3DXpert™

3DPRINTING EXERCISE

Create Project & Analyze Part

Tutorial_V10 - Updated: 14,0100,1592,830(3DXpertv14SP1)

3D SYSTEMS

Table of Contents

Welcome to 3DXpert. This exercise will guide you through the 3DPrinting environment of 3DXpert.

It focuses on 3DXPert's 3D printing functionality. Although the various steps to follow are detailed, it is recommended to have a basic knowledge in running the general software tools.

In this exercise, we will create a new project, add the part and analyze it, then use the various tools and options to prepare the part for printing by building a variety of metal supports.

One of the advantages of 3DXpert is that it does not work only on Mesh objects. In this exercise we will work on a B-rep* CAD model.

* "B-rep" is the short term of 'Boundary Representation'. In CAD, This is a method for representing shapes using their limits. For example, a solid object is represented as a collection of connected surface elements; a face is represented by a surface with boundary limits.

Part 1 – Creating the 3D Printing project and loading the part.

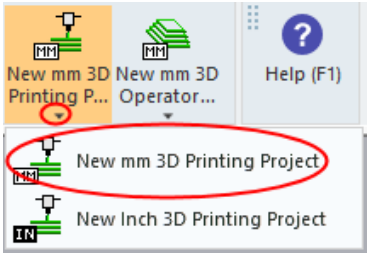
While it is possible to add several parts and arrange them on the tray, in this exercise we will work with a single part.
Another way to position several parts on the tray is through the 3D Operator environment (this is not discussed in this exercise).

Note:
In this exercise, for training purposes we will use a 'training' printer and material database. This printer is not suitable for actual printing.

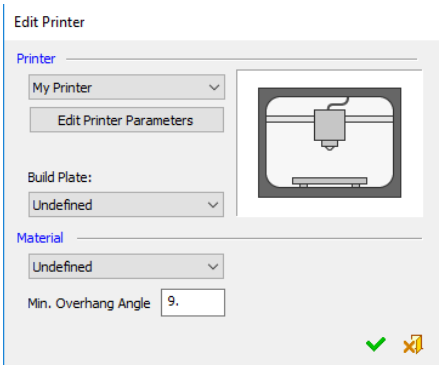
Installation of the 'Training' Printer:
1. Extract the file 'ProX DMP Training.zip' to the folder:

C:\ProgramData\3D Systems\3DXpert\14.0\Data\3D_Printing\Technology_Folder
The result is a new folder called ‘ProX DMP Training’.

2. Let’s open a new 3DPrinting project. From the menu bar press the ‘New 3DPrinting Project’ button.



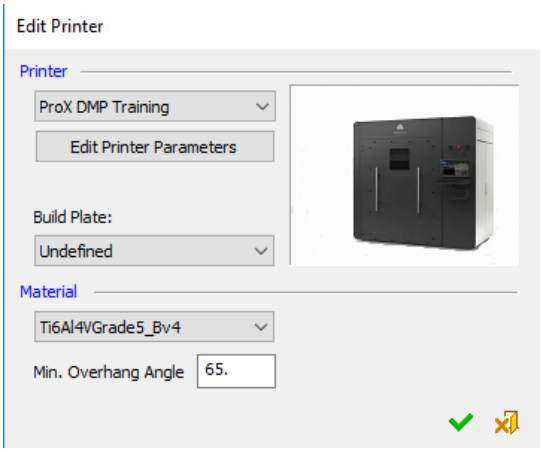
After the installation, creating the very first project, the system uses a default Printer called ‘My Printer’. This is a generic printer.



In this exercise, we will switch to the ‘training’ printer.

The system remembers that last used printer, so when you switch to a different printer, it will be used also in the subsequent project(s) that you create.

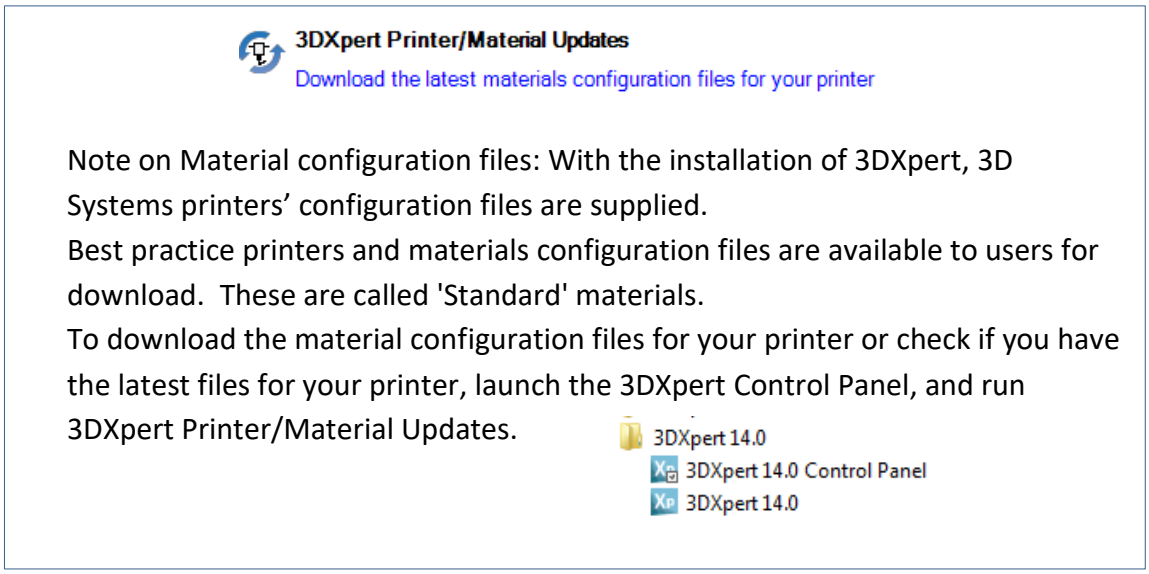
3. Through the Guide Bar, click ‘Edit Printer’ and define the printer you are going to work with. From the list of available printers, select ‘ProX DMP Training’.



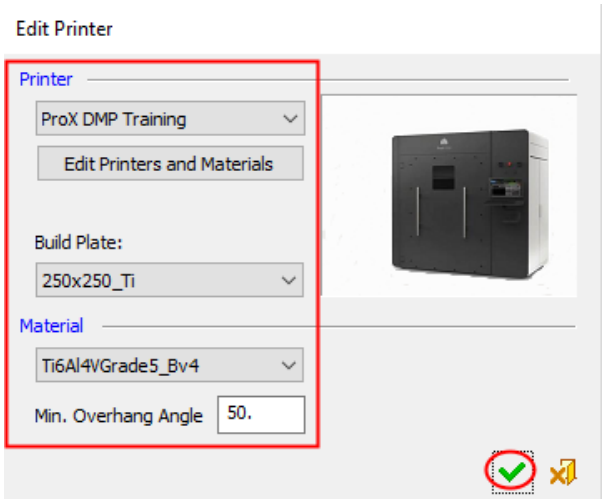
4. Select a **Build Plate** (for the purpose of this exercise, this can be any plate scheme). The plates that are available are the standard plates which can be obtained through the printer's vendor. For example, select from the list the plate **250x250_Ti**.

5. Define a Material. The list shows only the materials that are relevant for the selected printer. For this exercise, select the material **Ti6Al4VGrade5_Bv4**.

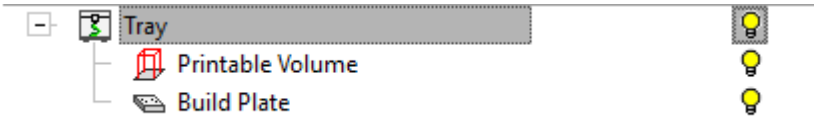
   The definition of material is essential, as without a material it is not possible to load\set a matching Build Style. 'Undefined' material means that Technologies will not be available later on, so if you do not define a material, the system will prompt when pressing the OK button.

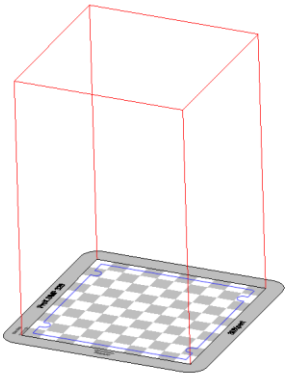


6. Back in the Edit Printer dialog, set **the Min. Overhang Angle as 50**. We will see later on how this parameter is used. Press **OK** to finish the project's setup step.

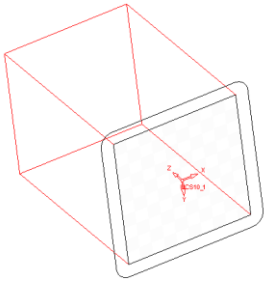


Note: For a clear view of the part, you can always hide and show the tray by pressing the bulb alongside the Tray part in the project's tree.





Also note that if you rotate the tray and look at it from the bottom, it becomes transparent, in order to clearly see the part.

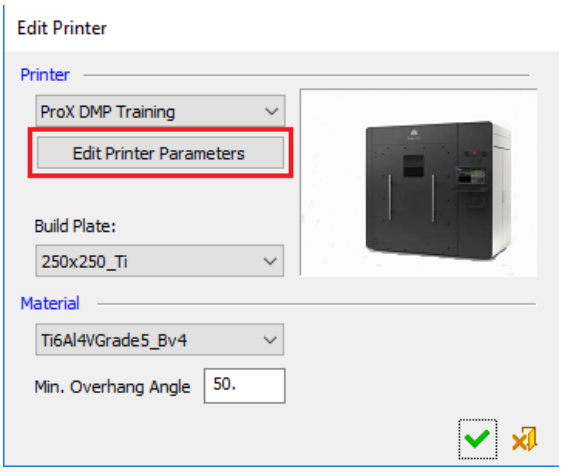


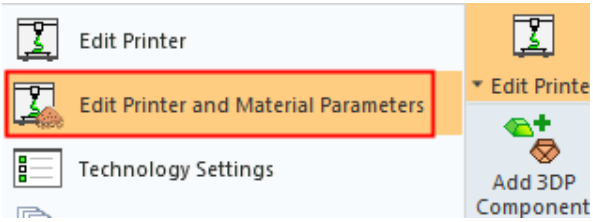
The tray from bottom.

The text on the tray describes the printer type, the Recoater\Roller direction and the Gas Flow direction, as set in the printer parameters (as we shall soon see).

