

Steps for Printing on the Dimension 768 SST:

1. General Knowledge

a. CatalystEX is a program created by Dimension to process your model for printing. It literally takes your model and slices it into .01" increments. This allows the printer to determine where it needs to print. When printing in plastic, we will have certain points in our print where undercuts in our model cause the printer to build temporary supports. These supports are printed with a material that will dissolve when set in a bath for an hour or two. Let's get started!

2. Open your file in CatalystEX

a. First, open Catalyst EX. It should be located on the desktop. Next we can open our .stl. When we open our file, it should appear in the modeling window on your screen. The model imports in whatever units you were using in your modeling program. If you need to scale your model, or even change the units from millimeters to inches, there are options to do so. We begin in the 'General' Tab.

3. 'General' Tab

a. CatalystEX features multiple tabs located across the top of its window. See Figure 1.15. The first tab we will investigate is the 'General' tab. This area provides you with a default settings and status bar along the top, but more importantly; Property Options along the right side of your screen.

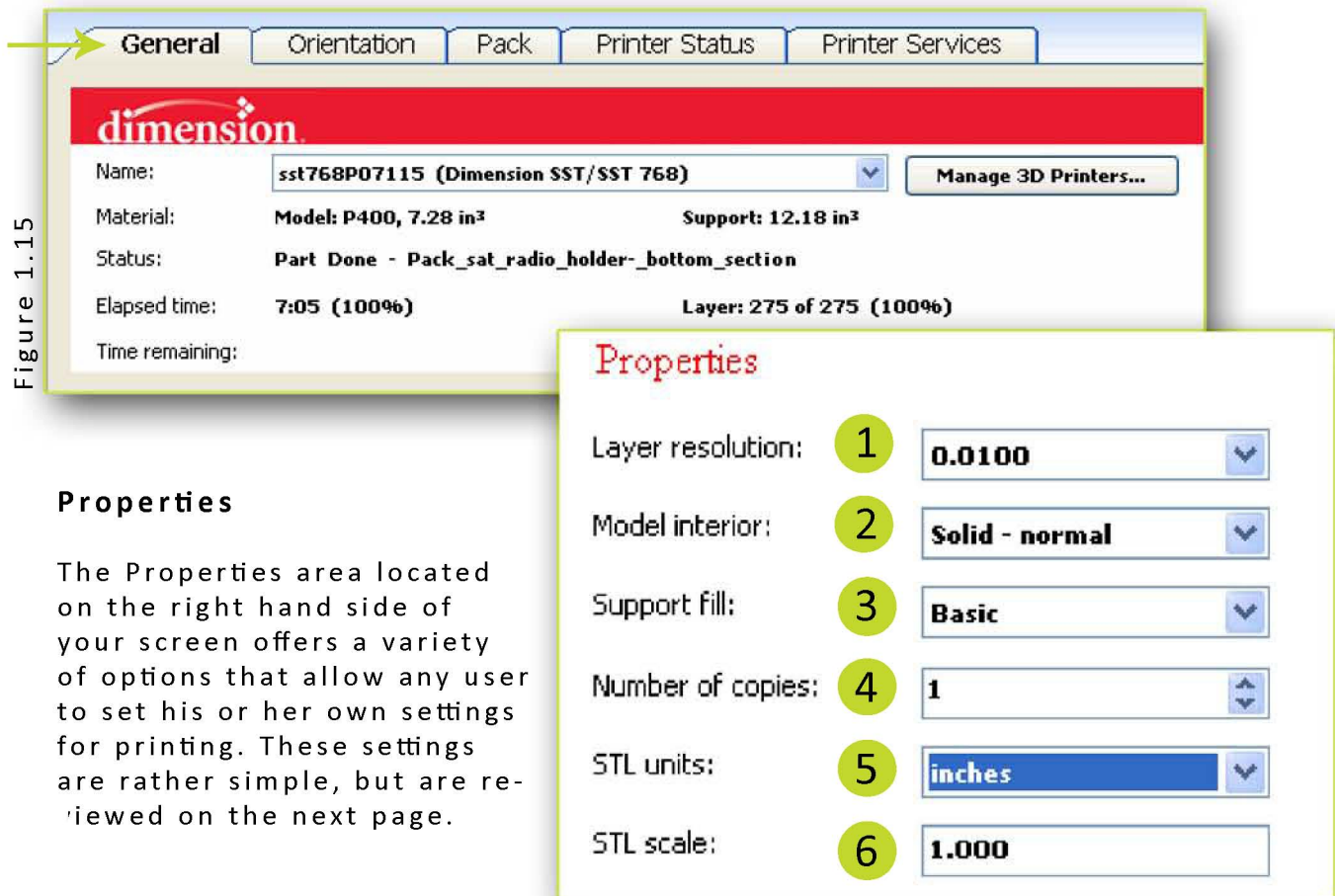


Figure 1.15

Properties

The Properties area located on the right hand side of your screen offers a variety of options that allow any user to set his or her own settings for printing. These settings are rather simple, but are reviewed on the next page.

Figure 1.16

Steps for Printing on the Dimension 768 SST:

3. 'General' Tab Cont.

1 Layer Resolution

'Layer Resolution' determines the height of each layer of material extruded to produce a part. Available resolutions are: .010 or .013 inch (.254 or .330 mm) Resolution will effect build time and surface finish. A shorter height creates a smoother finish, but will take longer to build. Only experience can help you make your decision.

2 Model Interior

'Model Interior' establishes the type of fill used for interior, solid areas of the part. The 'Solid - normal' option is used when a stronger, more durable part is desirable. Build times will be longer and more material will be used, thus creating higher costs. The 'Sparse' option creates an interior that will be "honeycombed/hatched". Sparse also allows for shorter build times, less material use. When printing surfaces or wall thicknesses smaller than .25", 'Sparse' interiors are not recommended.

