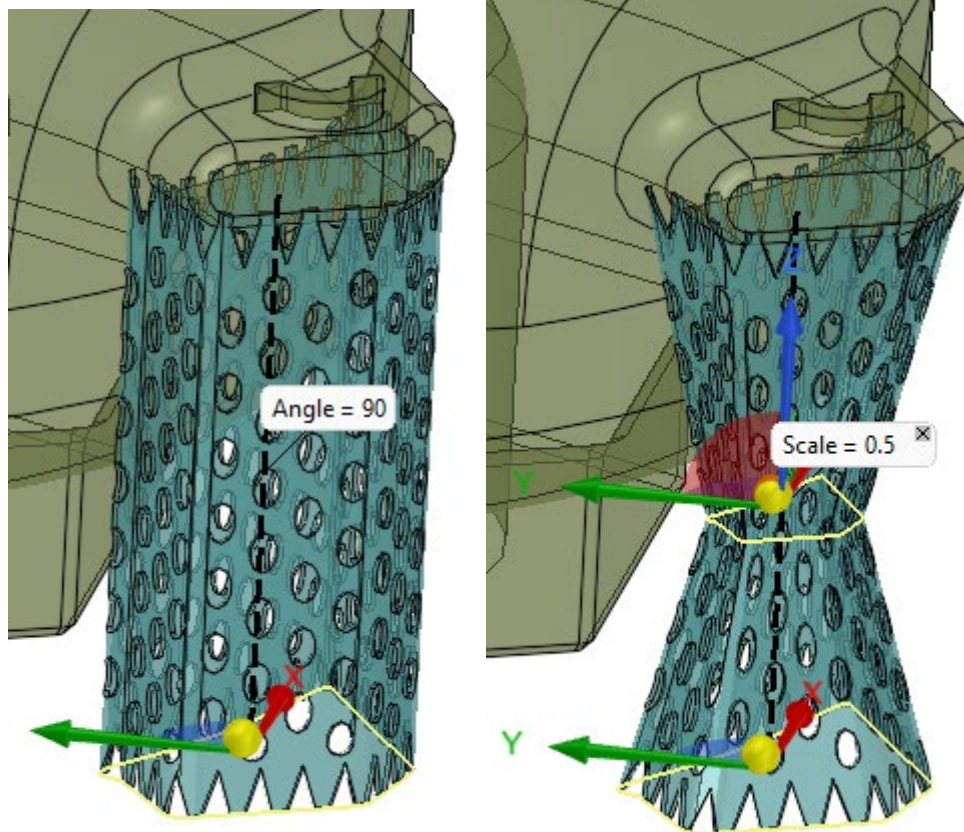
While we are in Wall Support, let's examine another option, Tilt & Scale.

In Wall Support, Solid Wall Support and Solid Support, it is possible to apply scaling also on non-tilted supports.

37. To add scaling to any point along the support height, click Tilt & Scale.



38. Pick anywhere on the support's reference line to add a break point, **right mouse click** and set the **scale** factor in the scale value box.



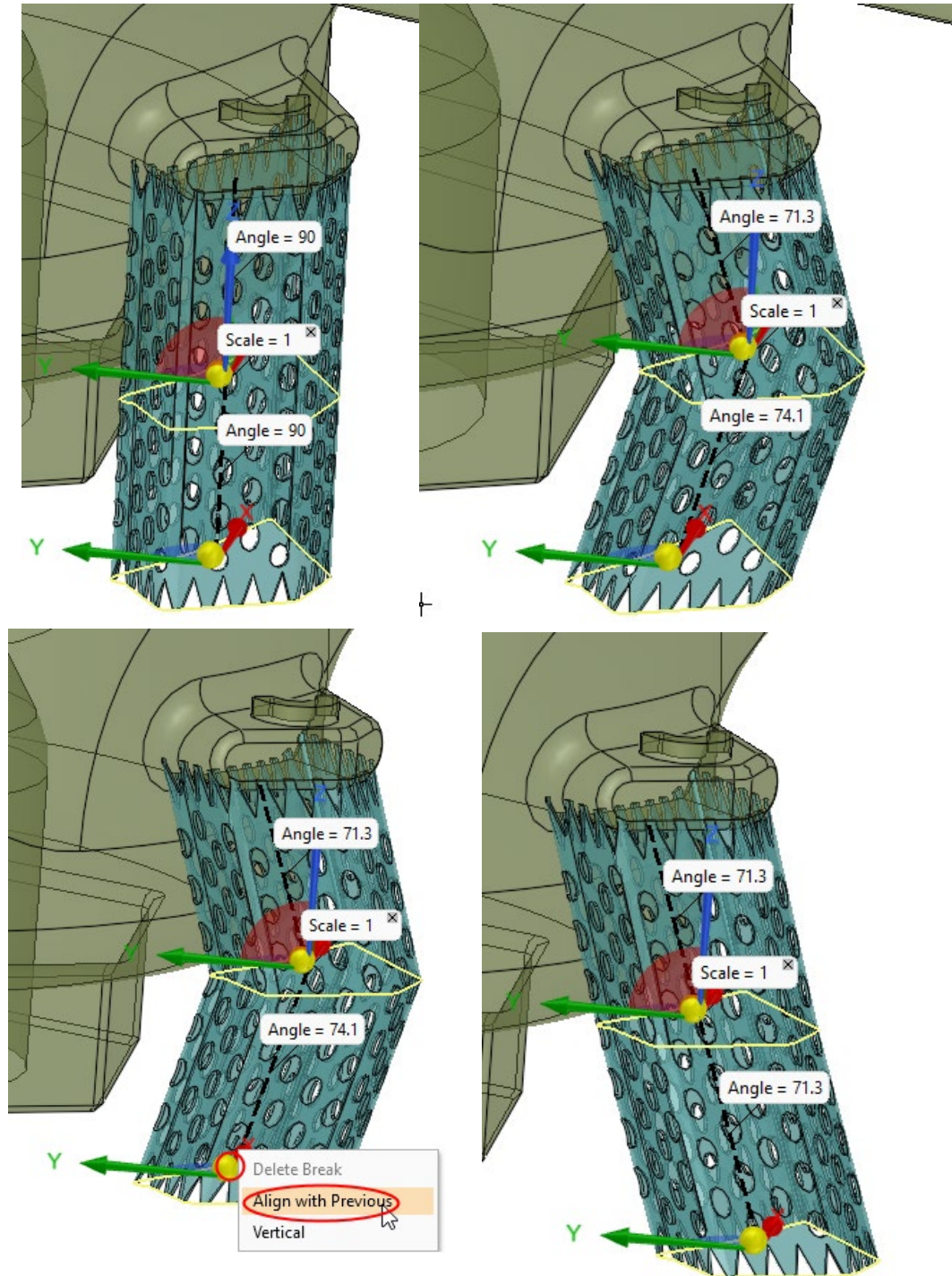
Let us review an additional option in Wall Support.

39. Set the scale factor back to **1.0**.



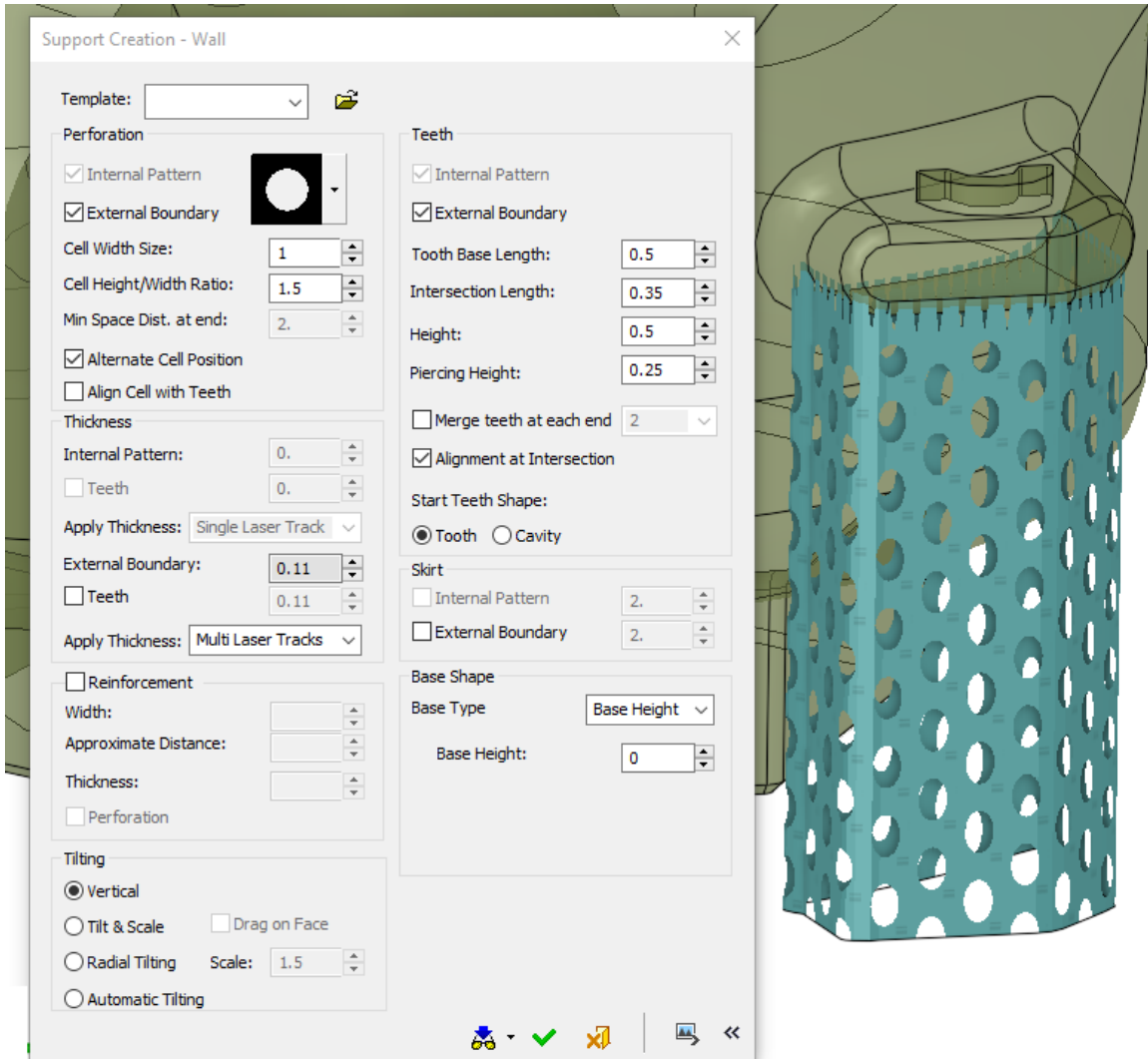
40. Now tilt the support by dragging the break point to any direction. Right mouse click the last (i.e., lowest) point (only) of the support and notice the options Align with Previous and Vertical.

Align with Previous will make the last component of the line to be component-linear with the line that came before last, and Vertical means that the last line is pointing straight downwards (from the previous point).



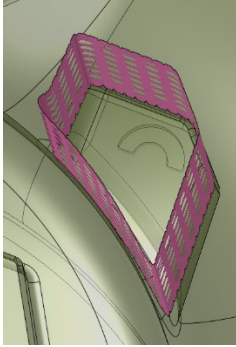


41. Set the parameters as shown below:



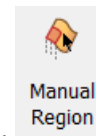
42. Set the option back to Vertical. Press **OK** to create the Wall Support.

As you can see in the following image, we have produced a wall along the boundary of the region.



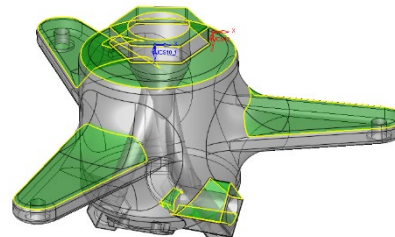
This will not be enough, as the area in the middle requires support too. We will therefore need to modify the contour on which the wall support is built. There are various ways to do this. For example, let's add a manual wall.

43. Click **OK** on the Feature Guide to exit the Wall Support function.

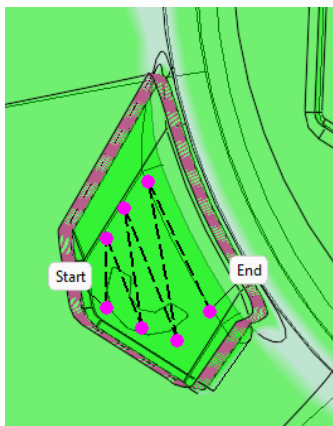


44. Press the Manual Region button from the Support Manager.

Once entering Manual Region, the areas which are within the minimum overhang angle are colored green. The areas are updated as you edit the overhang angle.




45. Rotate the part so that you are looking at the region from the bottom and using the default **Line** option, draw an inner contour, like this:

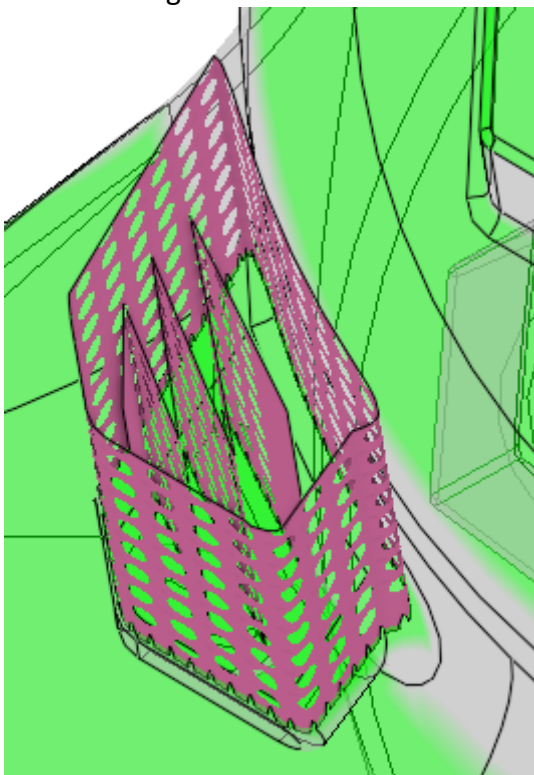


46. When done press **OK** on the Feature Guide.

The Contour is added to the table as Manual Region 10.