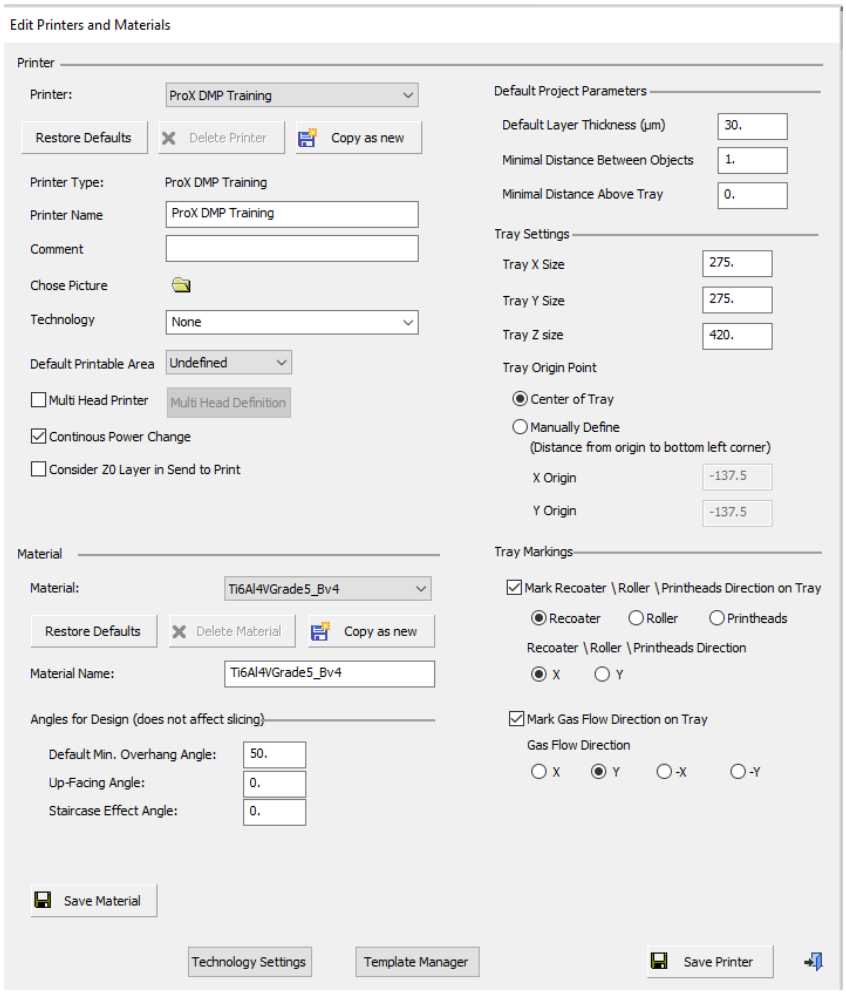
7. Notice the Process Guide on the right side of the screen. This guide contains most of the functionality to enable preparing the part for printing.
   The upper one is Edit Printer, which we already used.
   In addition to the Printer, Build Plate, Material and Angle, there may be various other parameters you would like to control.

8. Click **Edit Printer Parameters** to enter another dialog. (You may also access the dialog directly from the 3D Printing Guide Toolbar)

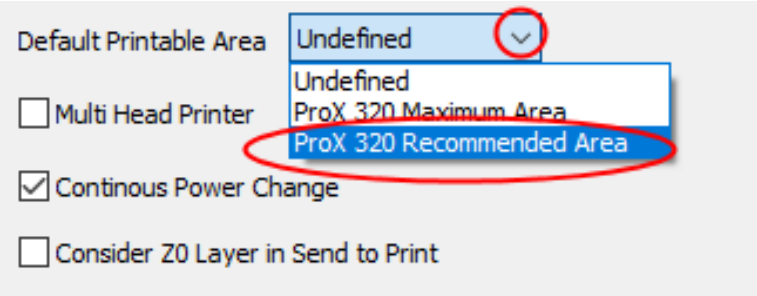




This dialog contains various parameters of the specific printer you are using.
Most of them are dictated by the printer and should not be changed.
Notice among others the XY& Z tray sizes, the direction on which the roller or recoater move and the gas flow direction. The gas flow ensures that the smoke emitted during the printing operation will not create a mask that interferes with the laser beam.
These parameters will be used later on by the Slicer, as the data is prepared for printing.

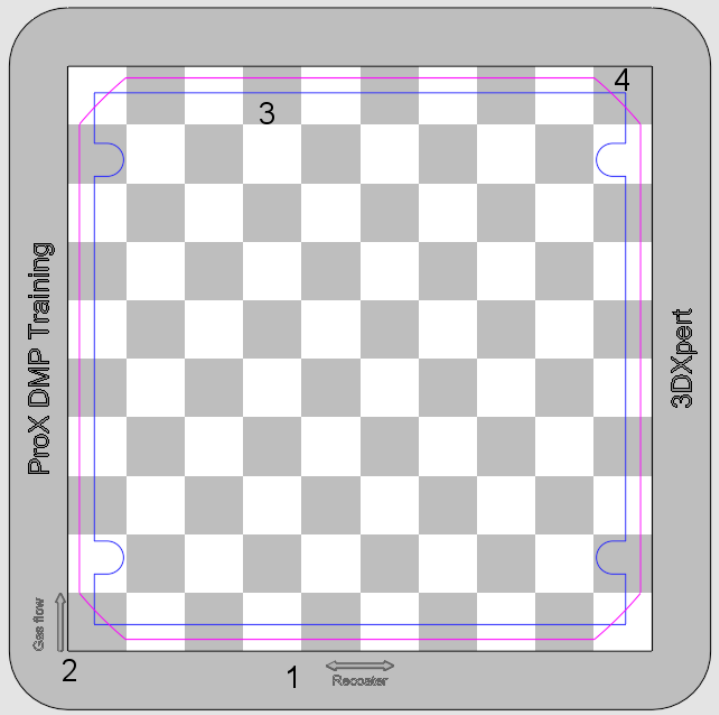


9. Set the **Printable Area** (the area on the tray where the laser beam is in optimized working conditions) as Recommended Area:

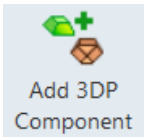


10. Press the **Save Printer** button and then Close to exit the dialog. Press **OK** to confirm and exit the Edit Printer dialog. Look at data presented on the tray:

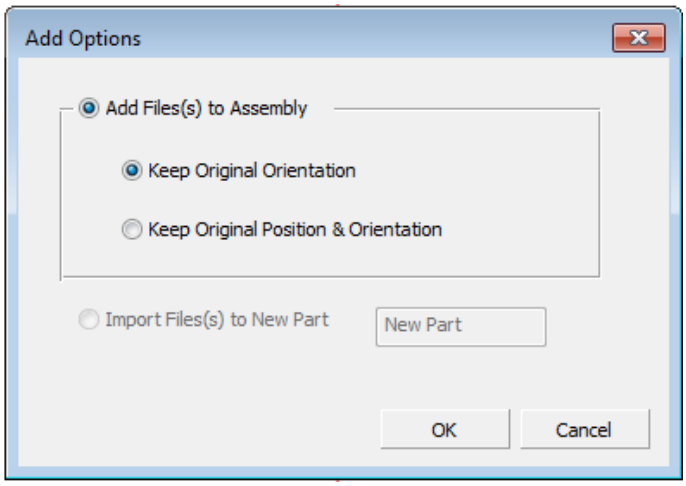
1-Roller/Recoater direction
2-Gas Flow direction
3-Build Plate scheme (blue)
4-Prinable Area scheme (purple)



The Build Plate and Printable Area are taken into account in the printability checks (see later).

11. Let's add the part we are going to print. Press the **Add 3DP Component** 

. As the 3DXpert explorer opens up, browse and select the file **Manifold_05.elt**. Press 'Select' or double click the file. A dialog shows up.
12. In this Add Options dialog, you can set how multiple parts will be added to the 3DXpert assemblies. Or, alternatively, you may also add them the same part.



Let us review the options:

1) **Keep Original Orientation** - This option enables to bring each part as originally positioned. However, the parts are automatically nested on the tray and not placed in the center of the tray.

2) **Keep Original Position & Orientation** - This option is suitable for loading an existing assembly, as this keeps the parts in the same position as originally located in respect to the assembly. Note, that the system does not load the parent assembly itself. Only the parts are added, and even if the assembly contains sub-assemblies, only the parts from the assembly are added, each as a separate file. NOTE: In order to keep the parts' relative position, make sure to select all of them when using the Position Body tool (see later).

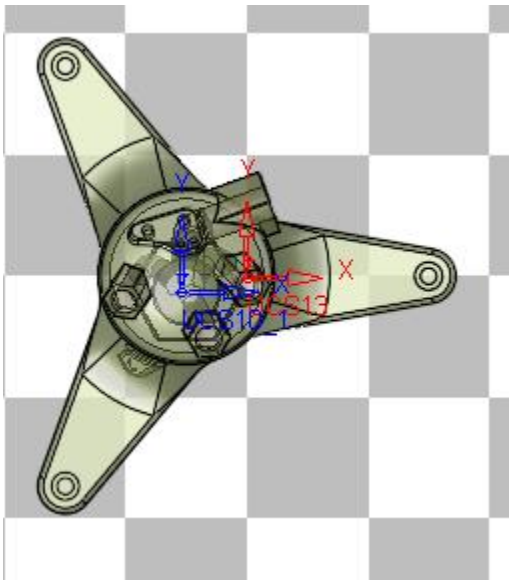
3) **Import Files to a New Part** - This option is enabled only when adding several parts. In many cases, you do not want to keep the existing assembly with multiple parts (as original saved) but rather to combine several designed parts to a single part.

This mode enables to import all the components from the assembly or multiple parts to a single part, so that this part includes all the relevant objects.

If only a single file selected (as in this exercise) and it is a part (not an assembly), this option is disabled.

13. Select '**keep Original Orientation'** and press **OK**.

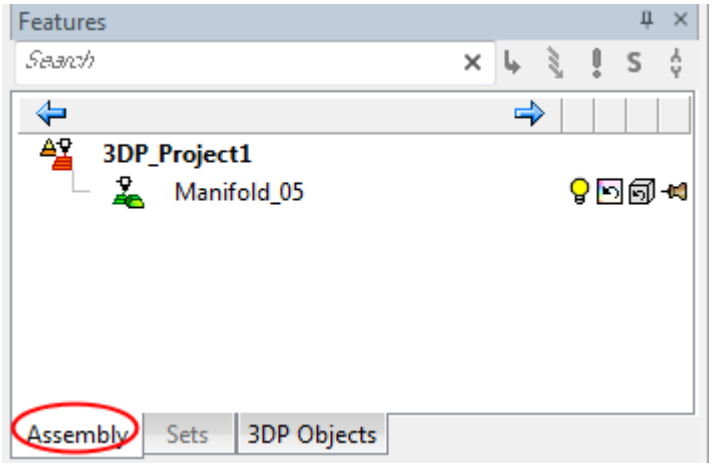
The part has now been added to the project.



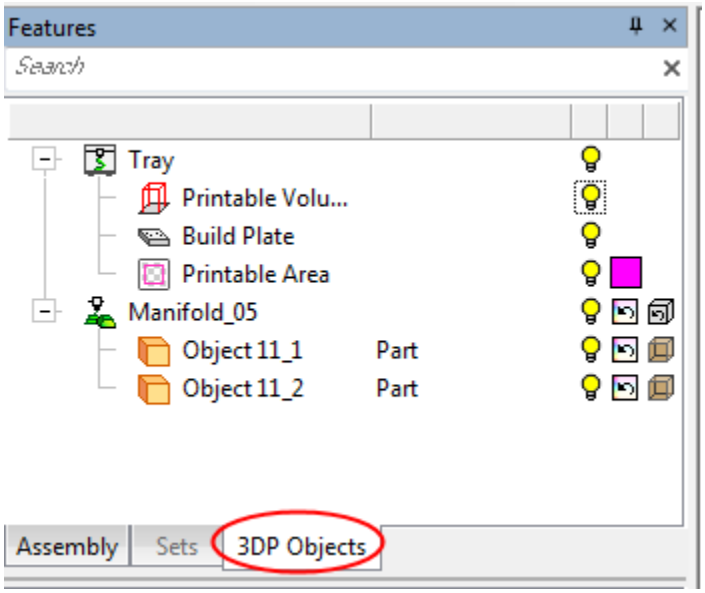
The project data tree is organized in two CAD modes - a CAD standard Assembly Tree and a dedicated 3D Printing Objects Tree.