3 Support fill

Support material is used to brace the model material during the build process. It is removed when the part is complete. Support fill options will effect the support strength and build time of the print. The 'Basic' option may be used for most parts. It provides more support than Sparse. Areas of the model that are considered to be of marginal strength will be supported. The 'Sparse' option minimizes the amount of support material. Sparse supports are accordion-pleated. This reduces structural support and is usually recommended for objects that aren't too complex and have flat bottoms. Lastly, the 'Surround' option surrounds the entire model by support material. Typically used for tall, thin (narrow) models (e.g., pencil). I would recommend laying your part down in the print area rather than printing tall and thin.

4 Number of Copies

Select the number of copies you want to Print or Add to Pack. The number of possible copies will be limited by the size of the modeling platform. The modeling platform for the Dimension 768 SST is 8"w x 8"l x 12"h.

5 STL Units

Select 'inches' or 'millimeters' units of measure for your STL file. STL files are created with the units of measure defined as inches or millimeters. This information is not passed to CatalystEX when it opens the STL file. You may need to set the appropriate units of measure for the file so that you may work with the part in the Model window, or to fit the model part within the build envelope.

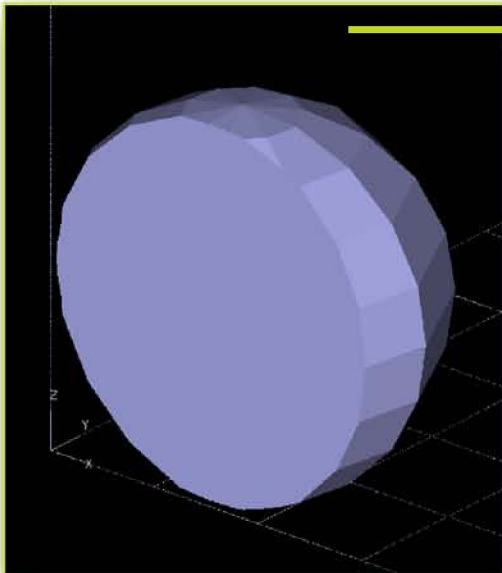
6 STL Scale

Before you process a part for printing, you can change the size of the part within the build envelope. Every part has a pre-defined size within the STL file. After you have opened the file you can change the size of the part produced from the STL file by changing the scale. The Scale always relates to the ORIGINAL (or "Save As") STL file size definition. For example: a cube that is defined by the STL file as being 2 X 2 X 2 can be built to 4 X 4 X 4. Simply change the scale to 2.0. If, after changing the scale to 2.0, you decide that a size of 3 X 3 X 3 would be preferred, change the scale to 1.5 - the scale relates to the original size of 2.0, NOT the resulting 4 from the first scale change.