Name /	Part Name	Region Type	C	S
Manual Region 10	Manifold_05#1	Open		

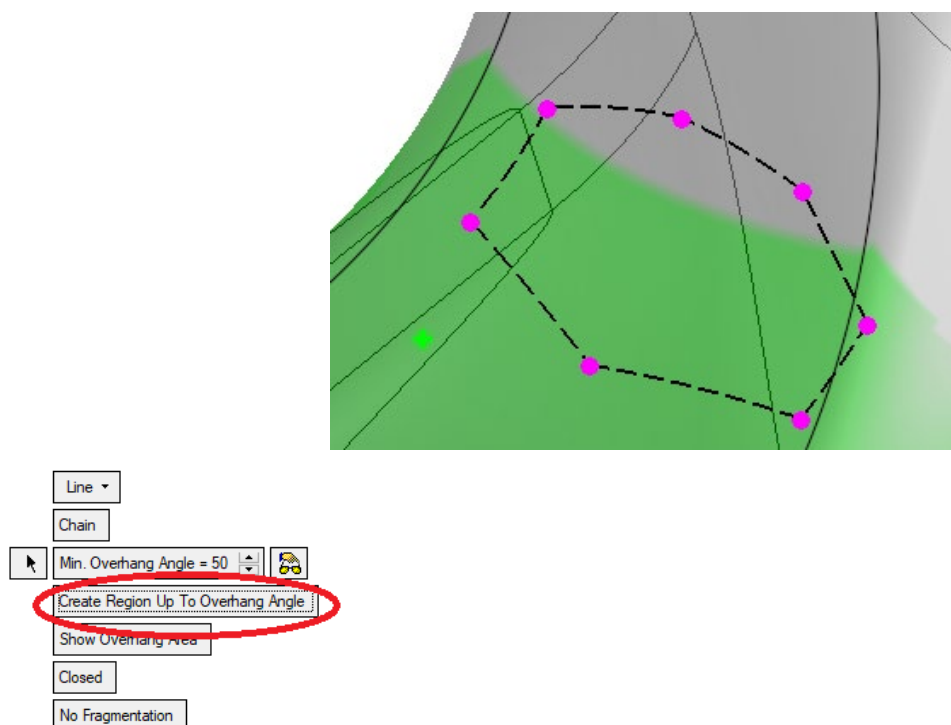
Select this region and create the Wall Support. Select **OK**. Now the supports also cover this region.



Additional notes on Manual Regions:

Manual Regions enables you to either draw the contour or use an existing 2D wire to define your region.

If you draw the lines, you can pick any purple point and drag it or add breakpoints by picking anywhere on a dashed line and dragging it.



Create Region Up to Overhang Angle:

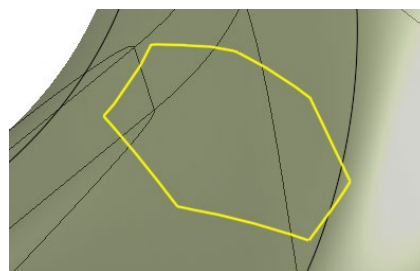
The system only considers the contours within the 'allowed' (green) area, based on the overhang angle.

In the image above, notice that a portion of this contour is lying outside the green area.

Create Region – result:

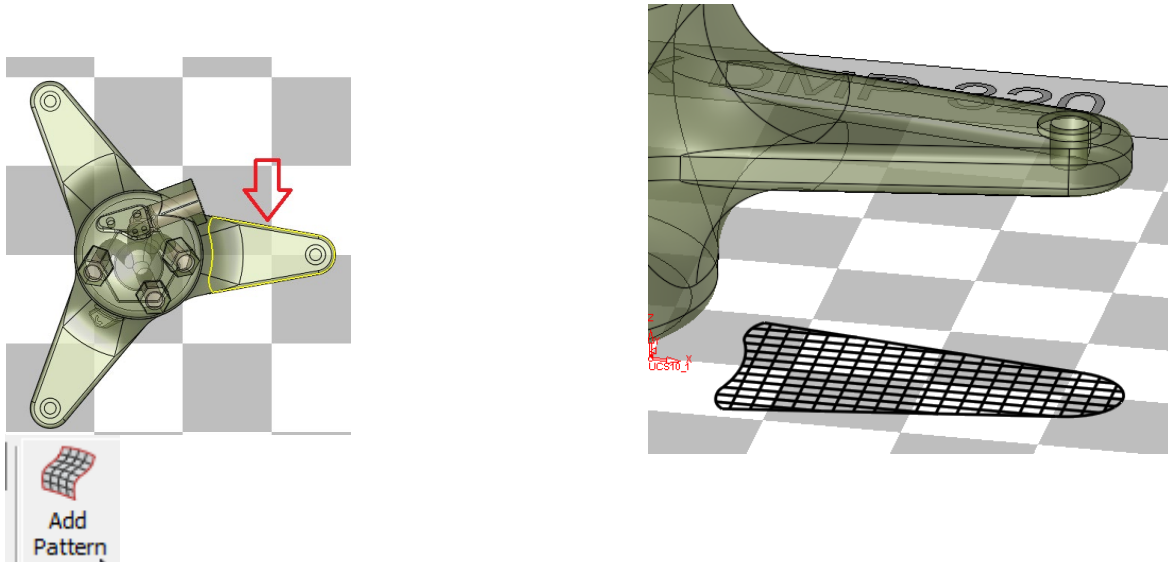


Up to Overhang Angle



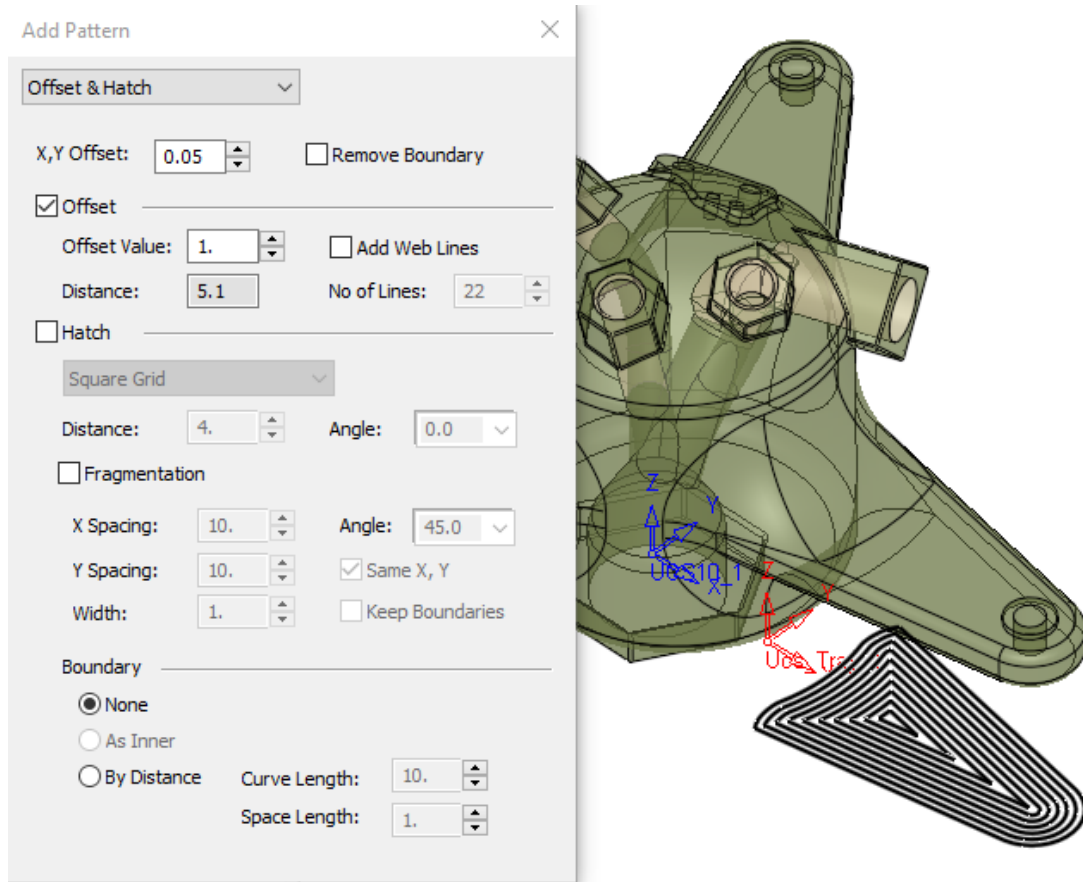
On Full Range

47. We can, however, create regions in a more efficient way. Select the region showing here and click Add Pattern.



A 2D pattern is projected from the region's boundary onto the tray. This pattern can be modified as required. The pattern shown above is a Hatch type.

48. From the dialog box, switch off the Hatch on check on the Offset type.

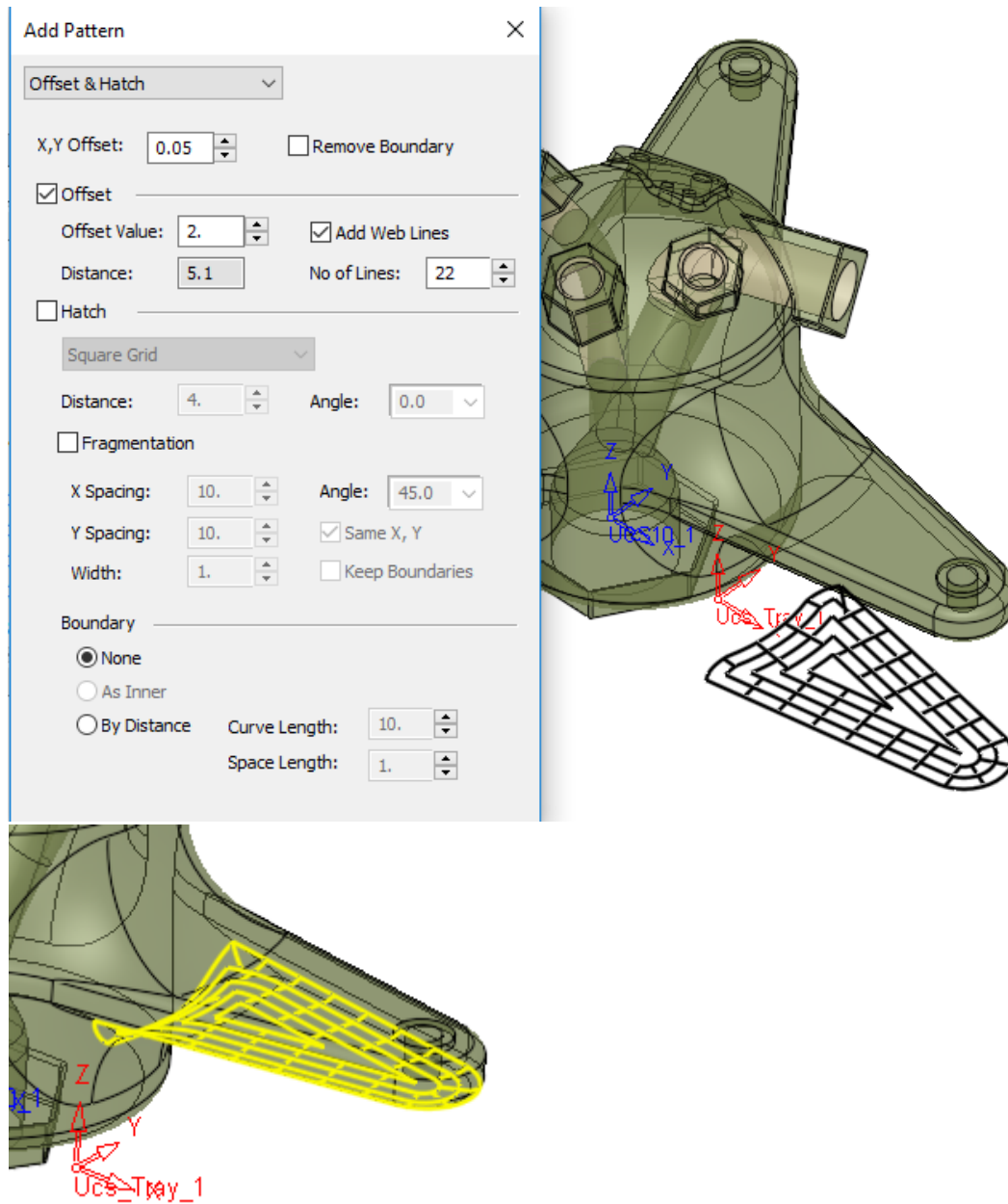


49. Click the **Add Web Lines** box and set the size of the offset.

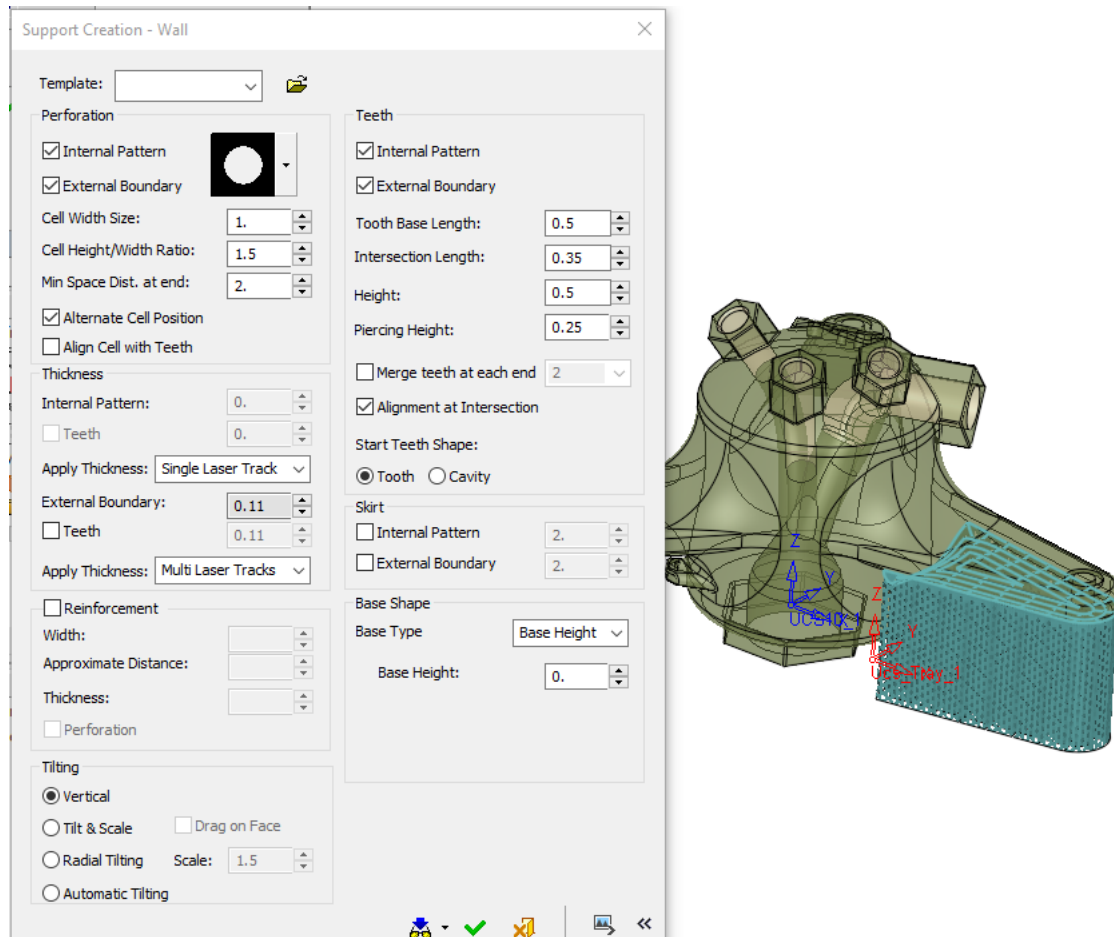
Set **Offset = 2**, **No of Web Lines = 22**. Note that there is some area left in the middle. It is possible to include both Offset and Hatch types in the same pattern. Yet for this exercise, let's keep only the Offset option.