14. To switch between these mode click the relevant Tab (as shown by red circle).

The Assembly Tree



The 3DP Objects Tree



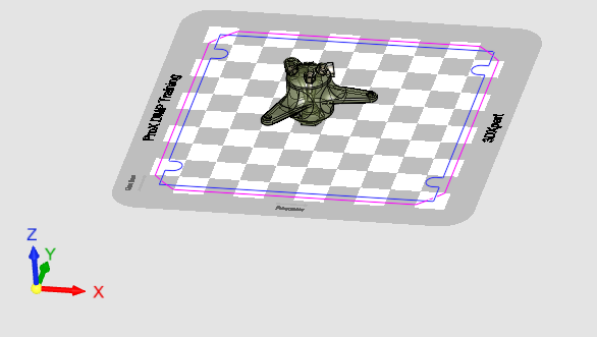
Although we have added the build plate and a single part, there are two objects visible in the 3DP Objects tree (Object 11_1 & Objecr 11_2).

The reason for this is that the added part (whose name is shown in the assembly Tree) is composed internally of two objects (we will come back to the handling of these objects later on in this exercise.)

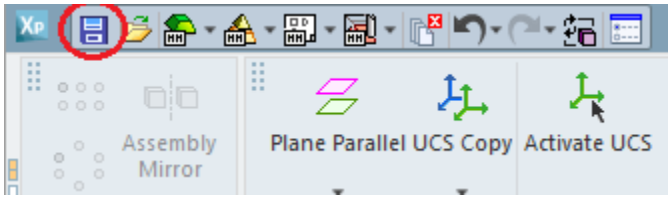
The default names of the objects are assigned by the system and can be renamed by the user.

The Objects Tree also includes additional Technology data that we will discuss later on.

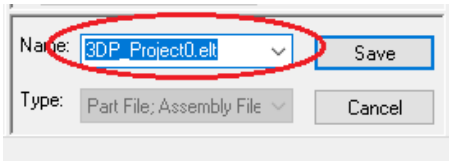
The part is now placed inside the tray area:



15. At this point, before we continue, let's save the project. Click Save.



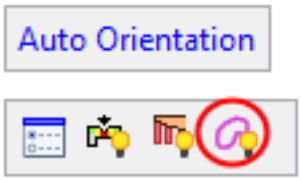
16. The Browser opens up. Keep the default name and press **Save**.

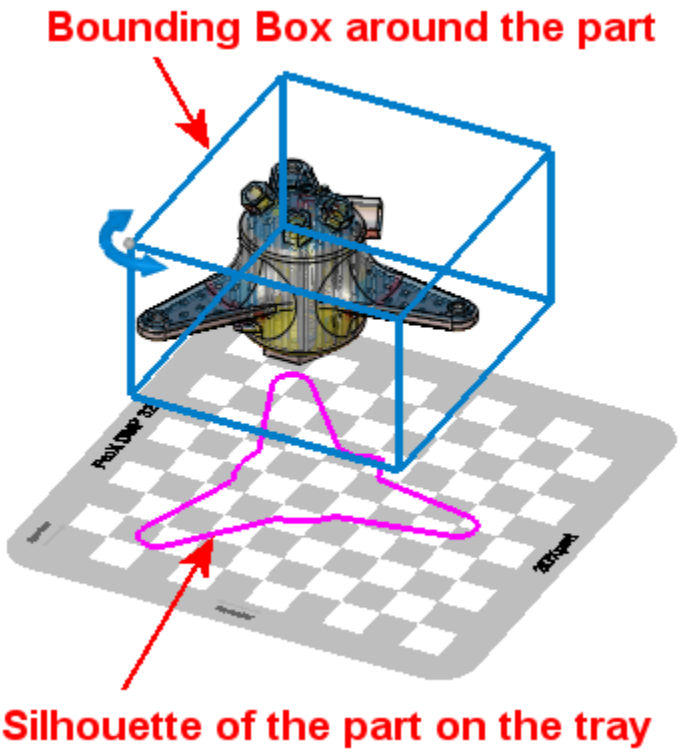


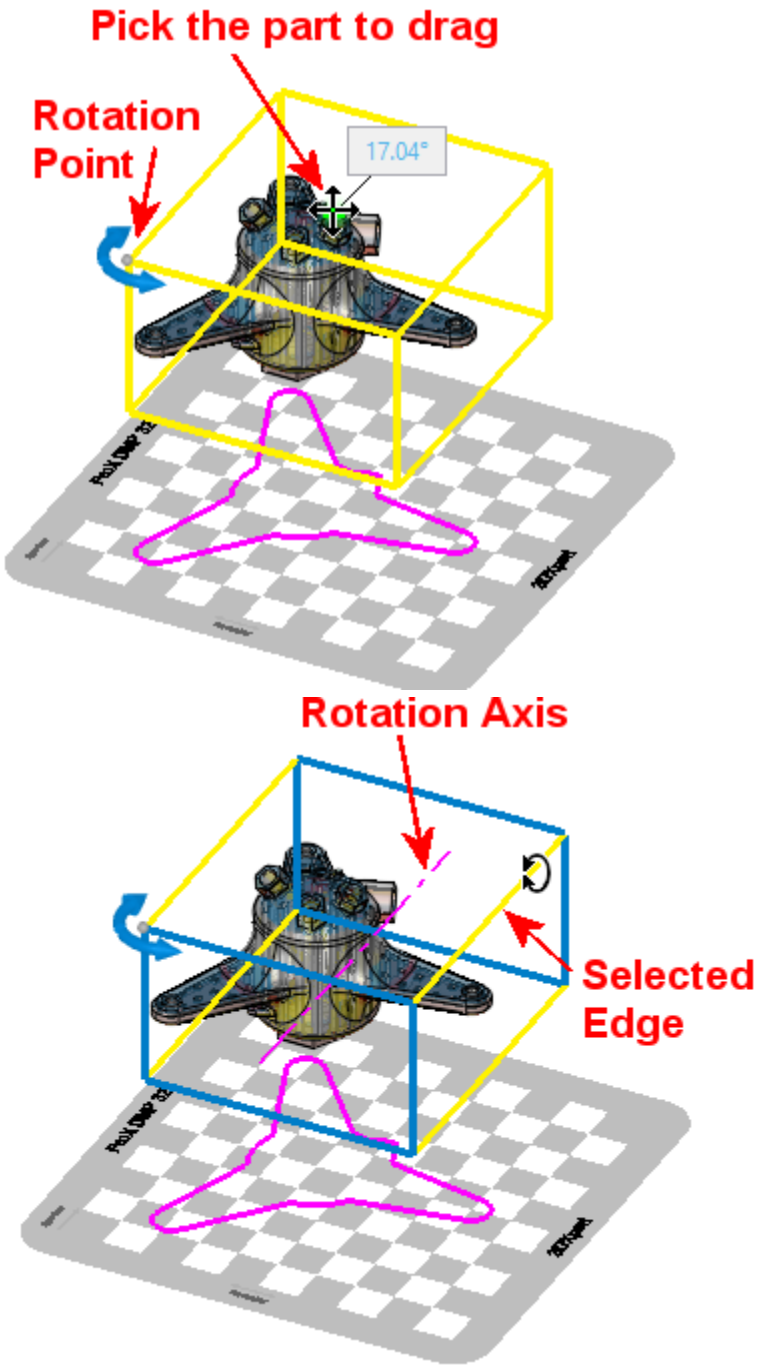
Part 2 – Position Body

17. The current placement of the model is temporary. To properly position the part inside the tray, press **'Position Body'** from the Guide Bar

18. Notice the bounding box around the part. You can drag it around freely by picking the part and moving the mouse, or rotate it by picking any of the edges of the box and moving the mouse. See that the part cannot escape the boundaries of the tray. If during its rotation the part protrudes below the tray, once the mouse is released, the part will jump to position inside the tray. The part can escape the boundary of the tray (but not below it) if selecting ignore component
19. Ensure that the “Show Silhouette” button is pressed.

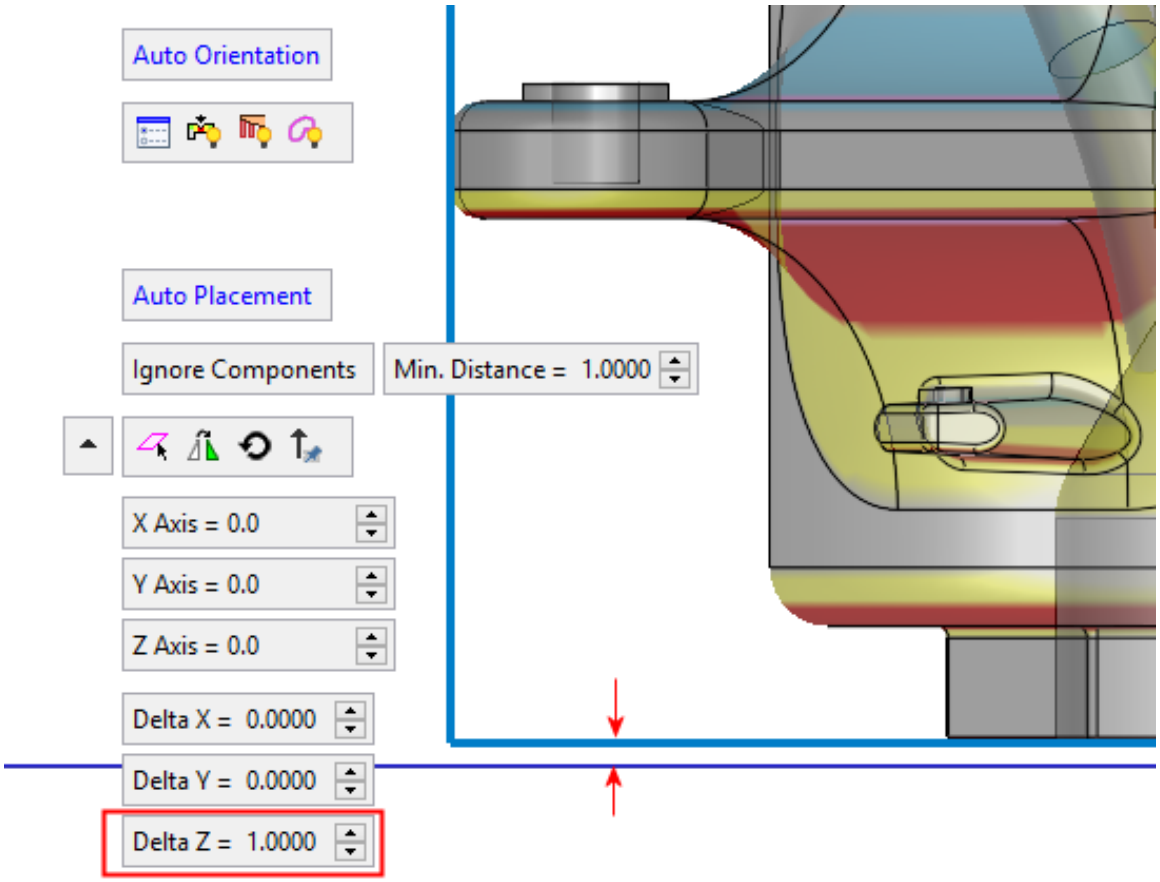






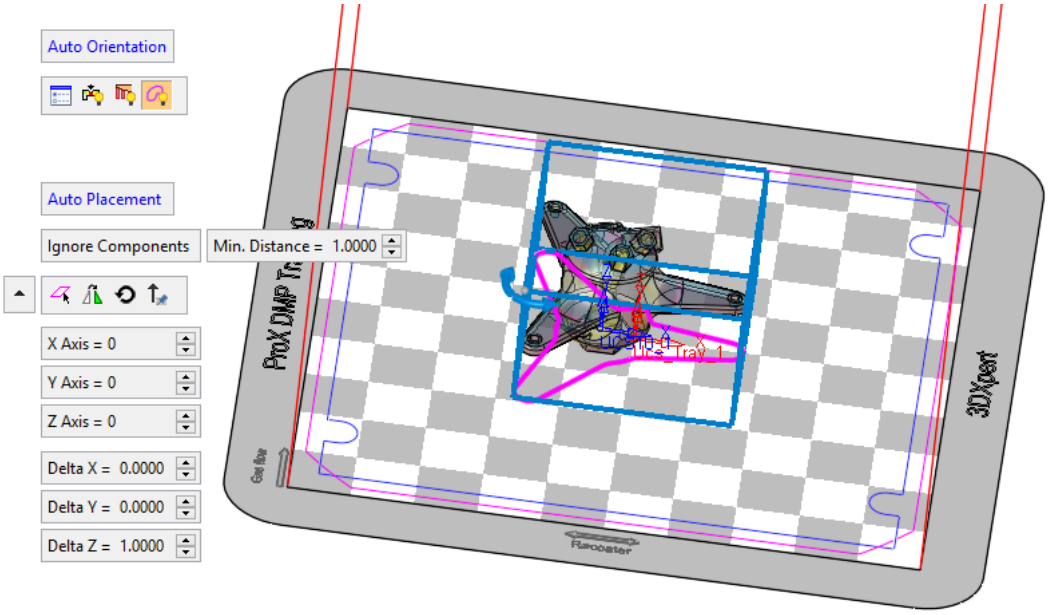
There are several tools to position the part according to the best practice. We will now go through various positioning functionality.

20. The position of the part above the tray is dictated by the Delta Z parameter.



21. Change the value to see it moving up and down.
22. During positioning of the part, the system runs a down facing analysis

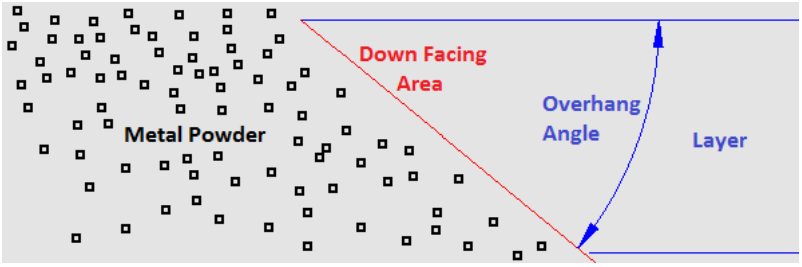
The down facing analysis is a visual analysis tool, based on the defined Overhang Angle. See that some the part's faces are colored in yellow or in red. These are the down facing areas (or, in other words. faces that 'see' powder below).
The value of the angle is taken from the printer parameters dialog, and can be changed here. As the value is modified, the system updates the analysis results accordingly.



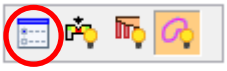
The part areas colored in red are those which require a support (again, according to the min. angle). The down facing areas colored in yellow will meet the powder but do not need a support.

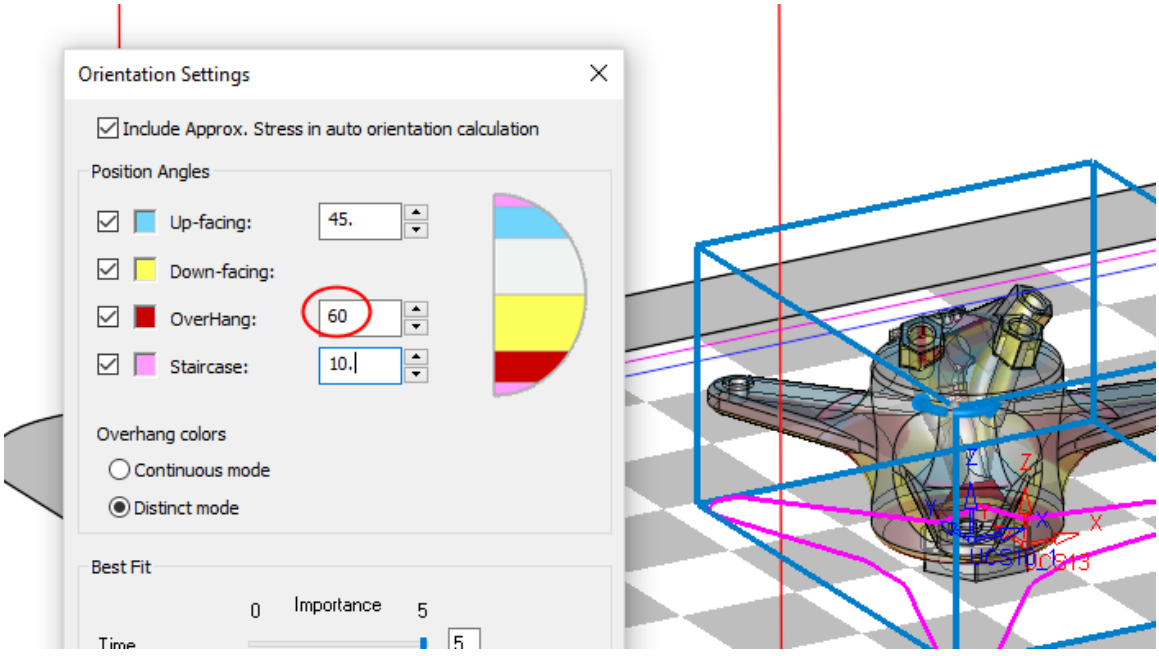
More on the Overhang Angle:

The overhang angle (previously defined in the Edit Printer dialog) is measured from the horizon.



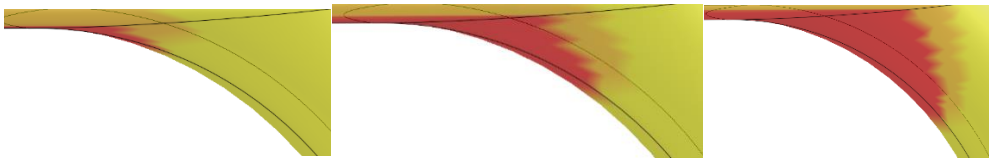
The 'Overhang Angle' parameters as well as some other parameters can be change through the **Orientation Setting** Dialog, which be explained later on.

23. Press the **Orientation Setting** icon.  . Here the 'Overhang Angle' parameter shows 50, as we have defined before.
24. Change the Overhang Angle from 50 to 60 and see that additional areas will be colored in red.



25. . To better see why, look at the following images.

As the angle increases, additional areas will require supports to sustain printing, as shown in the following example:

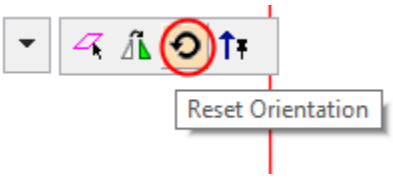


20 degrees 35 degrees 50 degrees

26. Rotate the part by picking its edge. The analysis will rerun and update the colors according to the 'new' down facing areas.
27. Rotate the part by entering values to the X,Y & Z Axes boxes. Note that when entering values to the X,Y & Z Axes parameters, the rotation is performed around the main tray axes.

In addition, when entering Position Body or using Auto Orientation \ Select Plane, all the angle values are reset to zero. This way it is easy to fine tune the position and set a delta from the current position. These changes also led to the new Reset Orientation button.

28. The Reset Orientation button restores the part to the zero position (i.e., as placed originally). Press the **Reset Orientation** icon.

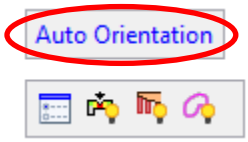


The '**Auto Placement'** button puts the part in the center of the screen at the lowest point, keeping the current orientation. (note that the Delta X,Y & Z values will get a value of zero).

So far we have discussed the manual position tools.

Let's see now the **Automatic Orientation tool**. This tool appears at the top.

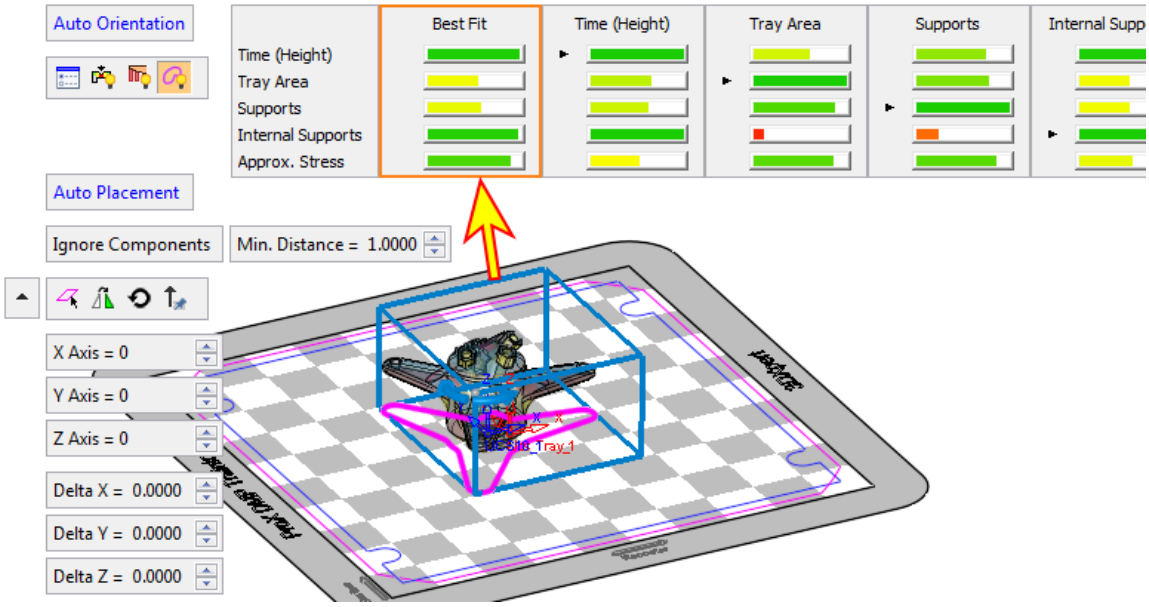
29. Press the **Auto Orientation** button.



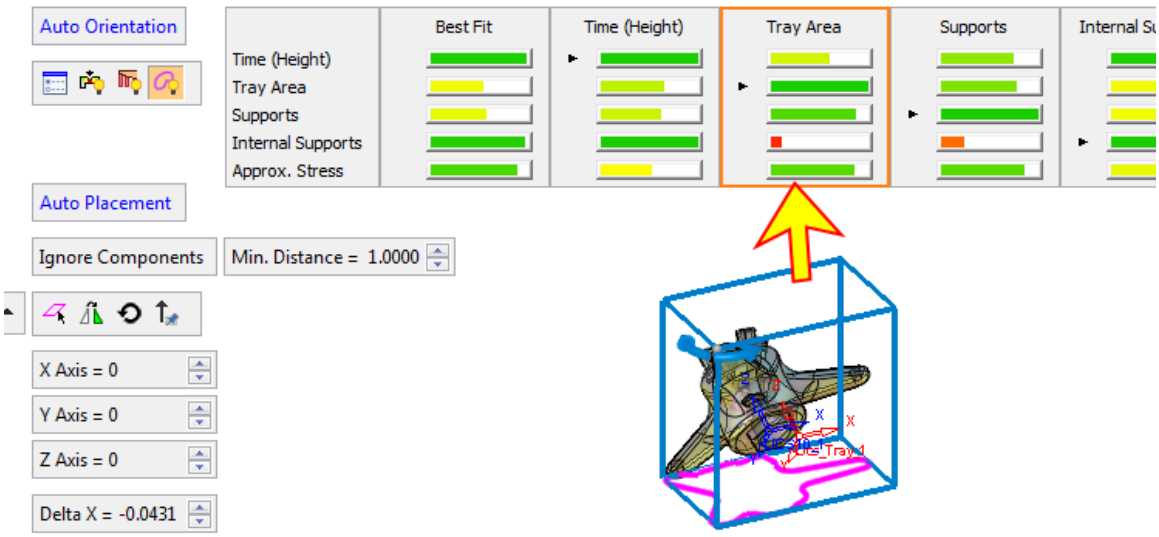
The system analyzes and automatically orientates the part on the tray, based on the following analysis criteria: Time, Tray Area, Supports, Internal Supports and Approximated Stress. The results are displayed in a panel showing the optimum orientation for each analysis criteria.