Note: Although we are not using this, **Fragmentation** is recommended here. Such supports are removed by hand, so dividing the walls means easier separation of the support from the model.

50. Click **OK** on the Feature Guide and see that the pattern you have drawn is projected back to the part.

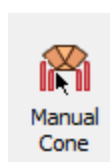


51. Now let's create a Wall support on it (select the region and click Wall Support) and see that a wall is created on every line of the pattern. Select **OK** to approve the support.

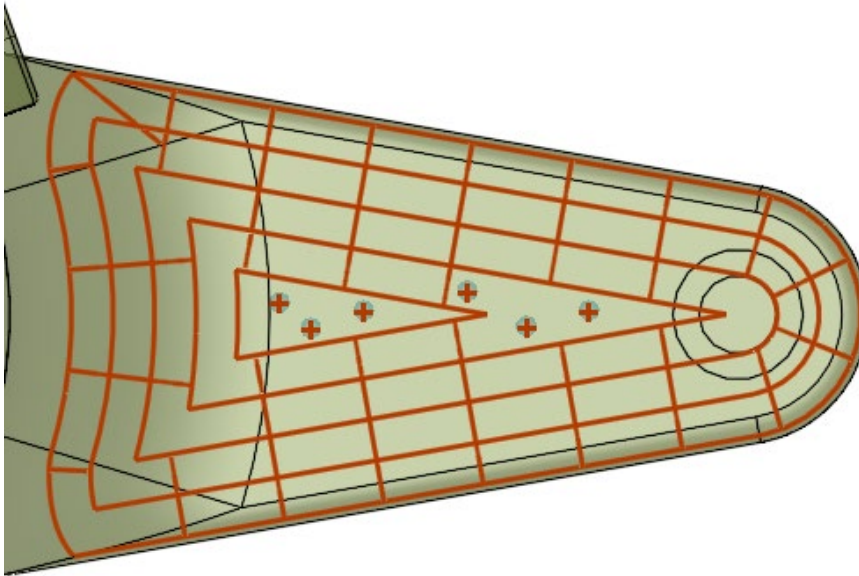


We have left some area in the middle of the region on purpose, as we want to add some cone support over there.

52. So to add these, press **Manual Cone** and add the cones by clicking on the geometry.







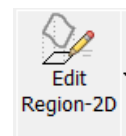
So now we have a combination of wall supports and cone supports in the same region.


Optional Step –Edit the region and create an Island:

53. Click the Wall Support just created.

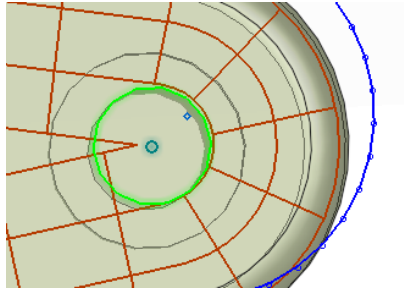
This marks the relevant row in the table.

54. Right mouse click the region and press **Edit Region – 2D**

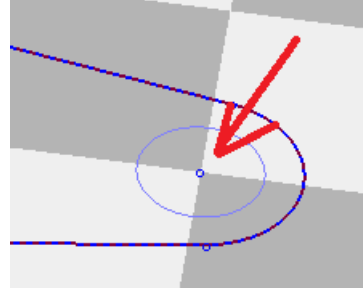


55. As the Sketcher opens, press the Circle function  and draw a circle around the hole's center point.

56. Add references in the Sketcher – press **Add Reference** , hover over the part and pick the hole's center.



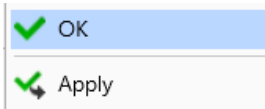
Pick reference



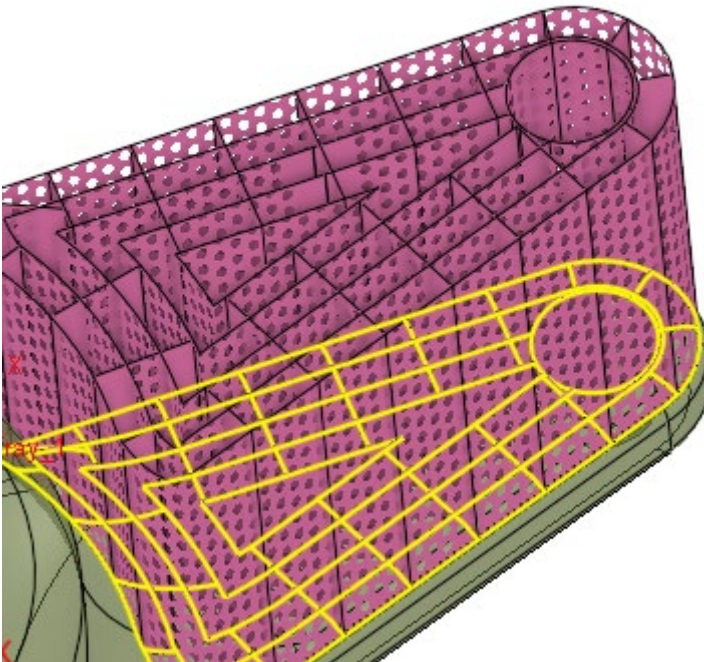
Create a circle around the center point

57. Click the left mouse button to approve the circle. The diameter of the circle is not important for this example.

58. Right mouse click and press **OK**.

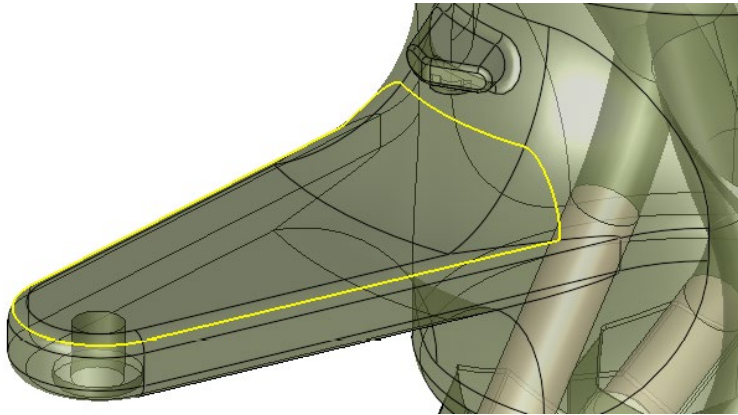


What we have created is an island – the supports are updated according to the new boundary.



59. Let's see some more Pattern creation options.

Select the region of the third 'wing':



60. Click again the **Add Pattern** button. This time set the pattern back to Hatch (check **OFF the Offset**) and set it to type of **Honeycomb** (choose from the list of available hatch types).

61. Set Fragmentation **ON** and click the **Keep Boundaries** box (marked by the red box).

Offset & Hatch

X,Y Offset: 0.05

Remove Boundary

Offset

Offset Value: 2.

Add Web Lines

Distance: 5.1

No of Lines: 22

Hatch

Honeycomb

Distance: 2

Angle: 0.0

Fragmentation

X Spacing: 10.

Angle: 45.0

Y Spacing: 10.

Same X, Y

Width: 1.

Keep Boundaries

Boundary

None

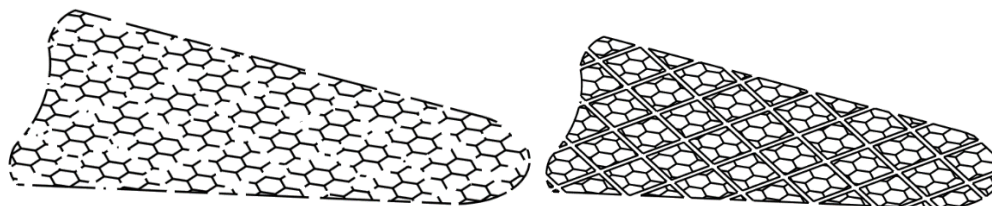
As Inner

By Distance

Curve Length: 10.

Space Length: 1.

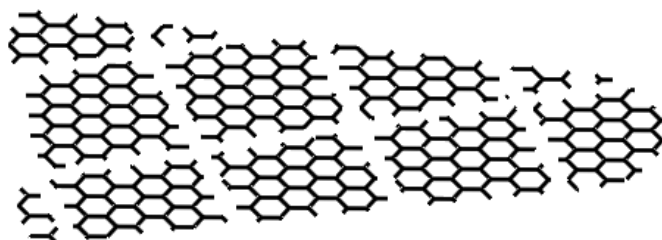
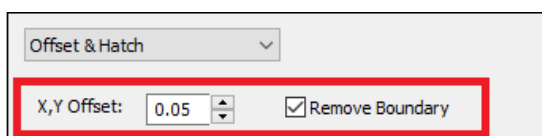
This adds the contour along the fragmentation boundary lines.  
This can be used as a visual aid. In practice, these boundaries are not needed.



Keep Boundaries – OFF

Keep Boundaries - ON (Visual Aid)

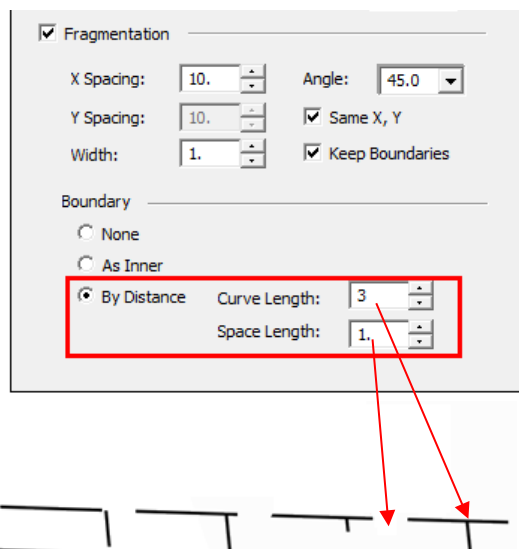
62. In addition, you can remove the outer boundary of the pattern by checking the Remove Boundary option and also reduce the size of the pattern (as a compensation for the laser beam size) by setting an offset.



Remove Boundary (outer)

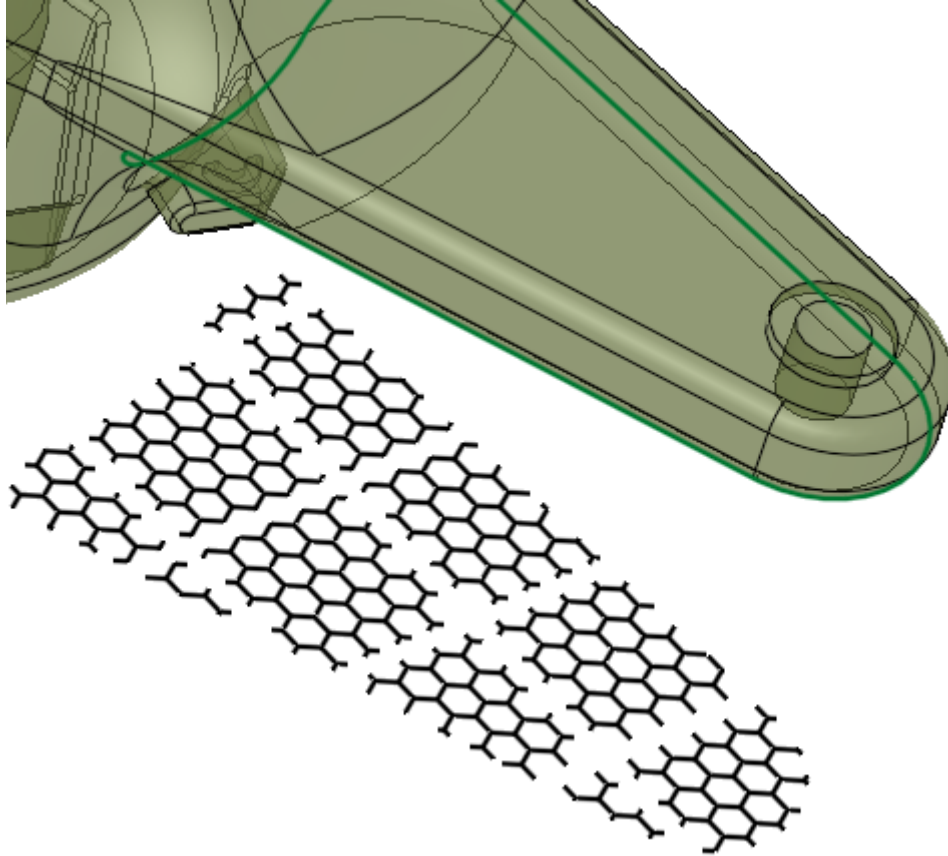
63. Uncheck **Remove Boundary**, to restore the boundary of the pattern.

64. See the options alongside Keep Boundary. Switch to '**Fragmentation by Distance**' and define fragmentation for the boundary by entering the length of the solid line and length of the space between the lines.

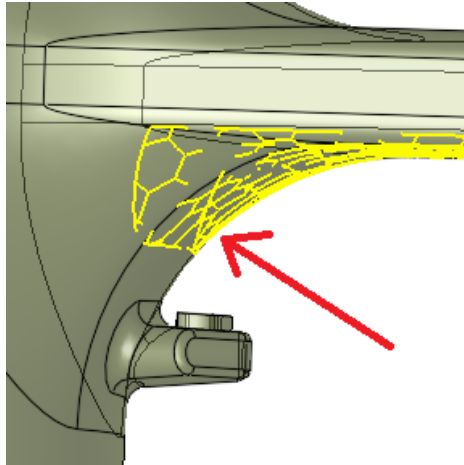


If the pattern also has a fragmentation, you can set the boundary to have the same fragmentation as the pattern in it (As Inner).  
As a default, a 0.05 mm inwards offset is used. Since wall supports have thickness, this default value makes sure that these supports are created within the region.

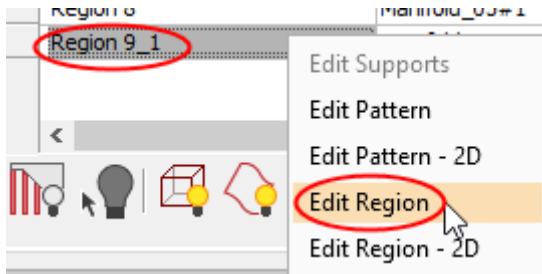
65. Uncheck the Keep Boundaries option and click OK on the Feature Guide and see that the pattern you have drawn is projected back to the part.



66. Before creating the support, let's suppose we would like to remove this portion of the region:

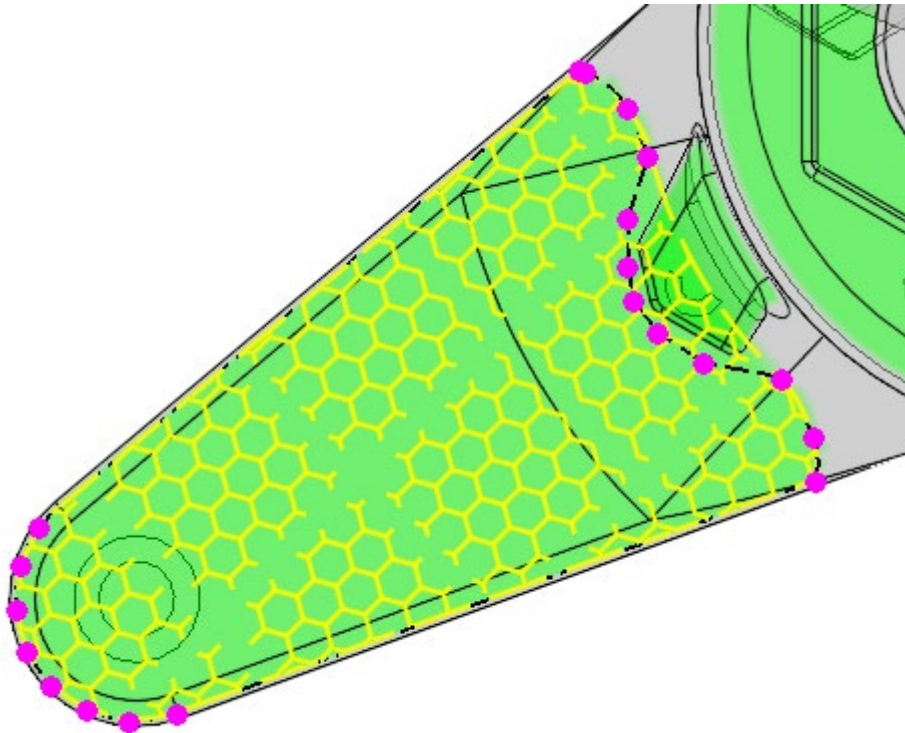


67. Right mouse click the same region through the Support Manager again and select Edit Region.

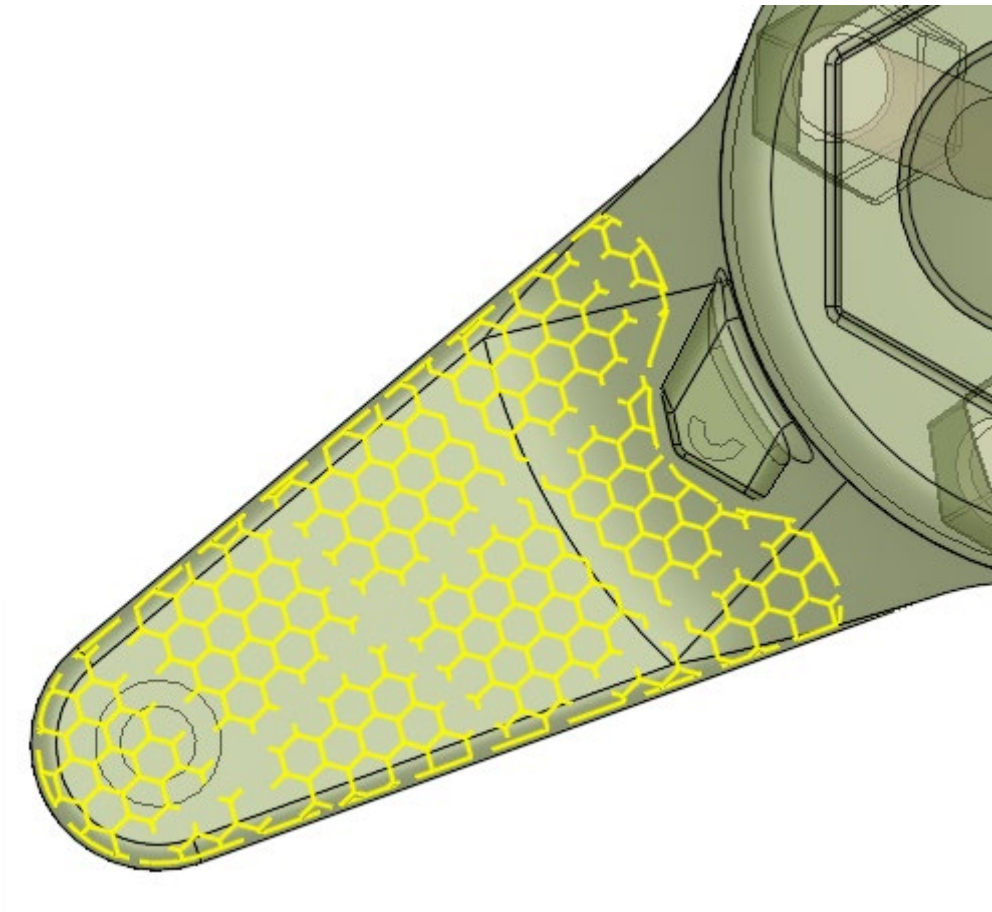


While 'Edit Region – 2D' uses the Sketcher tool, 'Edit Region' mode enables the 3D editing of the Region, which works much like the Manual Region creation.

68. Drag the break point so that the outer boundary of the region will look like this:  
Select OK.



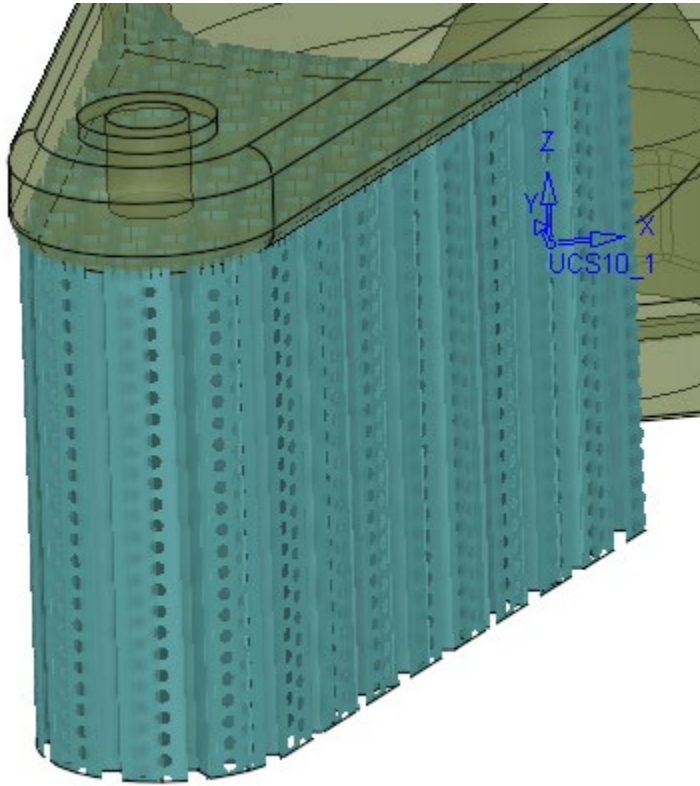




Now you can create a support on this region.

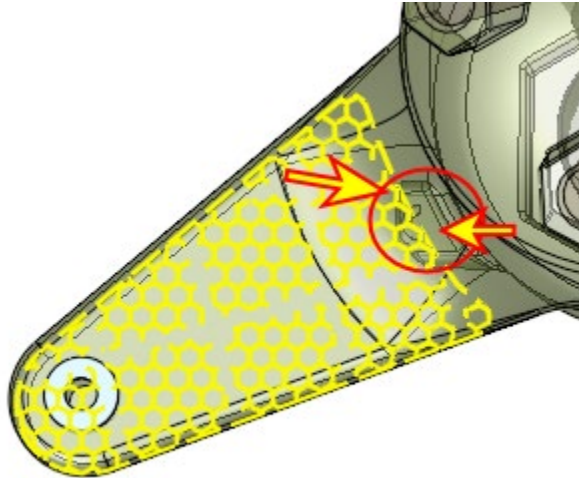
Note: It is probable that for actual printing, we may need some type of support for the area we have removed here. Yet, in this exercise we will ignore it.

69. Click the Wall Support button.

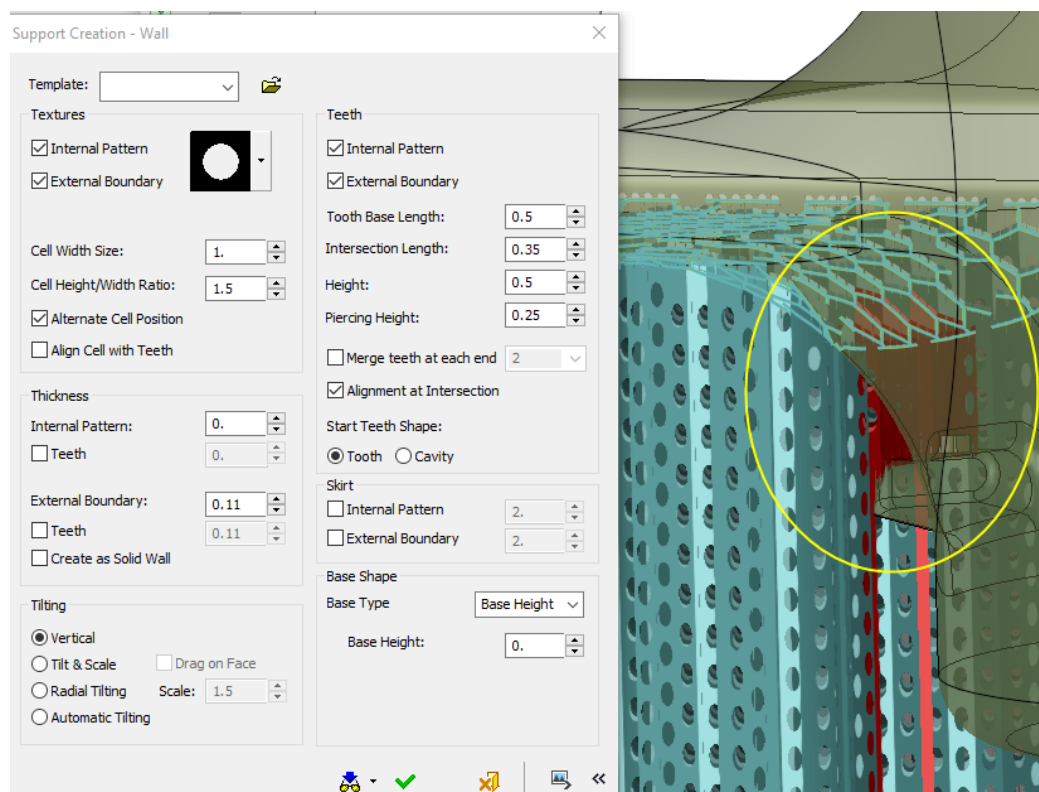


Note:

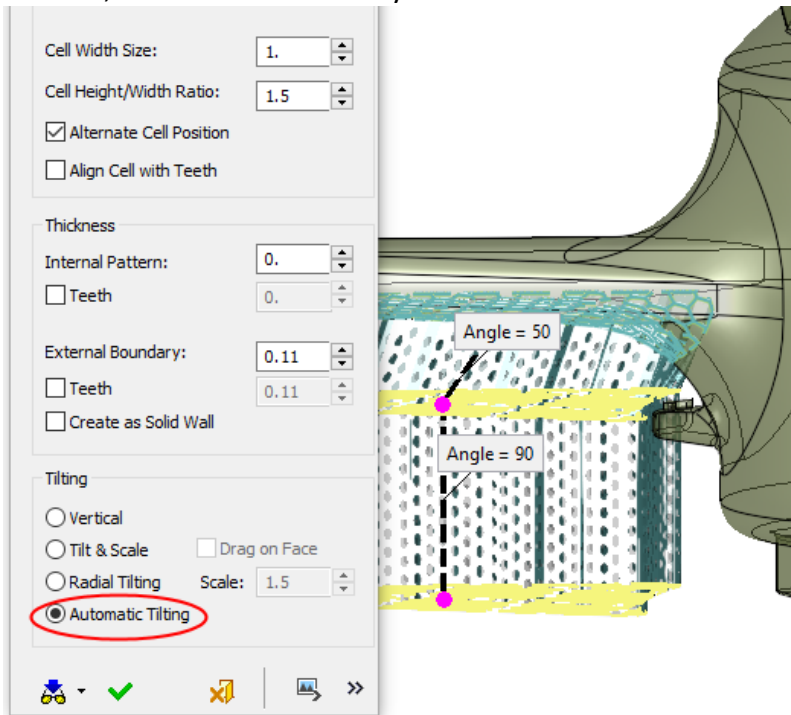
Another way to avoid interference between the support and the object is to apply 'Automatic Tilting'. In the following example, the pattern interferes with the object.



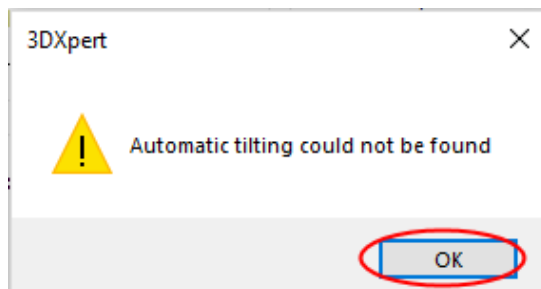
Therefore, a vertical support also interferes with the object.