By default, the system displays an optimized orientation that **Best Fits** all the analysis criteria, based on the weight importance given to each of criteria in the Orientation Settings dialog, or the default weight values defined in the Best Fit Positioning Preferences.

Within each calculated analysis criteria in the panel, the best possible fit for a specific parameter is indicated by an arrow and a color coded marking, where the darkest green indicates the best fit.
When an analysis criteria in the panel is selected, it is highlighted and the graphics window displays the resultant part position on the tray. In the image below, the **Best Fit** group is marked as selected.

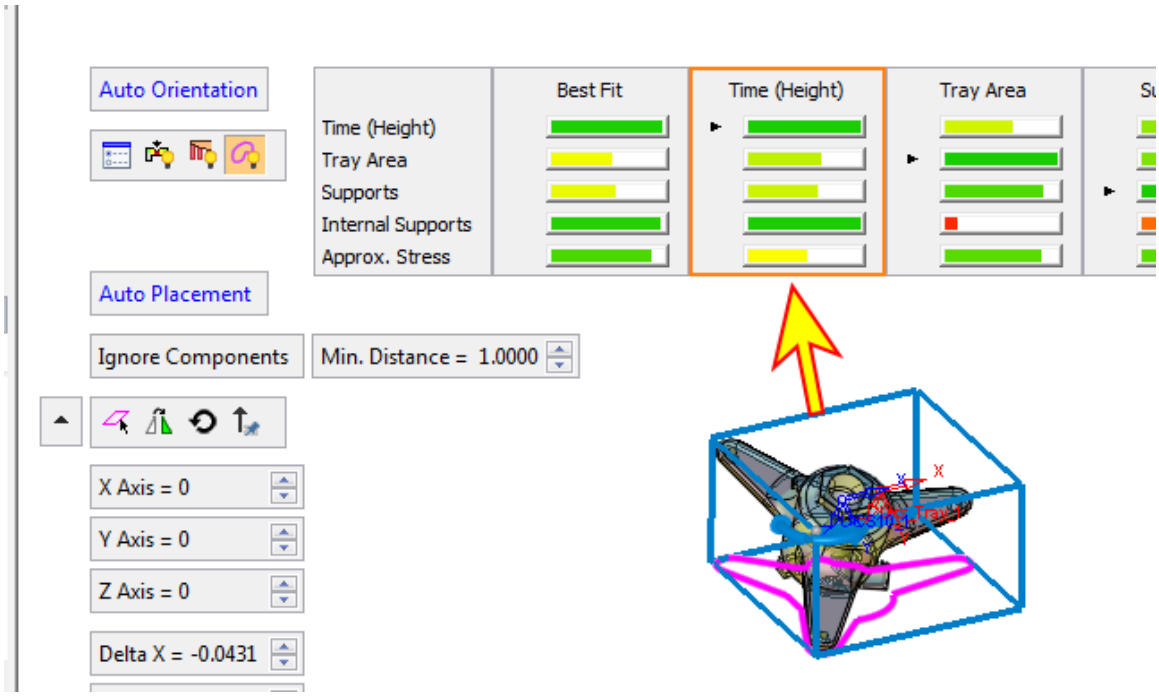


30. Press the **Tray area** selection in the panel. This results in the orientation that will yields the least amount of tray area used (based on the part's bounding box). The bounding box is positioned so that its projection on the tray is the minimal

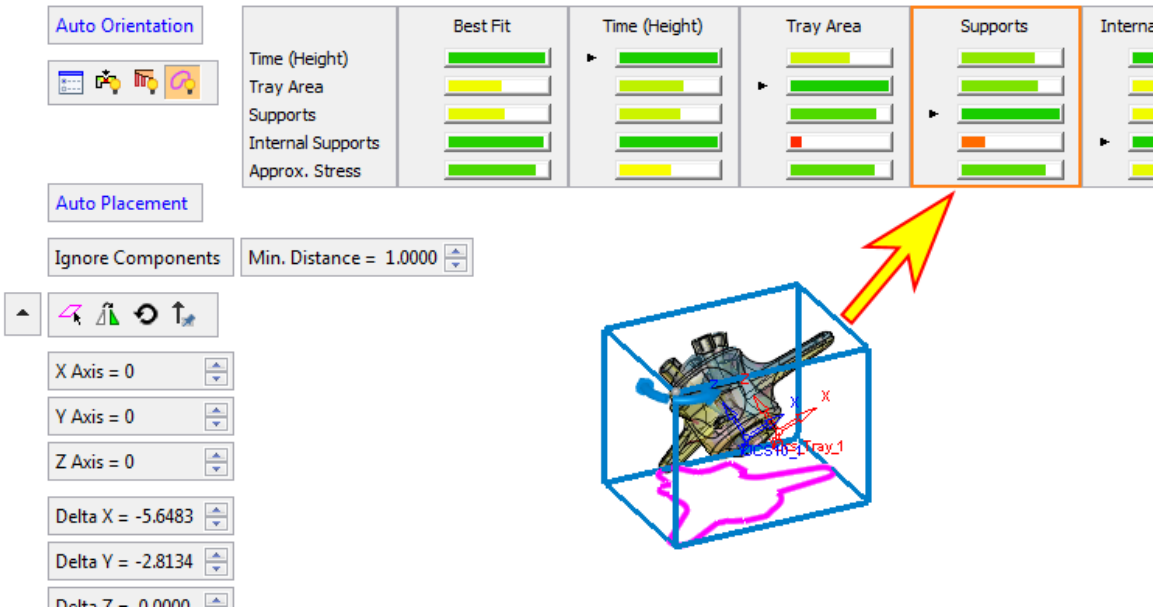


31. Now select the option **Time (Height)**. This results in the orientation based on the smallest height value, by which the part will be positioned (based again on the part's bounding box.).

The height of the bounding box is the smallest possible



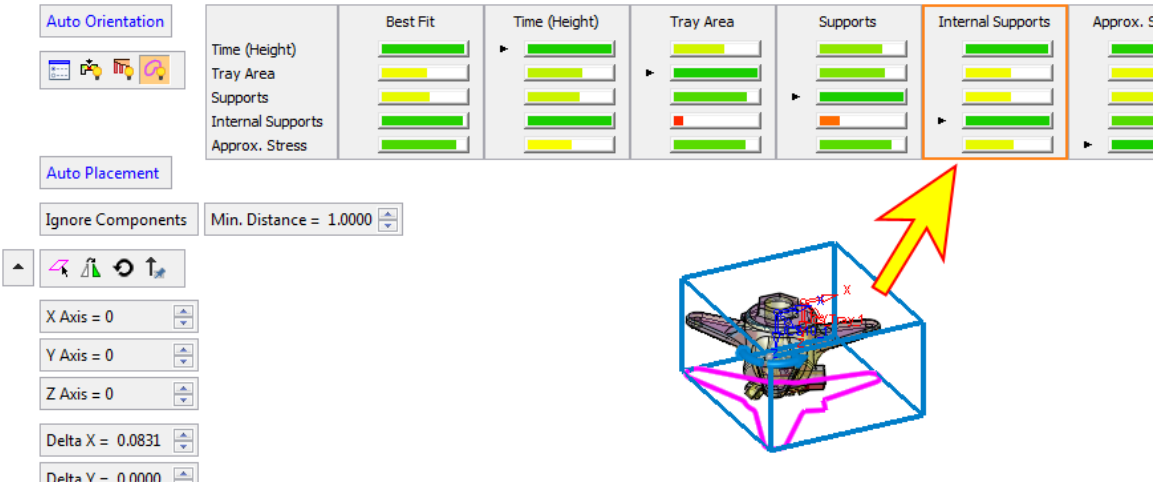
32. Select the option **Support**. This gives the orientation with the minimal support area.

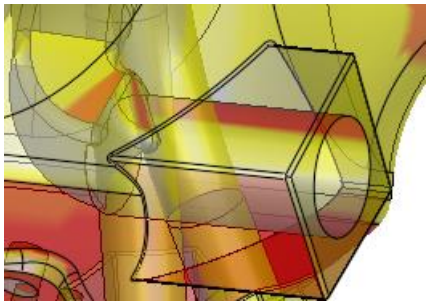


In this orientation, the amount of external red faces is small.

In all these positions so far, supports are required inside the pipes of this manifold part. Of course, these cannot be removed easily, definitely not by machining

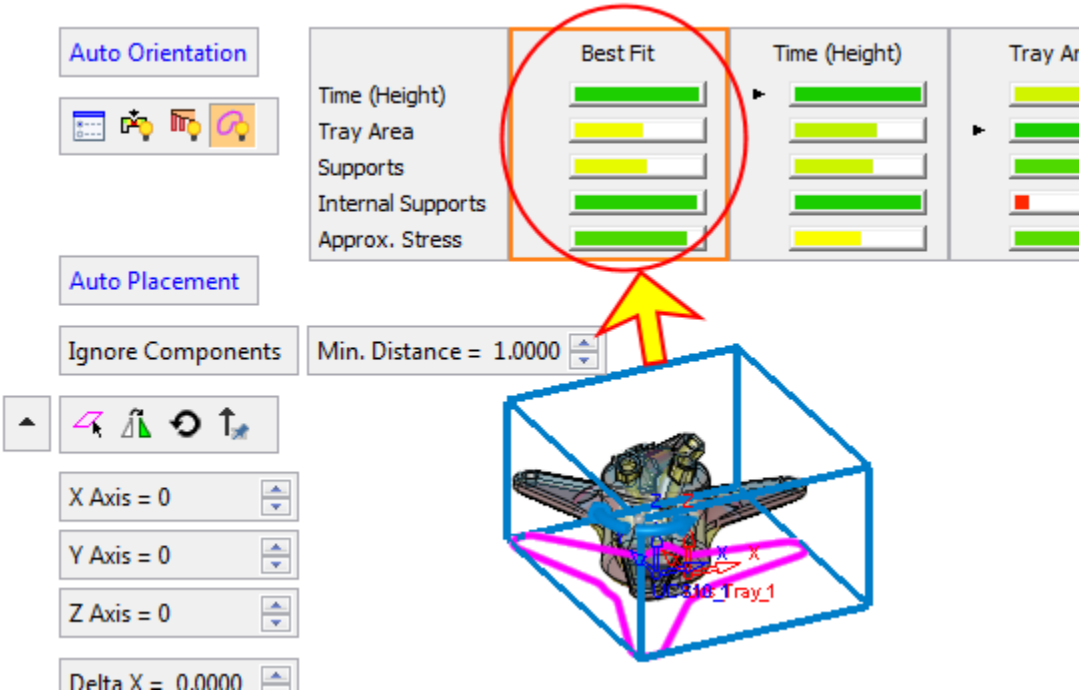
33. Therefore, select the option, **Internal Supports**. This orientation is based on the minimal support for internal open areas



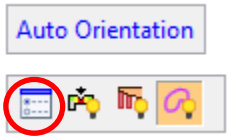


You can see that in this orientation the pipes in the part are mostly colored yellow, except for some small area at the bottom.