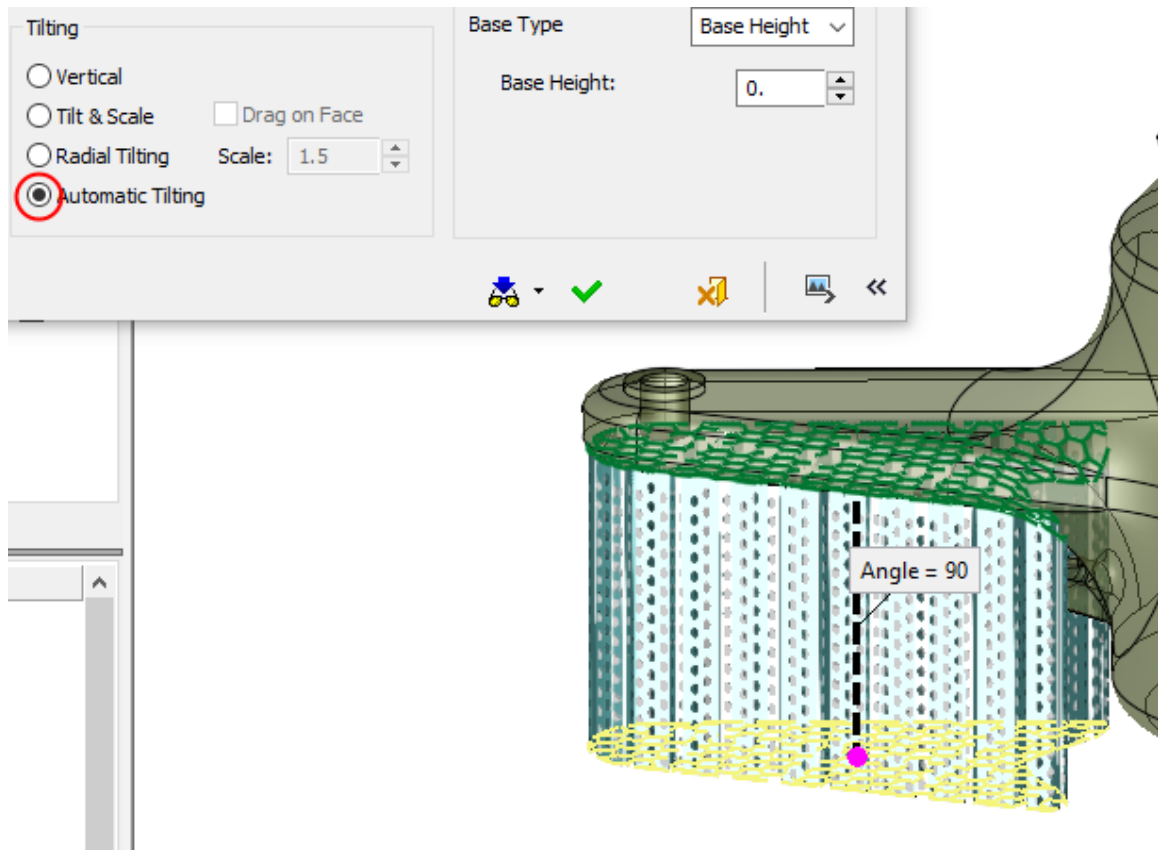


The Automatic Tilting option automatically tilts and orientates the support when it is close to, or intersects the body.



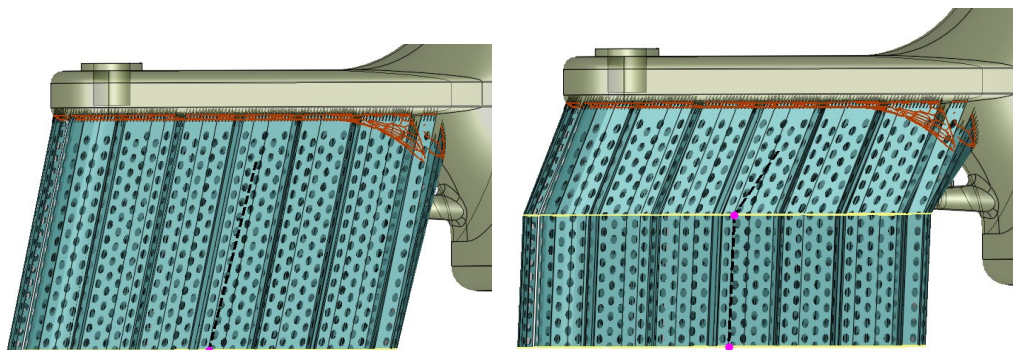
70. Set the '**Automatic Tilting**' option. In this case, the support does not move away as it does not interfere with any object. Press **OK** in the dialog.





Note that the guiding line is set in the middle of the support. You can still manually drag it, if you wish. If you do so, the system sets the option as 'Tilt & Scale'.

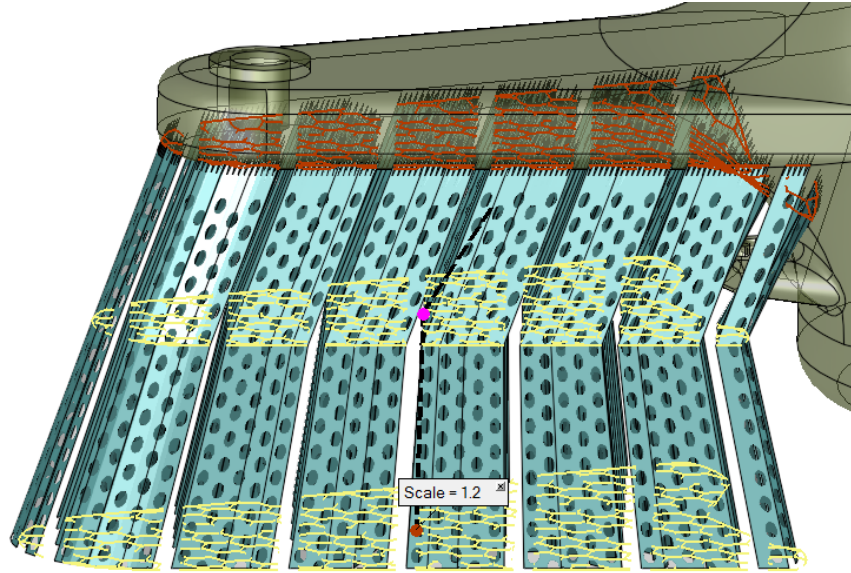
71. Now break the line around the middle.



Tilted line's end point

Add a break point

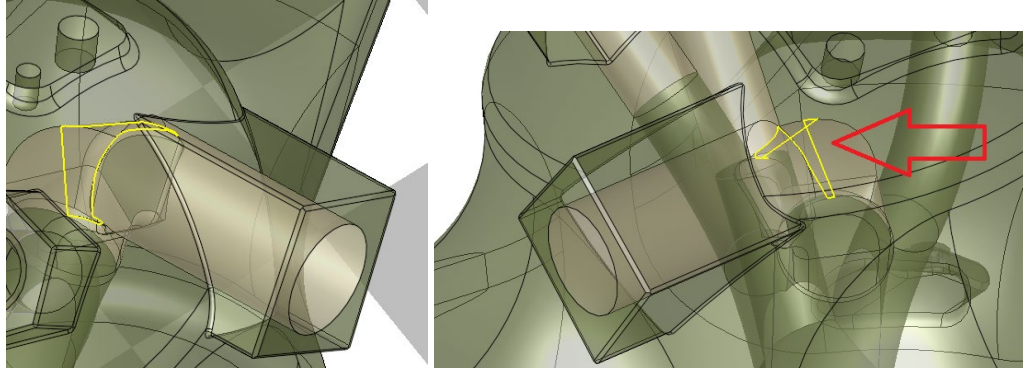
You can also add a scaling factor to each section by clicking the point and setting the scaling factor. This can create a skirt like wall support.



72. Press the **OK** button to approve the support creation.

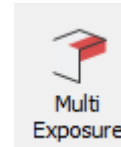


Let's look now at the region shown here - the inner area inside the tube.



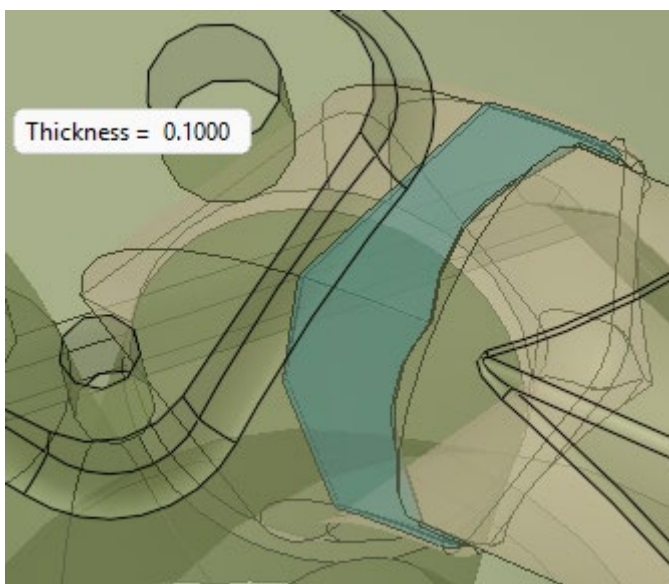
It is not possible to put a support in this tube, because we will not be able to remove it later on by machining. Therefore, we will define this region as a Multi Exposure area.

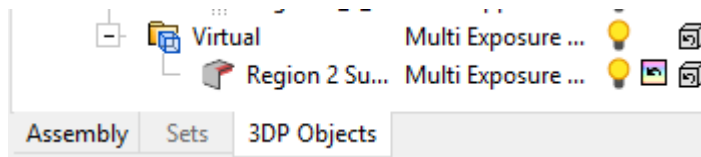
Once we define a region as Multi Exposure, a different printing technology will take place on that area. In short, the laser beam will pass each layer here several times (a weaker beam) so, in other words, the area will self-support itself.



73. Select the Region and click the Multi Exposure button. , set **thickness** to **0.1** and then press **OK** on the Feature Guide. By this, you have defined a volume, we call it a 'virtual object' and it defines the area that will be printed with that technology.

For more details on technologies, see later on in this exercise.

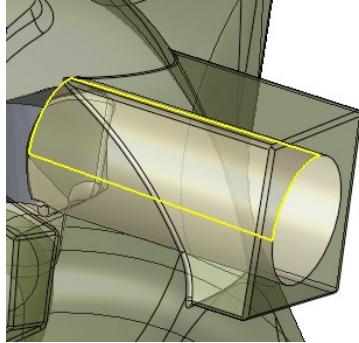




Result: A new object



Now take a look at the next region:



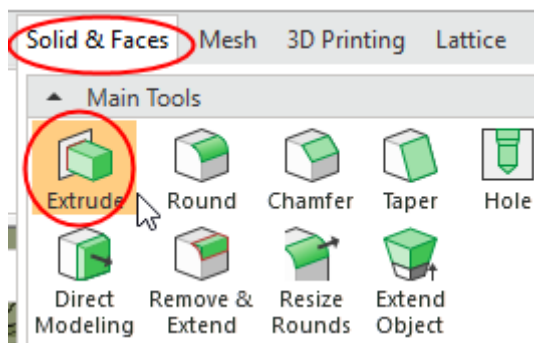
Actually, what we want to do here is close this hole by an object, since we want to drill this hole later on. We fill it with support material by creating an object to which we can assign any build style we wish. For this, we will use standard CAD solid tools. This is an advantage of working with B-Reps.

74. Close the Support Manager.

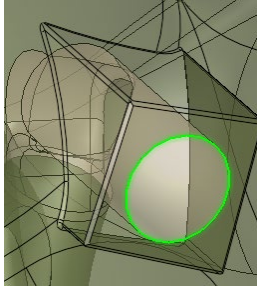
75. Activate the Manifold part by double clicking it on the project tree. Its name should be visible in bold letters.



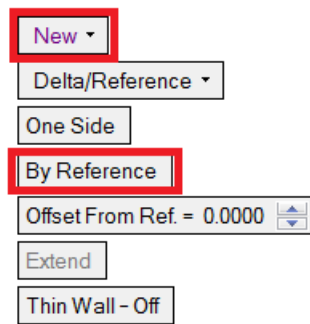
76. Press **Extrude** from the menu bar.



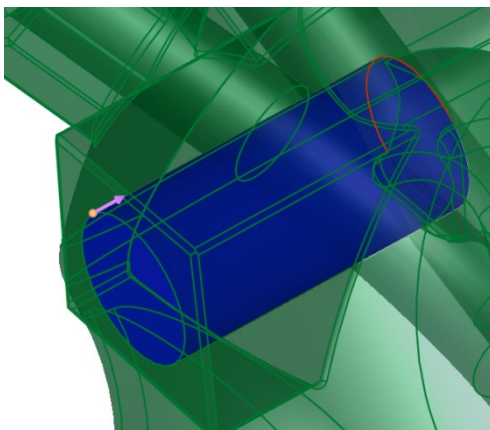
Pick the cylinder's outer contour.



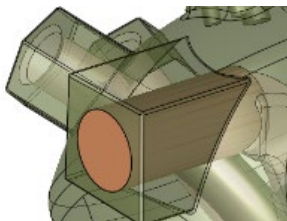
77. Switch from **Add** to **New**, **By Delta** to **By Reference**



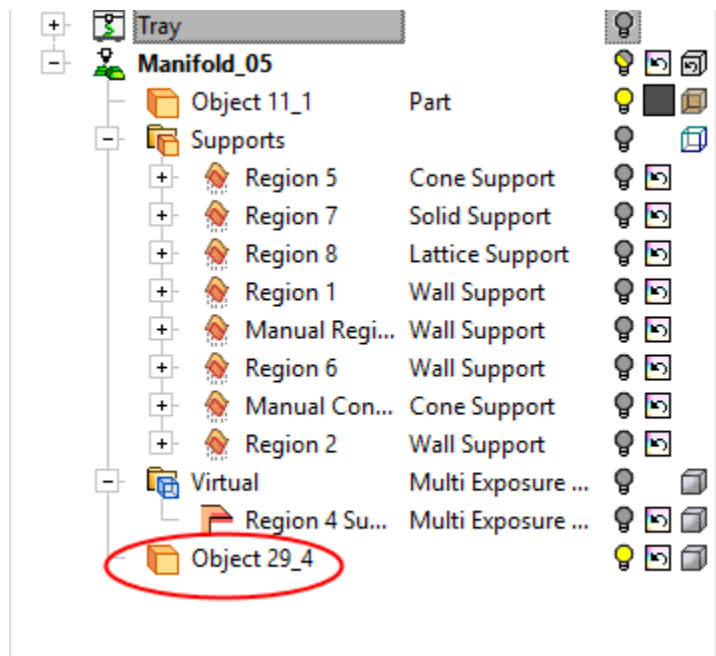
78. As the reference, pick the curve shown here in red:



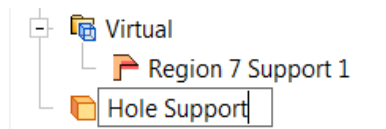
79. Press **OK**. The result is a new object. Later on, we will tell the system how this object is printed.



80. Click the 3DP Objects tab in the project tree.

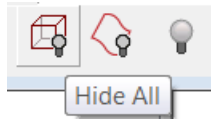


81. Right Mouse click the last Object in the list and **Rename** it as 'Hole Support'.

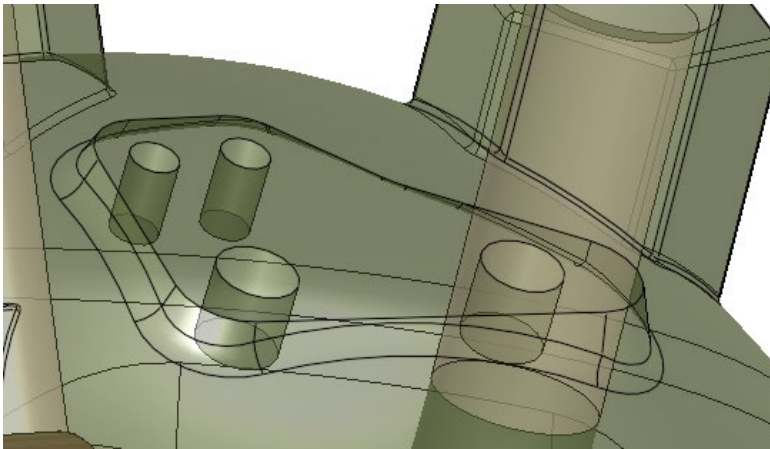


## Part 2– Prepare for Printing.

82. Hide all supports by pressing Hide All from the bottom of the Support Manager.

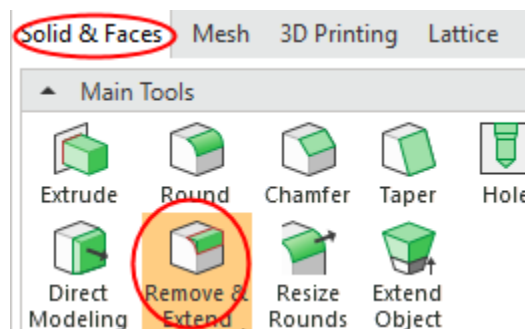


83. Look at the part and see that it has some holes on the top.

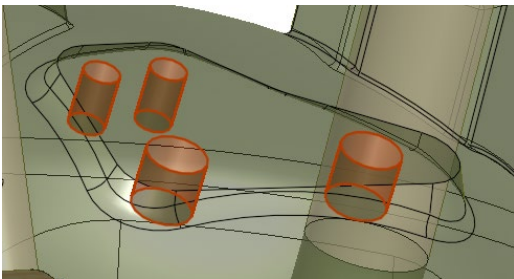
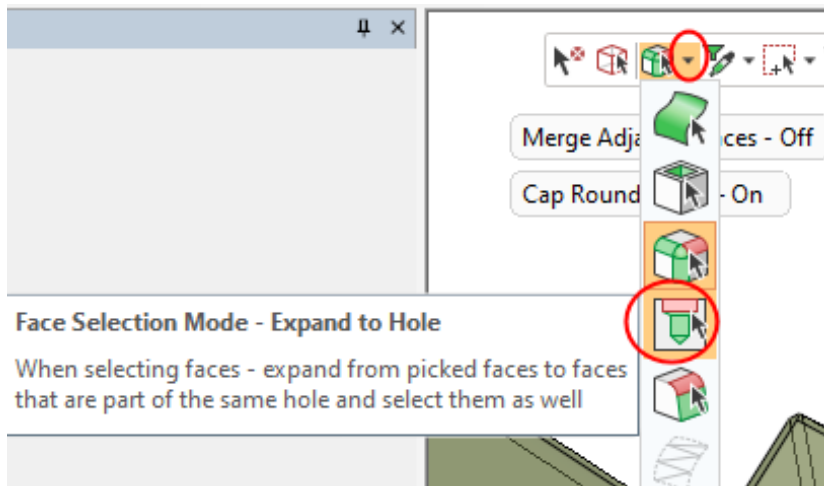


After the part will be printed, these holes will be drilled. So, what we want to do now is to fill these holes with material. Since we are working with a B-rep model, we will again use CAD tools.

84. Activate the Manifold\_05 part, press **Remove & Extend** and pick the holes.

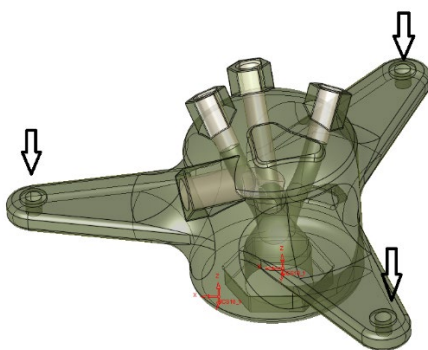


Basically, it is required to pick the bottom and cylindrical faces in each hole. However, picking holes is fast if using the Expand to Hole filter from the floating tool bar and when picking any one face of a hole, the system will find the rest.

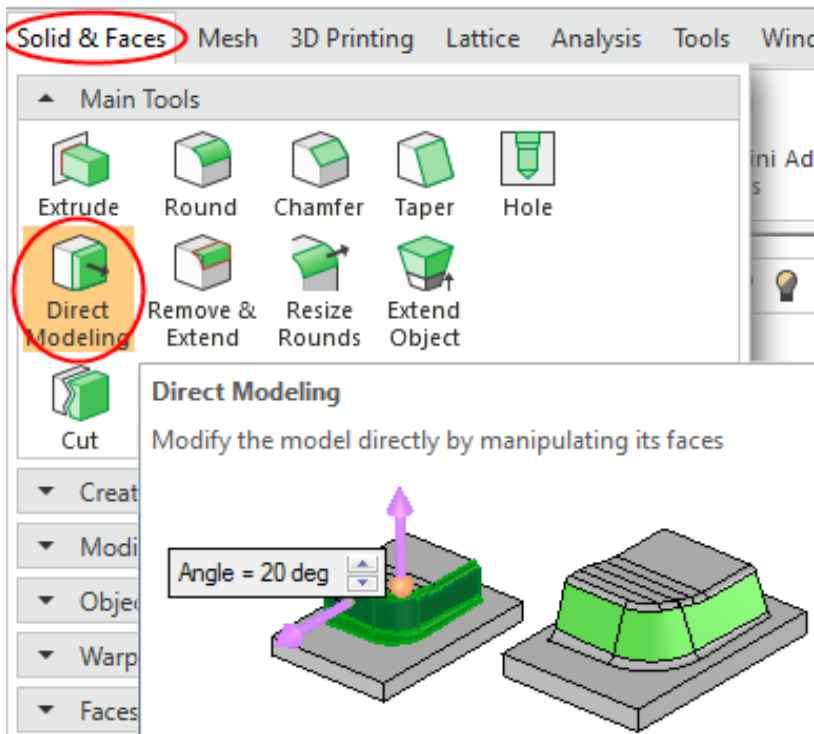


85. Press **OK**. The holes were removed.

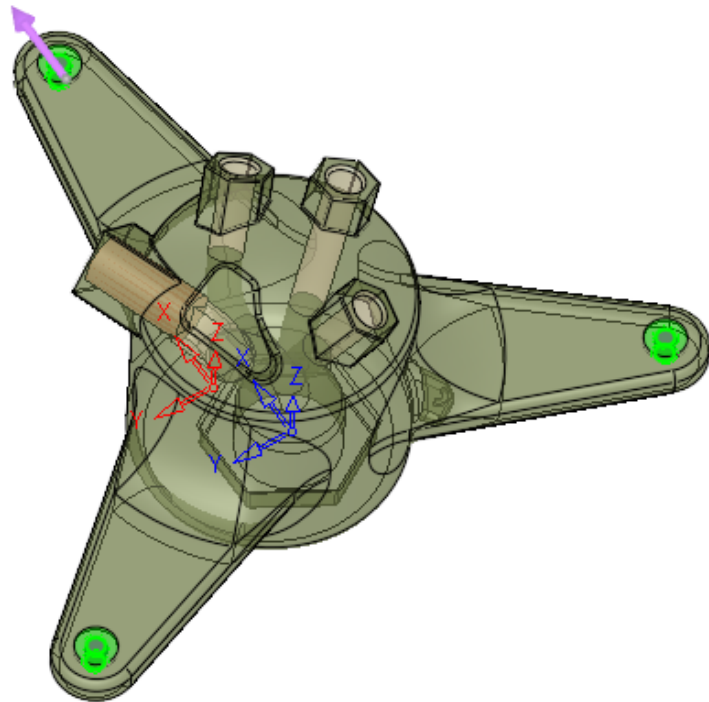
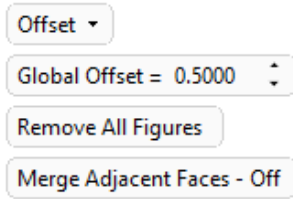
86. Now look at the holes on the wings of the part. For the example, we assume that these will be machined later. We want to make them smaller.



87. Press the Direct Modeling button and select the holes.



88. Switch to the option Offset, check that the arrow points inwards and set the value of 0.5.



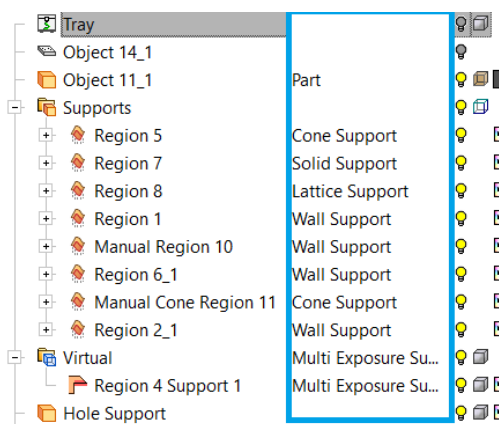
The holes are now smaller.



Each object to be printed should have a related Technology. Each Technology contains its own printing parameters. The printing parameters are saved in 'Buildstyles', so for each Technology a matching 'Buildstyle' can be applied.

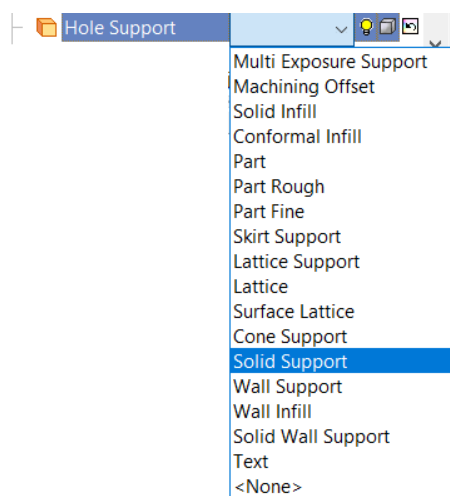
Later on when 3DXpert's Slicer engine runs, the technologies will be transferred to it. The part and the supports created by the dedicated tools we have practiced so far, automatically get a matching technology. Look at the Objects tree and see that there is additional information for each of the objects.

The main part (Object 11\_1) has Part technology and all the Supports that we have created earlier, automatically received their equivalent Technology.



The Hole Support that we have manually created does not have a technology attached to it because we have created it using manual CAD tools.

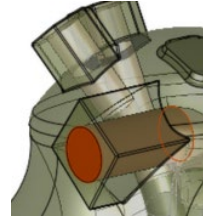
89. In order to assign a technology to this object, **click** to the right of the Hole Support to open the technology list and select Solid Support.



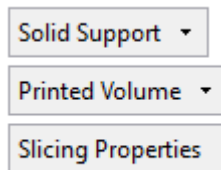
Note: The same can be done also from the Assign Technology function



Click the upper option to open the list of available Technologies, select Solid Support and pick the cylindrical object



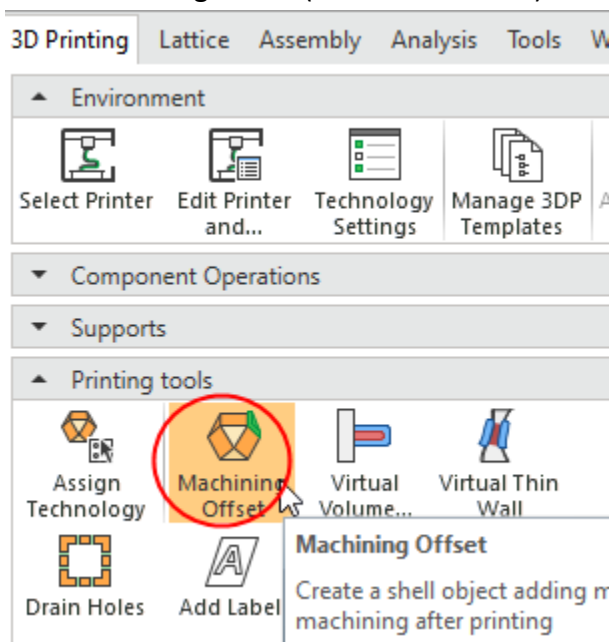
See that it is set as Printed Volume.



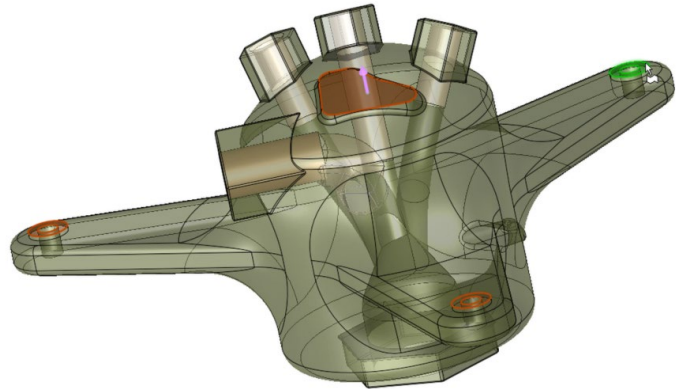
**Press OK.**

We will now work on the areas that will require post printing operations (in this case, by machining.)

90. Click Machining Offset (From the toolbar).



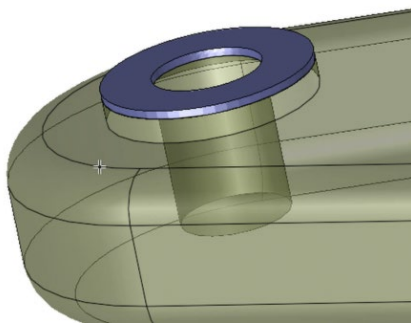
91. Use the Faces option, (note the selection method to Filter Faces) and pick the following 4 faces:



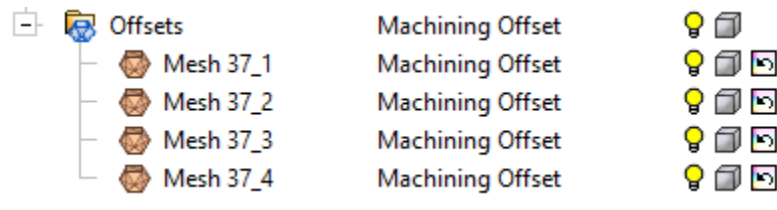
Since we want to machine these faces of the printed part, we need to add some material over them.

92. Set the offset value as **0.5mm**. Press **OK**.

93. Zoom in to see that the system created new objects



The offsets go into their own folder automatically. If you forgot to set the technology to Machining Offset and during work the objects receives another technology, it is easy to change the technology for **the entire folder** (you can do it through the Objects' Tree).