34. Press the option Best Fit again.



35. As mentioned earlier the **Best Fit** option displays an optimized orientation that **Best Fits** all the analysis criteria, based on the weight importance given to each of criteria in the **Orientation Settings dialog**, or the default weight values defined in the **Best Fit Positioning Preferences**;
36. Press the **Orientation Setting** icon.



The **Orientation Settings** dialog is displayed

This dialog enables you to set various presets before starting the **Auto Orientation analysis**.

Here you may set the display, value and color of the Position Angles.

The **up-facing angle** defines the angle above which the faces are considered as up-facing. Up-facing areas are those that are not considered as down-facing.

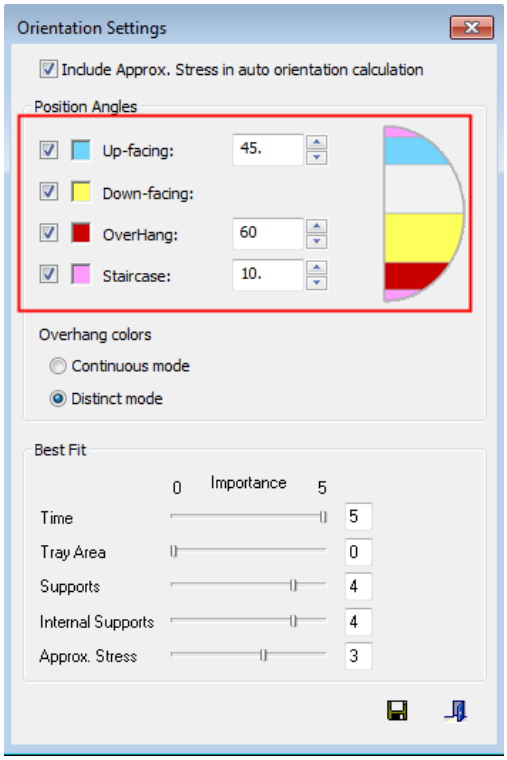
(The default angle depends on the material selected and is stored in the relevant Material.XML file)

A **down-facing surface** is a surface that is facing downwards, towards the build tray, given the selected build orientation

Overhang: This angle defines the degree of overhang after which some support structure should be added.

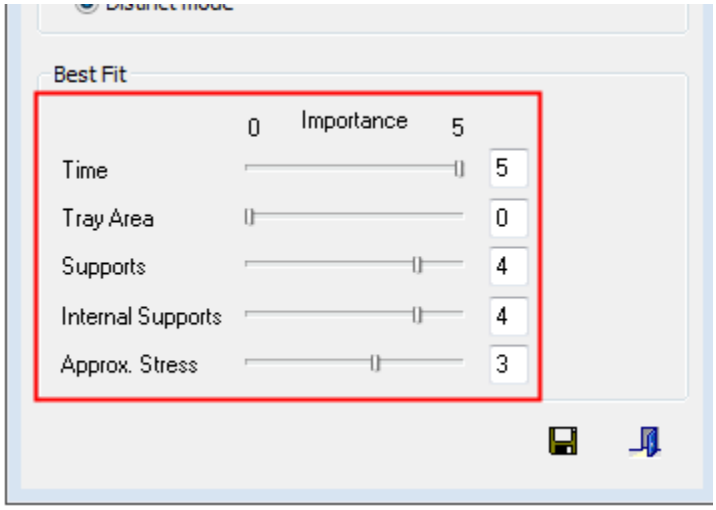
The Default Min. Overhang Angle is set in the relevant technology XML file and can be changed in the Edit Printer Parameters and Technology Settings dialogs.

The **Staircase** Effect is where edges or lines appear jagged, similar to staircase steps

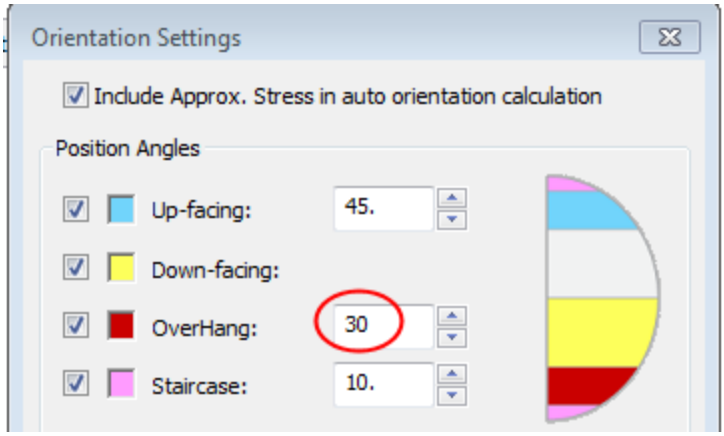


Set the level of importance (from 0 to 5) of the following criteria when calculating the best fit, where 0 = least important and 5 = most important. The importance weight can either be set by using a slider or an edit box. The default values are defined in the

Preferences, as will be explained in the next step. The number of orientations to be calculated to find the best fit is also defined in the same Preference.



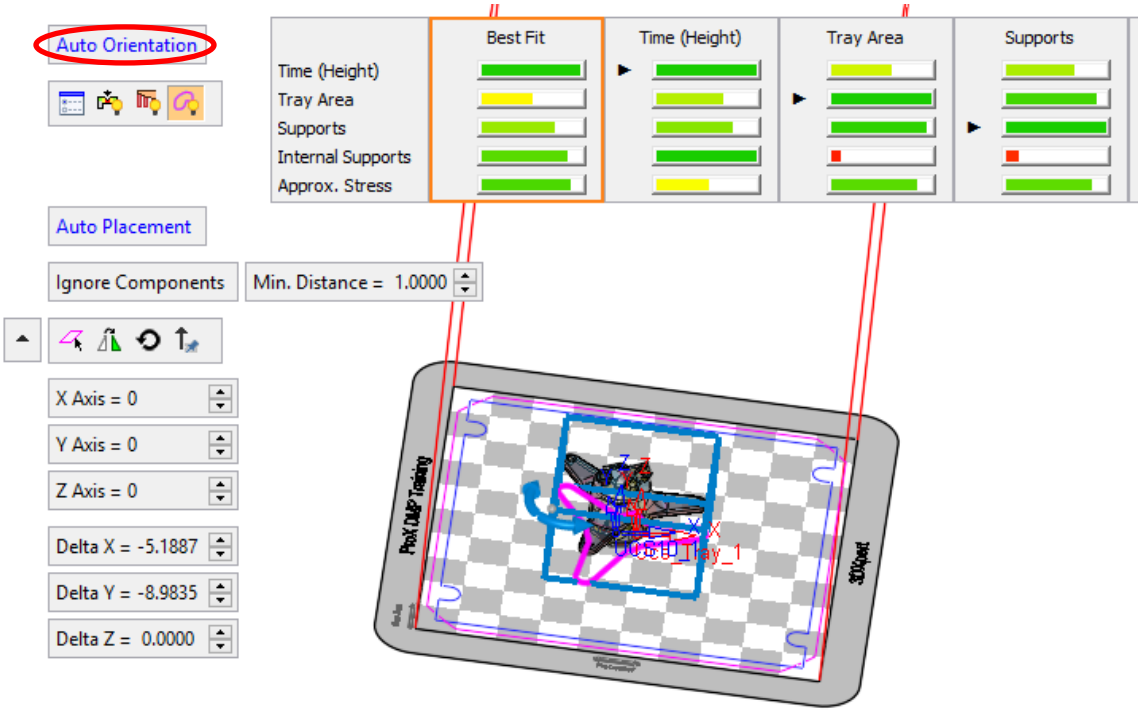
37. For this exercise, we will not change the Criteria.
38. Change the **OverHang** angle to **30**.



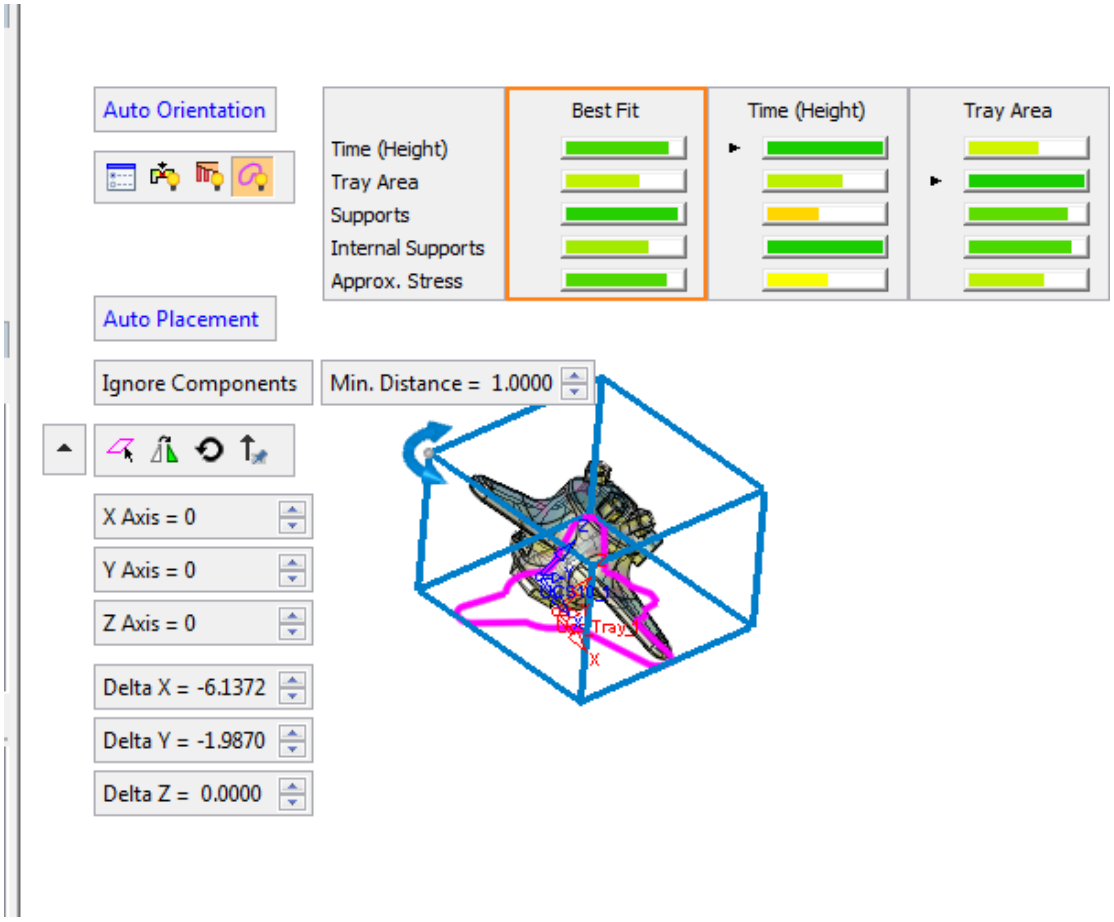
39. Exit the dialog.

40. Press the **Auto Orientation** button to recalculate the analysis. Notice the new orientation of the part according to Best Fit criteria.