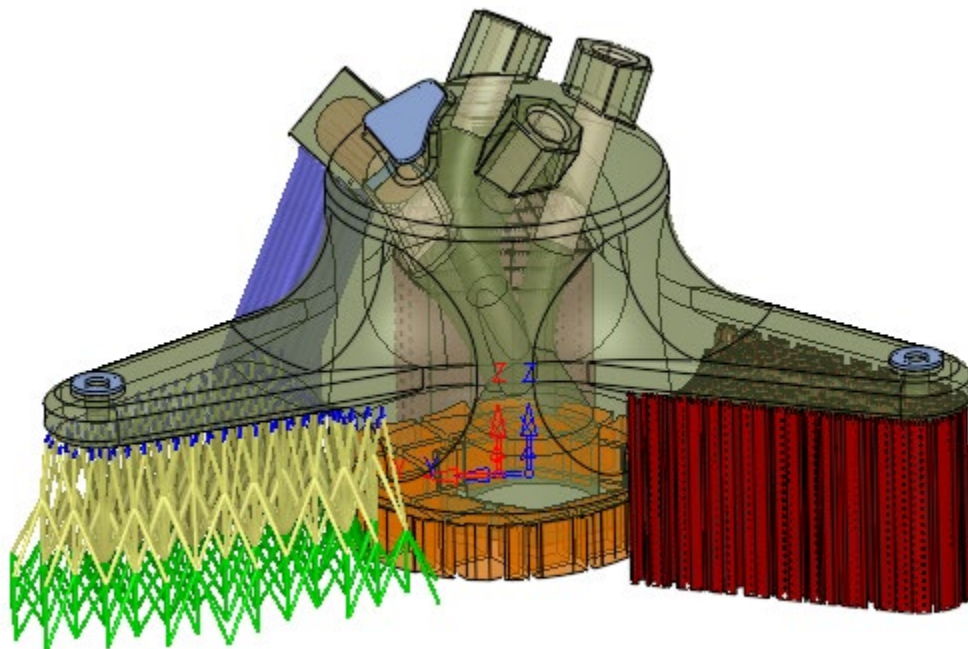



94. **Save** the File.

## Part 6 – Slicing and Sending to Print

We have finished the preparation of our part for printing. The next step is slicing.

95. Show everything, including the virtual objects.

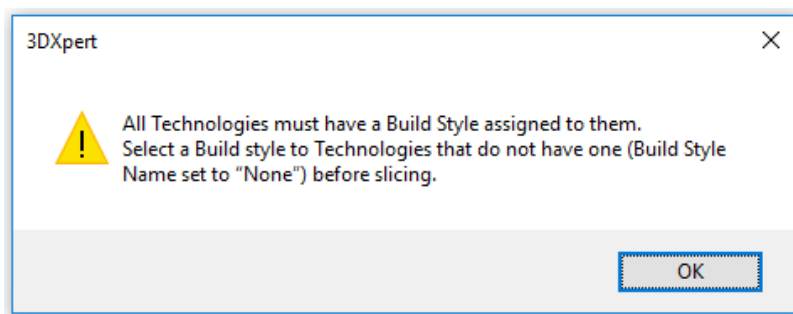


96. From the toolbar click **Calculate Slices**. 

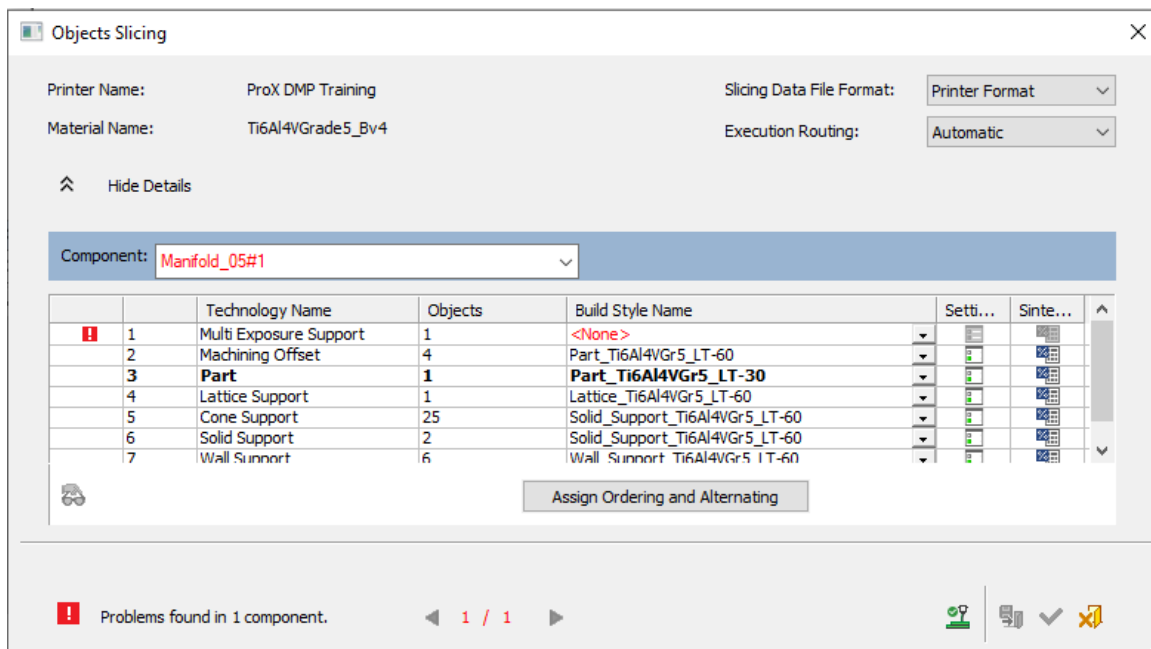
Default settings for the slicer (i.e., which Build Style (or configuration file) to use for each Technology) are set via the Technology Setting interaction; this is not in the scope of this exercise.

If one or more configuration files for the assigned technologies are missing or if you do not have a license for the designated material, you will get the following message:

97. Click **OK**.



The Slicing dialog shows the Technologies we have assigned for this part, how many objects we have for each Technology and the Build Style that is attached to each Technology.



Notice that we do not have a configuration file for Multi Exposure Support.

98. For the purpose of this exercise, set the Build Style Name from <None> to any other name, for example use one of the Part configuration files.

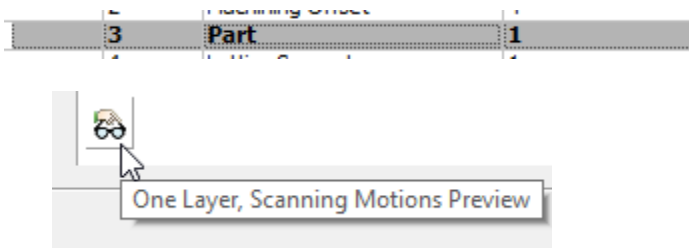
	Technology Name	Objects	Build Style Name
1	Multi Exposure Support	1	Part_Fine_Ti6Al4VGr5_LT-30
2	Machining Offset	4	Part_nor..._LT-60
3	Part	1	Part_Ti6Al4VGr5_LT-30

Notice that the OK and Preview icons are now active.

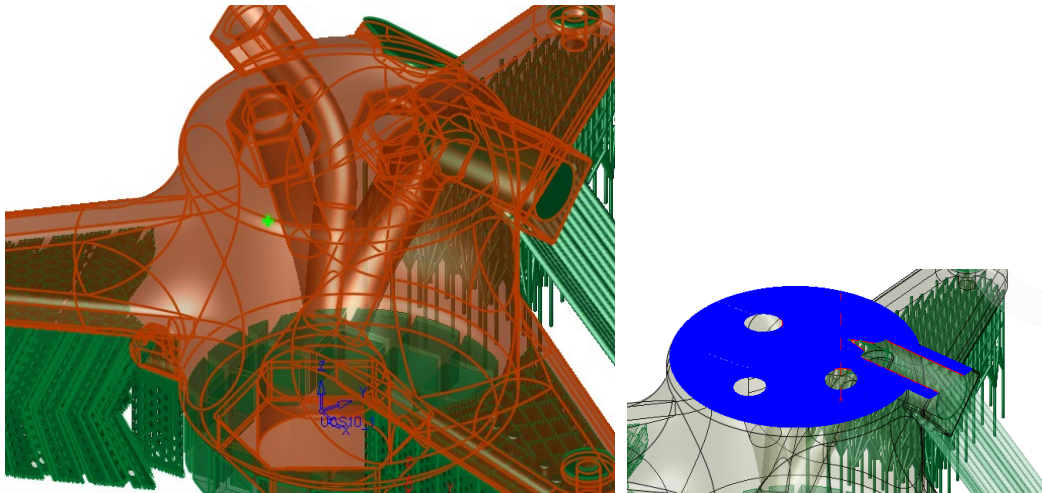


Before running the calculation, you can Preview a specific slice by selecting a Technology from the table and clicking the Preview button at the bottom of the dialog.

99. Select the Part technology and press the **Preview** button.



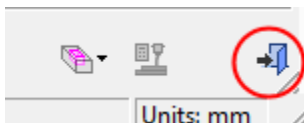
100. Pick a point where you want to inspect the slice. For example, pick somewhere along the cap of the part. After a brief calculation the systems shows the single slice.



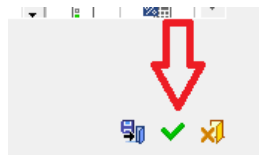
The colors you see in each layer will be explained shortly.

The Layer Control bar to the right of the display is disabled, as Preview is used for a single slice. You can see the layer's height at the bottom of the bar's dialog.

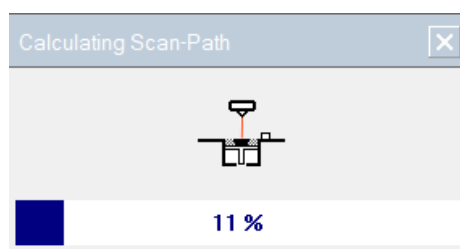
101. Click Exit to return to the Slicer dialog.



102. As we are back in the Slicer dialog, to calculate the slices press the **OK** button at the bottom of the dialog.



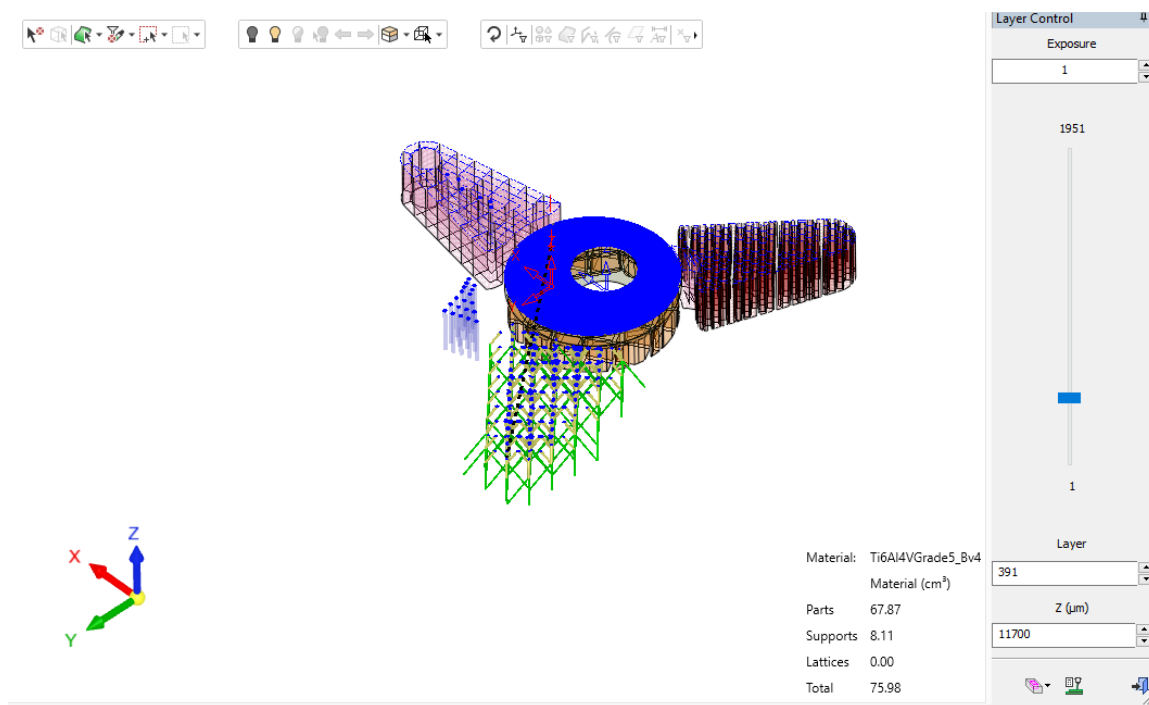
This will take longer to calculate.



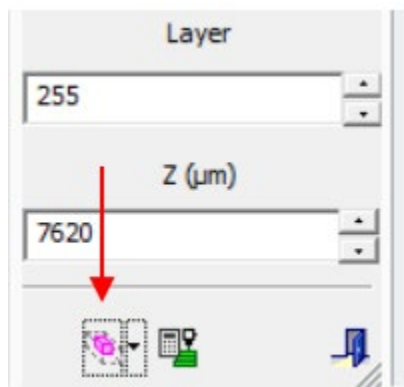
103. After calculation is over, from the toolbar press the **Slice Viewer**.



You can now go through all the slices by moving the slide bar.

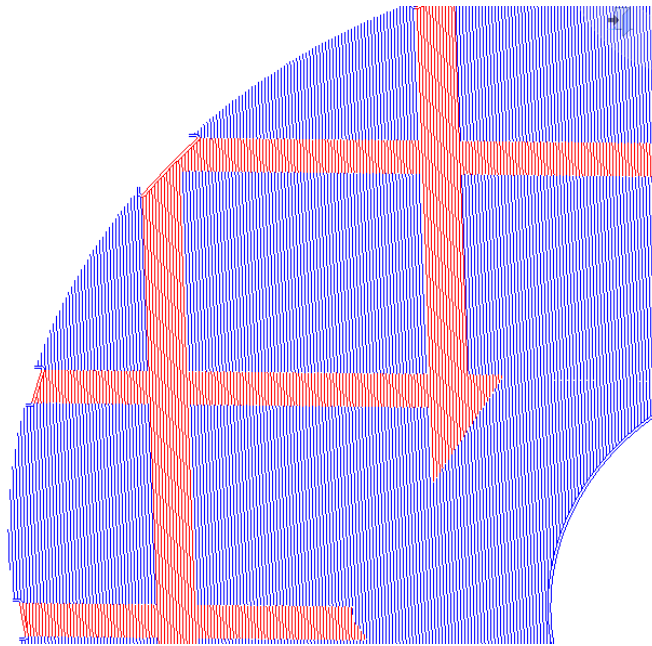


104. To see only the hatch patterns in each layer, hide the parts using the display button at the bottom of the navigation bar.





105. Look at the hatching in each layer and notice the colors:
- Where Down facing areas exist, they are shown in RED.
  - Up facing areas are colored GREEN (however Up facing areas are not defined in the buildstyles used for this exercise).
  - Middle facing areas are colored BLUE.



As you move from one layer to another, you can see that each layer is rotated with respect to the previous one and that some supports (for example, cone supports) are not hatched in each layer, because they are actually printed every other layer, according to their Technology settings.

In each layer, different hatching patterns (coming from different Technology settings) are smoothly connected.

106. Get a Print Estimation. 

As you work, material, time and cost data appears on screen.

Until slicing is calculated, the time and cost are estimated, based on the volume of the model, lattice & supports and the theoretical Build rate of the printer.

Material:	Ti6Al4VGrade5
	Material (cm <sup>3</sup> )
Parts	67.87
Supports	8.48
Lattices	0.00
Total	76.34


Once you slice the part, the estimation can be calculated based on the slicing results and is thus more accurate.