Before re calculation:

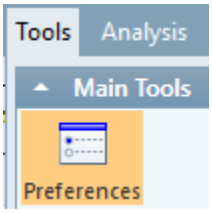


After re calculation using an **OverHang** angle of **30** :

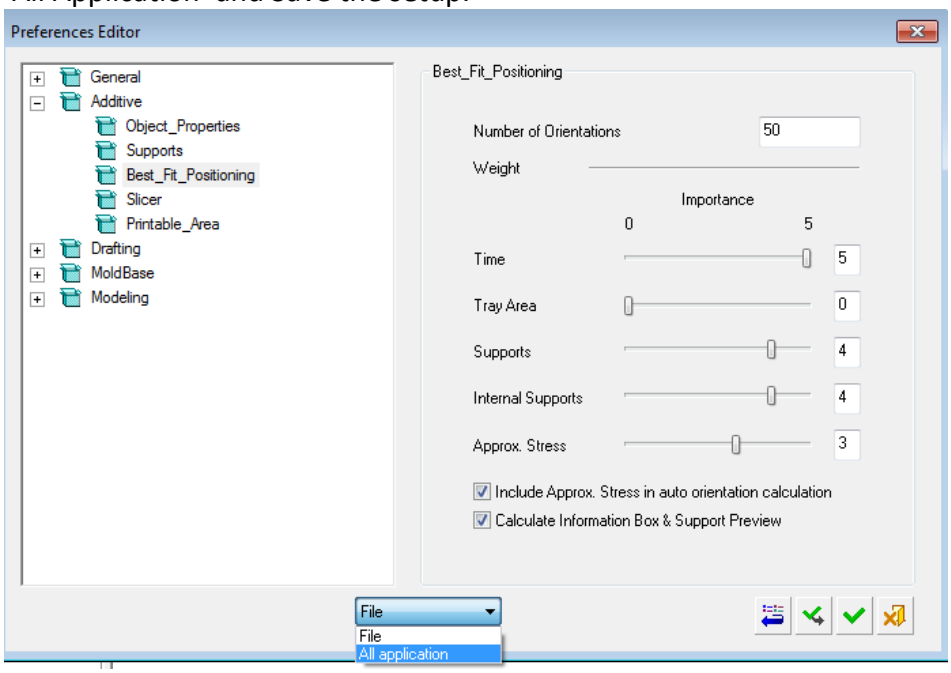


Setting up Best Fit criteria through the Preferences:

From the main menu, enter Tools - Preferences.

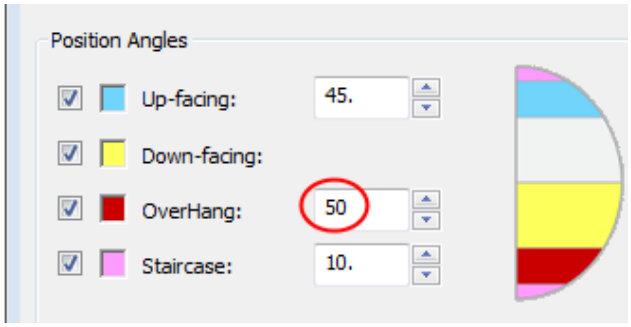


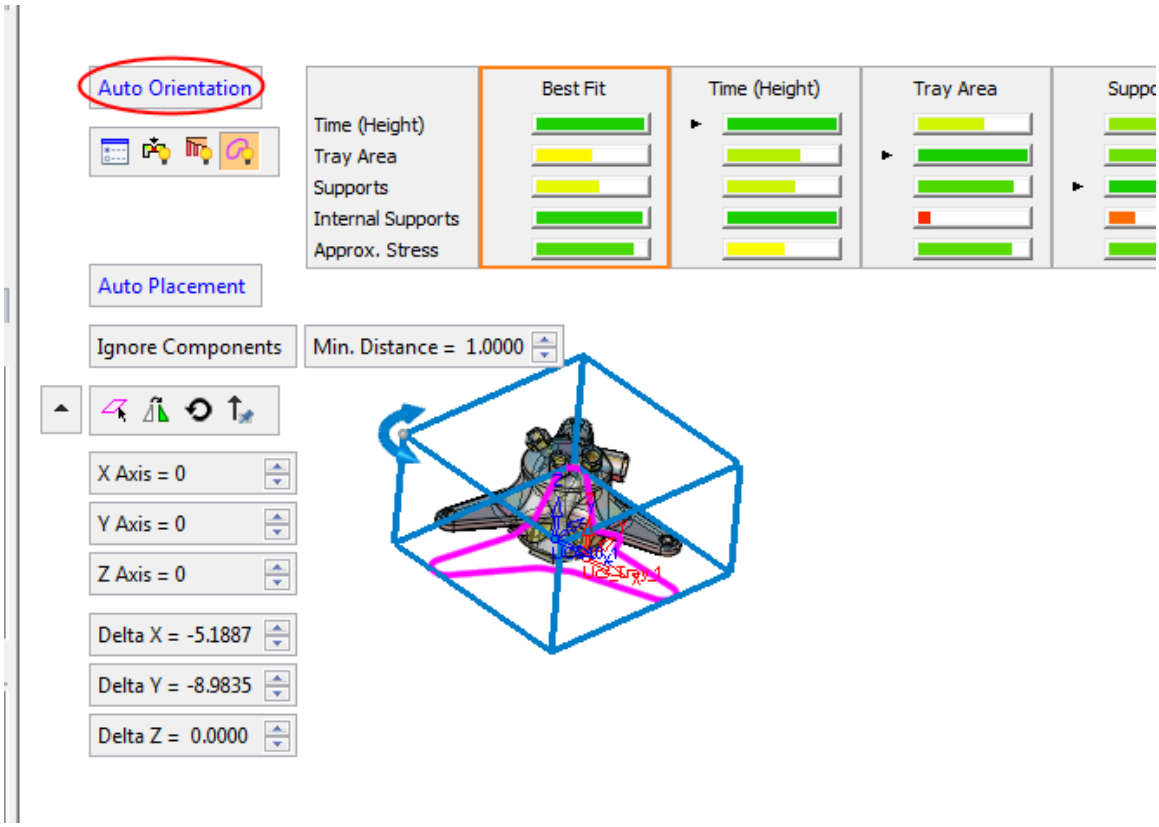
Browse to the Additive - Best_Fit_Positioning leaf.
Set the importance factor (0 = Not Important, 5 = very Important), switch from 'File' to 'All Application' and Save the setup.



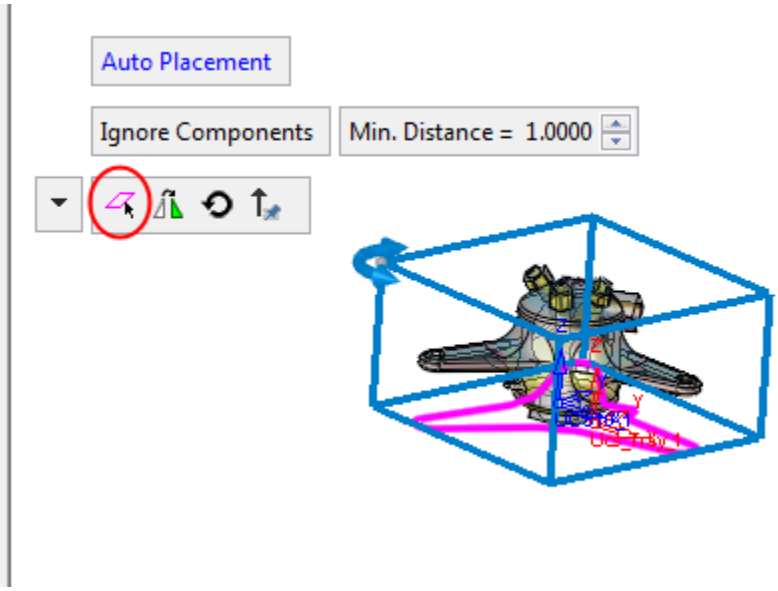
Enter Position Body, from the list select Best Fit, Press the Auto Orientation button and see the result.

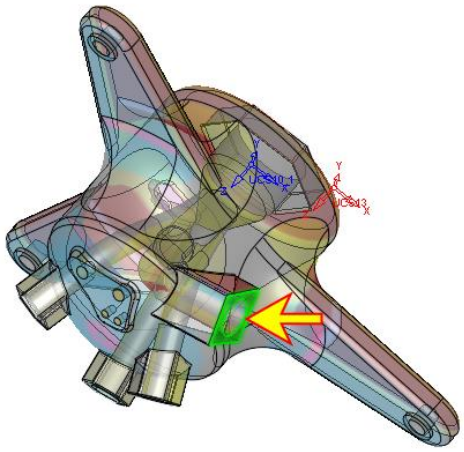
41. Change the **OverHang** angle of **50** and recalculate the analysis:

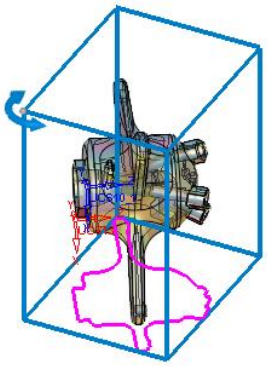




42. Sometimes you know how you want to place the part by placing one of its faces in parallel (or tangent) to the tray. Press the Select Plane button, and then pick the face highlighted in green :

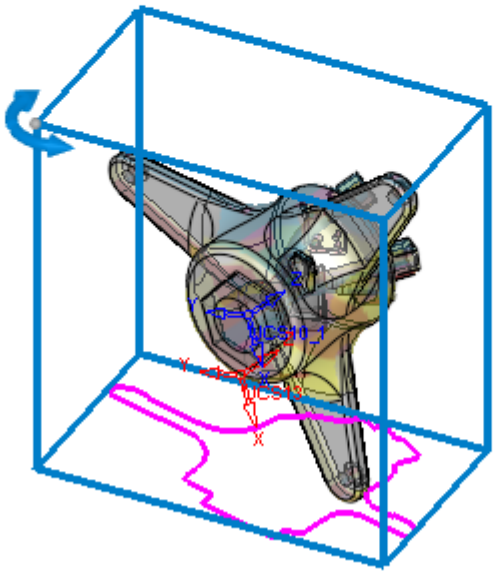




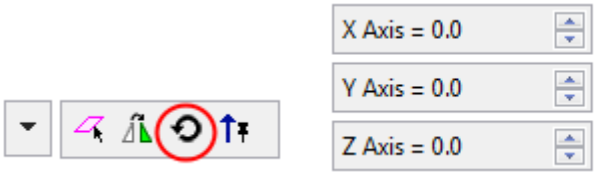


43. After placing the part with any of the above mentioned methods, you can fine tune the position by manually setting the rotation and position through the boxes.

44. Once you have manually rotated the part a new orientation named “Current” has been added to the analysis criteria options.

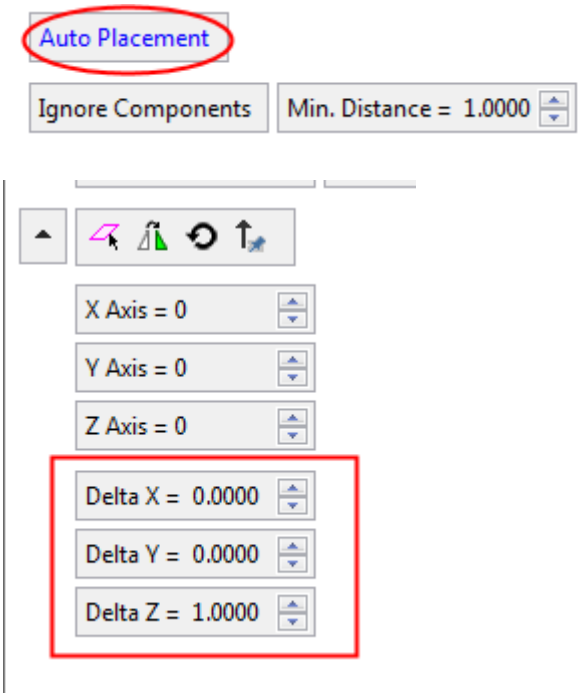


45. For this exercise, let’s make sure that the X,Y,Z axes are back to zero (i.e., press the Reset Orientation button).

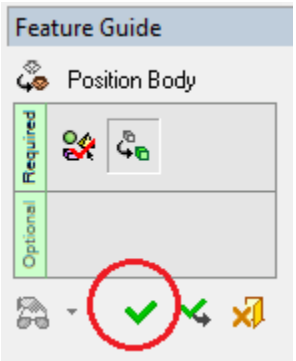


After Reset

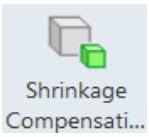
46. Click also Auto Placement to set Delta X Y & Z values back to zero: Set the Delta Z to 1.