107. Click Print Estimation and note the two modes:

**Volume Based Calculation** – based on the geometry and the given build rates. Does not consider slicing.



**Scanpath Based Calculation** – based on slicing results.

The values in Volume Based Calculations are colored, to enhance the fact that they are estimated based on user defined values and not calculated from a real scanpath.

 Print Estimation
 ✕

☐ Volume Based Calculation  
☒ Scanpath Based Calculation

	Material (cm <sup>3</sup> )	Time (hh:mm:ss)	Cost (USD)
<b>Parts</b>	67.87	09:06:23	339.35
<b>Supports</b>	8.11	00:29:52	40.55
<b>Lattices</b>	0.00		
<b>Between Layers ...</b>		02:04:21	
<b>Machine Time</b>			27.59
<b>Cooling Time</b>		00:33:38	
<b>Total</b>	<b>75.98</b>	<b>11:44:22</b>	<b>407.49</b>
<b>Powder Volume</b>	4,424.91		


\$ Create Report
 

The Material and Machines costs are set through the Estimation Parameters dialog. Enter the time required for a single powder recoating cycle.

The lower part of the Estimation Parameters dialog defines the On Screen parameters – check the options you wish to see on screen, as you work.

### Estimation Parameters

Currency USD - US Dollar

Material: Ti6Al4VGrade5\_Bv4

Material Cost 5.00 USD per cm<sup>3</sup>

Machine: ProX DMP Training

Default Layer Thickness 30.00 μm

General Printing Rate 1.00 mm<sup>3</sup> per sec.

Part Printing Rate 1.00 mm<sup>3</sup> per sec.

Support Printing Rate 1.50 mm<sup>3</sup> per sec.

Lattice Printing Rate 0.50 mm<sup>3</sup> per sec.

Machine Time Cost 2.35 USD per hour

---

Time between Layers ... 2.00 sec.

Cooling Time Factor 0.000 Sec. per μm

---

☒ Present on 3DXpert projects :

☐ Printer Name

☒ Material Name

☐ Max. Height

☒ Material ☐ Time ☐ Cost

☒ Parts

☒ Supports

☒ Lattices



☐ Between Layers

☐ Machine Time

☐ Cooling Time

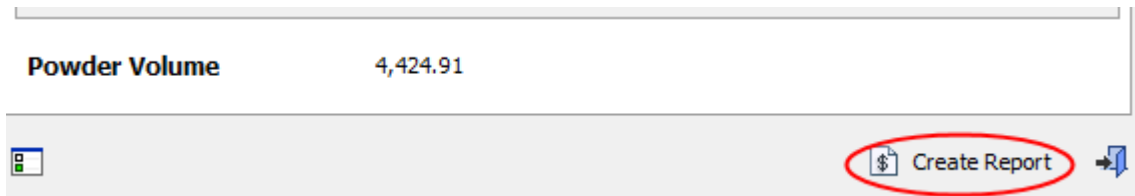
---

☒ Total

The values in Scanpath Based Calculations are calculated based on the slicing result of the part.

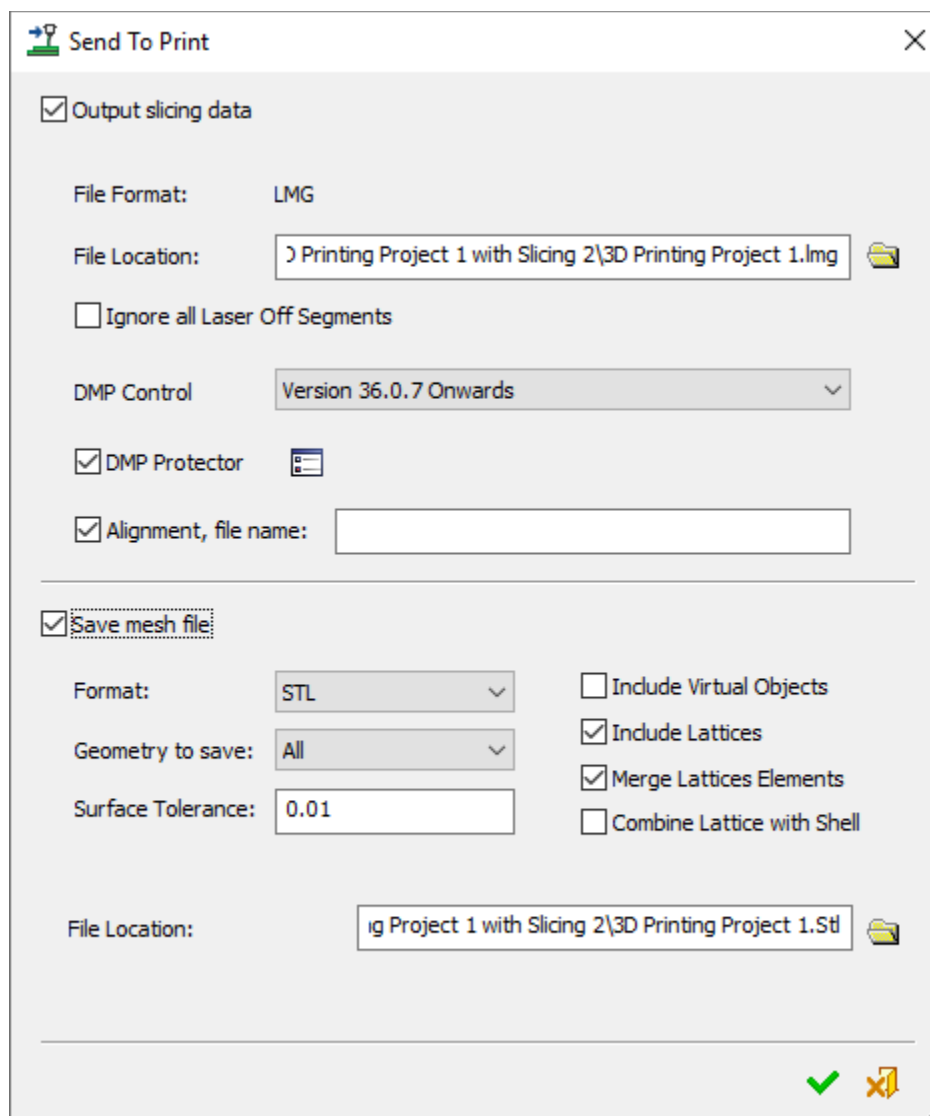
108. Click **Create Report** to export the data into a report in Excel or PDF.



The last stage is to create the file to load on the controller of the 3D printer.

109. Press the **Send to Print** button on the Guide. 

The Send to Print dialog is used to generate the specific output file in the format that the printer knows to read. In this exercise, the training printer is set to output 3D Systems' ProX DMP 320 format, so the file format is LMG, that the DMP Control software reads.



You can also choose to save the geometrical data as mesh.

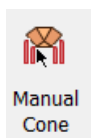
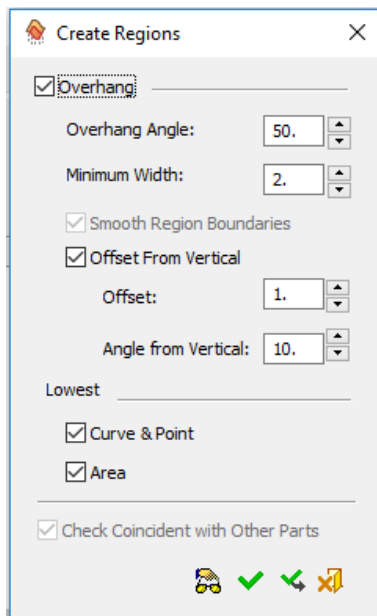
110. Press **OK** to generate the output file.

## Part 3 – Manual Cones – 2 Points

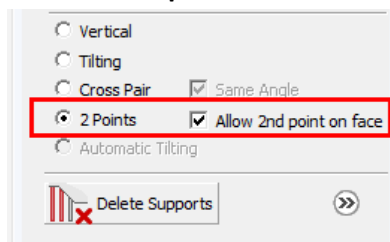
It is possible to easily create individual cone supports, using the part or tray as reference points with tilting. Practically, this offers an easy way to create an angled support using 2 anchor points on the part - a gusset.

This is done by 'Manual Cone Supports - 2 Points'.

1. Unpack the file cone\_support.ctf and load the project.
2. Enter the **Support Manager**, cancel the Create region dialog and enter Manual Cone .



- Click the **2 Points** option and check “**Allow 2<sup>nd</sup> point on face**”:

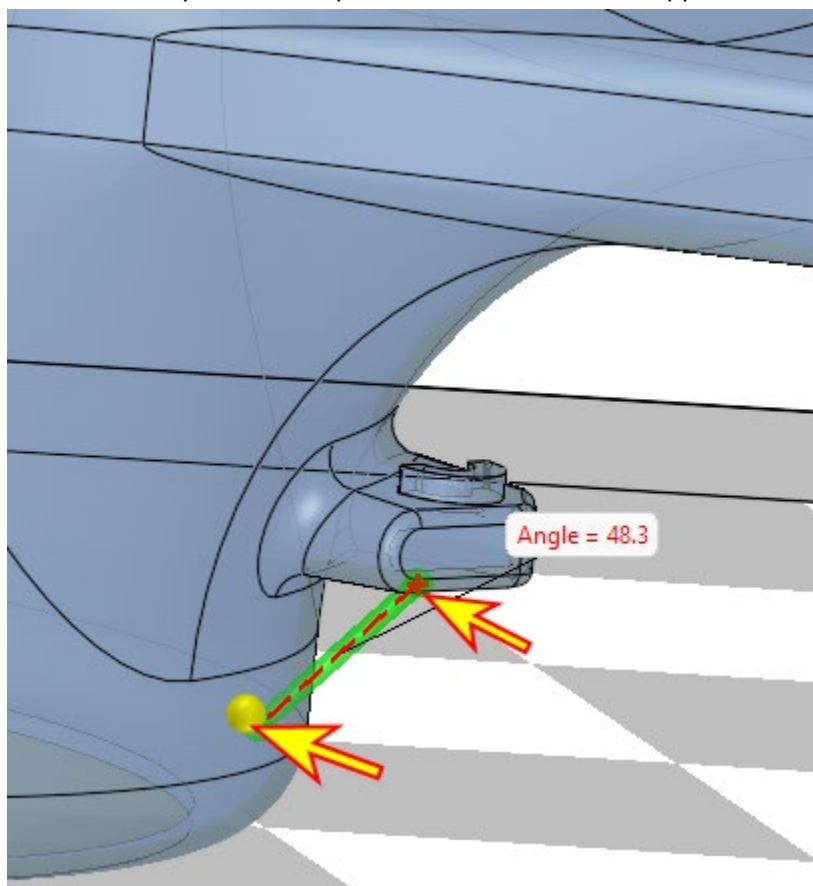


Note:

Commonly, it is required to make such cones by 2 points - between part and tray.  
 In the 2<sup>nd</sup> selection, the part may interfere with the selection on the tray and therefore selection on a part is disabled.  
 If a gusset is required, i.e., you need to pick two points on the part, then use the option “Allow 2<sup>nd</sup> point on face”.

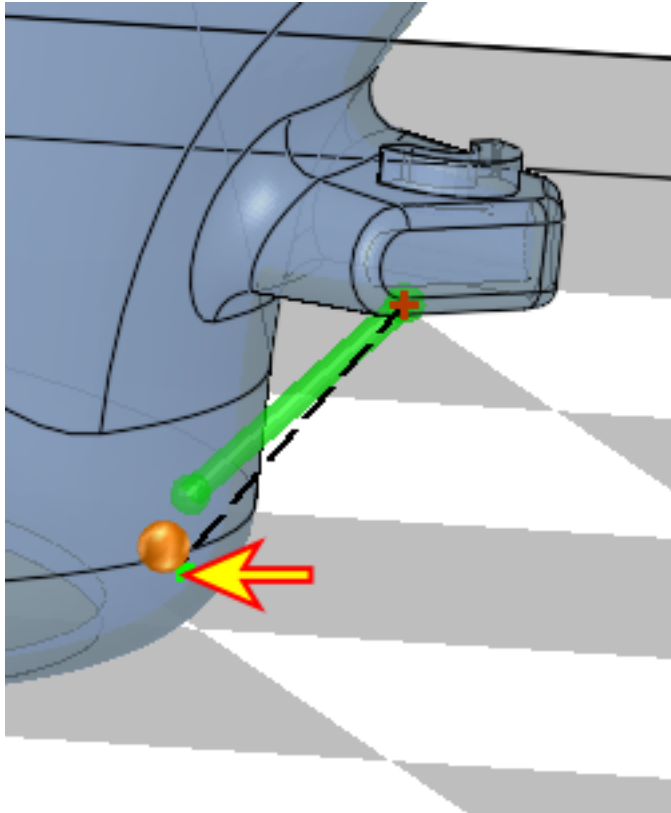
Each cone can be placed between any points picked on the part or the tray.

- Pick the reference point on the part so that the cone will support the horizontal area:

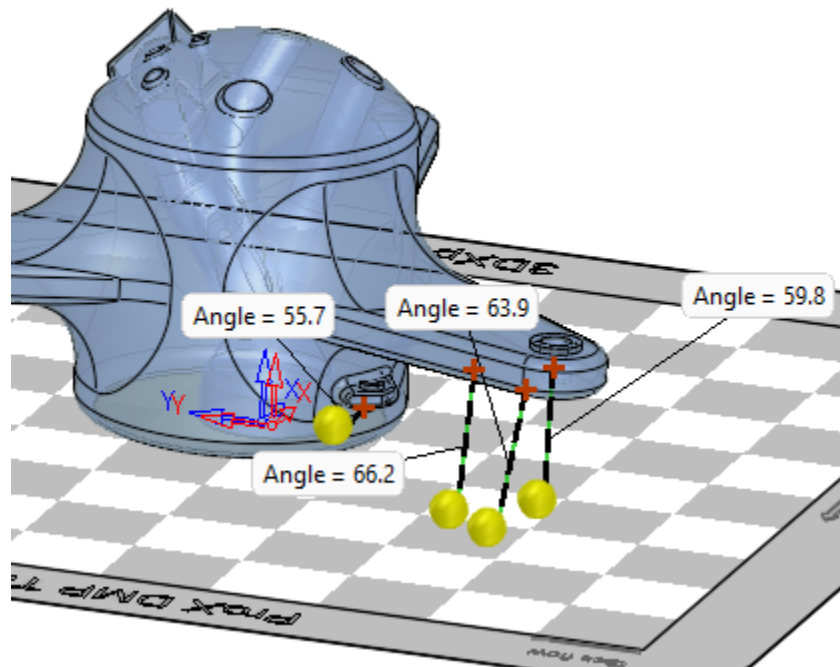




5. You can now drag each point to set the tilt or position it anywhere you like.



This method enables you to easily create multiple cones with the same parameters, with different tilting for each cone, as in the following representation:



End of Exercise.