47. Finally right mouse click on screen and press **OK** through the Feature Guide.



Note on Scaling (Shrinkage Compensation)



As the printed part cools down, it shrinks.

Therefore, it may be required to scale it up so that it shrinks to the correct size.

3DXpert's Shrinkage Compensation has various options, the primary methods are:

Uniform (same on X, Y & Z)

- By Percentage
- By Factor

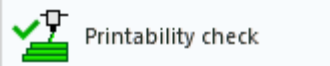
Non-Uniform (can be different on each direction):

- By Ratio. Enter scaling factor for each axis
- By Bounding Box. Enter X,Y,Z lengths of new bounding box

The default factor in Z is set to 1 because in printing, the shrinkage is not expected along the Z-Axis.
The part we use has already been scaled up, so we can continue with the next step.

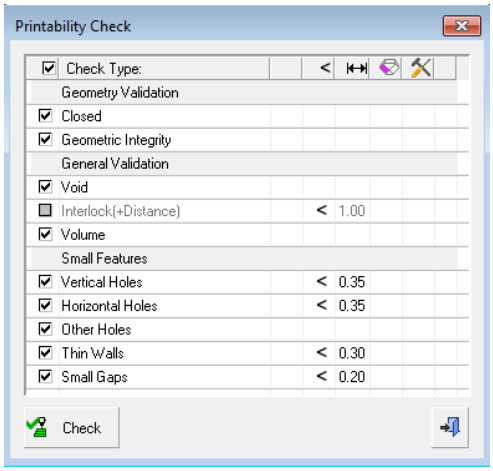
Part 3 – Analysis Tools

3DXpert offers a variety of Analysis tools, available from the 3DP Analysis menu



In this exercise, we will focus only on the **Printability Check**.

48. From the Guide Bar, click **3DP Analysis** and select **Printability Check**. Pick the part.
    The Printability Check interaction is launched, where you can use the boxes to select which of the checks you would like to perform.
    The checks are divided into 3 categories: Geometry Validation, general Validation and Small Features (in the model):



Geometry Validation

Closed – checks that the body is closed
(i.e., that it is a stitched solid body)
Geometric Integrity - that the body is geometrically OK (remember that we are
working with a CAD model)

General Validation

Void – that there are no areas,
where powder can be locked,
without the ability to remove it.
Interlock – checks for interference,
in case more than one part
is placed on the tray.
Volume – Ensure that all models are located within the tray limits
This check is not relevant to this exercise.

Small features

Vertical Holes – looks for vertical holes whose diameter is below the specified threshold
Horizontal Holes – looks for horizontal holes whose diameter is below the specified threshold

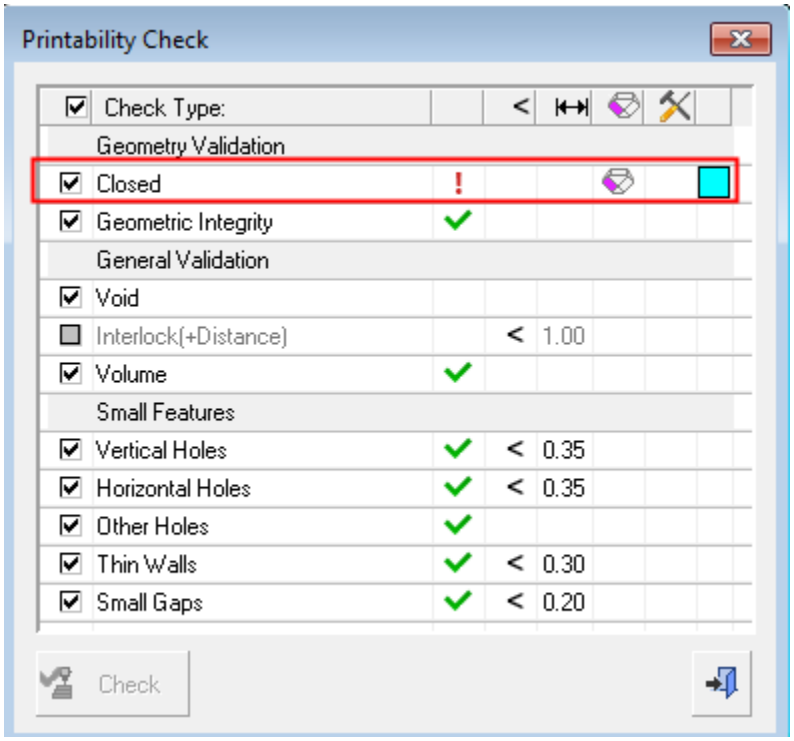
Other Holes – looks for other holes whose diameter is below the specified threshold

Thin Walls - looks for thin walls, whose width is below the specified threshold

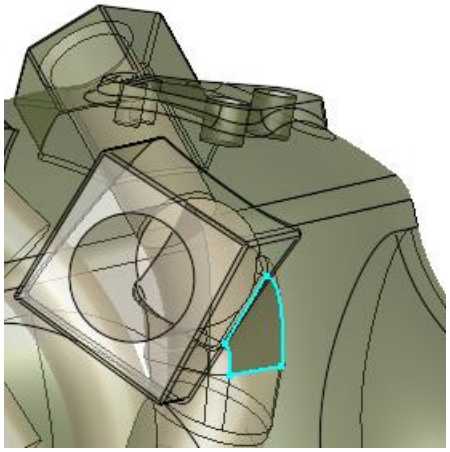
Small Gaps - looks for gaps, whose width is below the specified threshold

49. Press the **Check** button 

and as the results are displayed, see that there is one check that returns a warning - the model is marked as not closed.

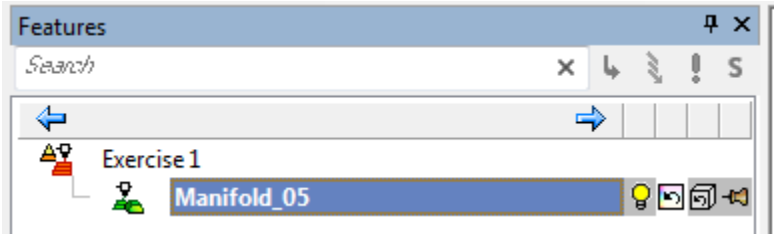


The system highlights the problem – a face that is not stitched to the body.

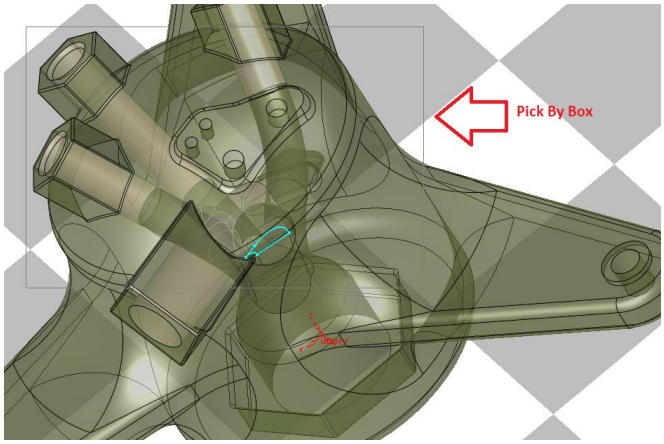


This face is the second object that is reported by the 3DP Objects Tree. We will now make this part a single object. As we are using a CAD system, we can use standard CAD tools to fix that.

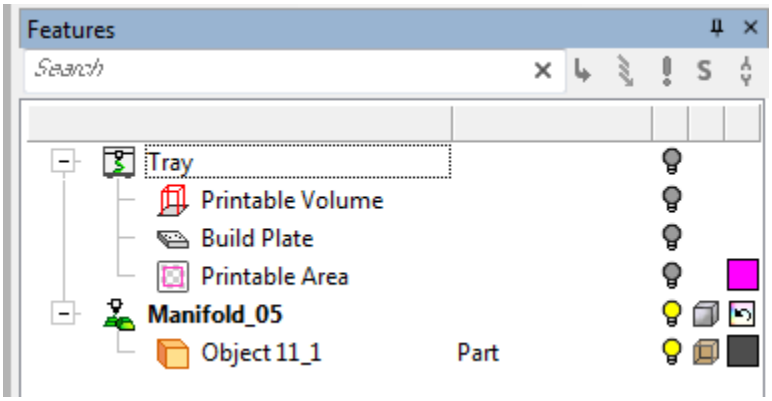
50. **Exit** Printability Check.
51. Activate the part by double clicking its name on the tree.



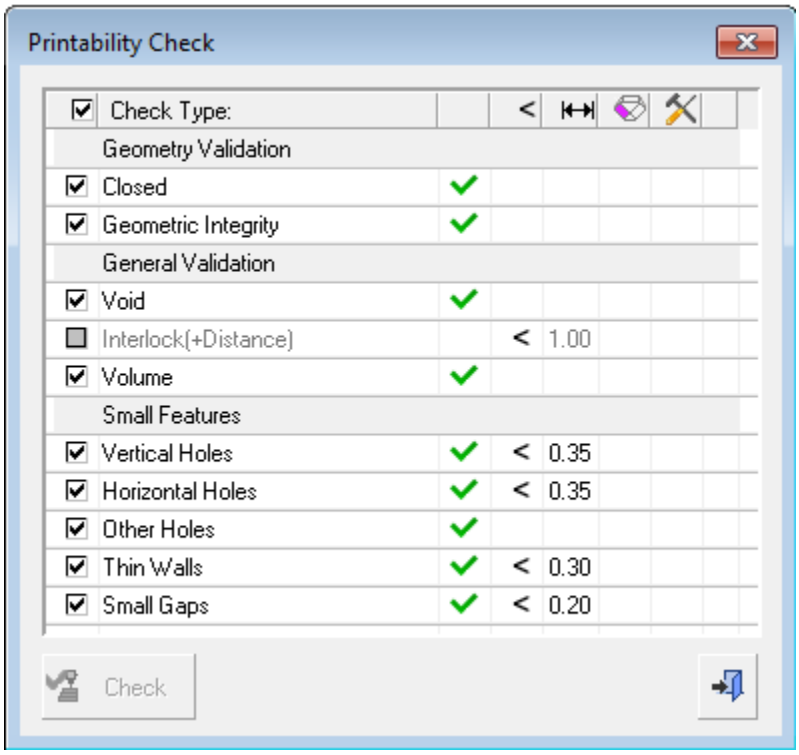
52. From the Menu Bar, click the Stitch function. 
53. Pick the object 'by Box' and on the Feature Guide press **OK**.



54. Click the 3DP Objects Tab and see that in addition to the build plate, the part now has a single object:



55. Re-Run the same analysis again. The result is OK, we have now a closed object.



56. Click the Close button to exit Printability Check.

End of Exercise