Steps for Printing on the Dimension 768 SST:

4. 'Orientation' Tab

a. If we click on the tab labeled 'Orientation', to the right of our 'General' Tab, we find options on how to orient our part. Ideally, you should orient your part in your modeling program. However, because you orient it in the way you think best, does not mean it actually is the proper way it should be oriented. We revisit the idea once again, that a print should utilize the least amount of Model and Support Material possible, while maintaining job completion in a reasonable time. In this tab, CatalystEX provides you with tools that allow you to manipulate your model's orientation.



Modeling Window

By now you may have discovered that the modeling window taking up a majority of the screen space. This displays your part in its current orientation and size. The modeling window also provides you with the limits of the print area. Most importantly, the platform area, or the 'c-plane', displays 1"x1" squares. This allows you to get a general idea of how large your part is. In addition, the bottom left of the screen will show you the exact part size. Use your mouse to zoom in or out, as well as orbit.

STL Size (inches)	X: 3.13	Y: 1.90	Z: 3.17
-------------------	---------	---------	---------



Auto Orient

It can be tough and time consuming trying to find which orientation of your part will give you the best price and time. This is why CatalystEX has an 'Auto Orient' Tool. This tool will attempt to generate up to 3 different orientation options for you. If Catalyst discovers that one option is ideal for your part, then it will only present you with 'Option A'. You must click on the 'Auto Orient' button before you may view different options. With the current half-sphere model we are working with in this tutorial, its most ideal orientation happens to be with its flat side down. See Figure 1.17. If your 'Auto Orient' button doesn't work, we might have a problem with your model. Troubleshooting your model is explained in the first portion of this tutorial. For reasons on why your model maybe throwing errors in CatalystEX, please review the next page. Otherwise, continue onto page 9 of this Tutorial.

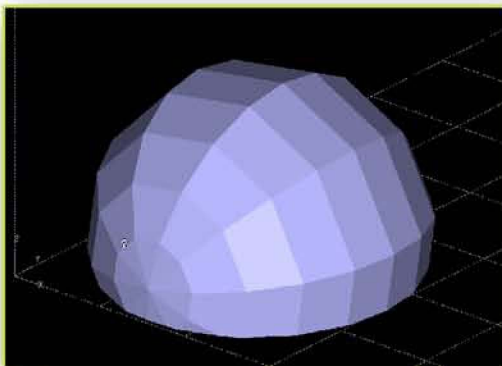


Figure 1.17

Steps for Printing on the Dimension 768 SST:

4. 'Orientation' Tab Cont.

Troubleshooting

Reasons for your model not processing have to do with either missing facets on your part, or naked edges. If your part will not process with the 'Auto Orient' button, you should see a message at the very bottom of your screen that looks like Figure 1.18. We can pull up an '.stl report' to let us know what the problem is by selecting 'Tools' from the Catalyst Menu bar at the top of the screen. Next, by clicking on 'STL Report'; we open a new window that describes the .stl's information. At the bottom, it will show you 'Error Conditions'. See Figure 1.19

Error: STL has missing facets or reversed normals.

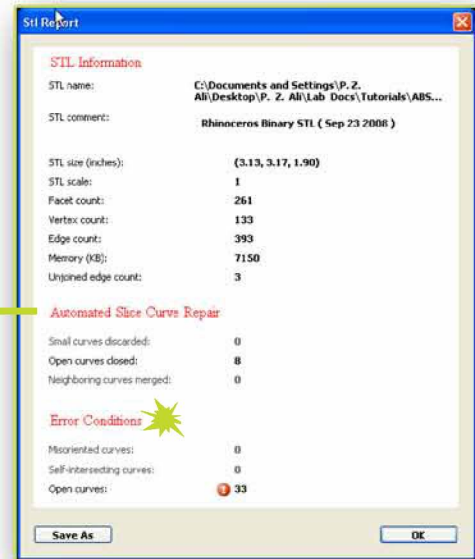


Figure 1.19

Automated Slice Curve Repair

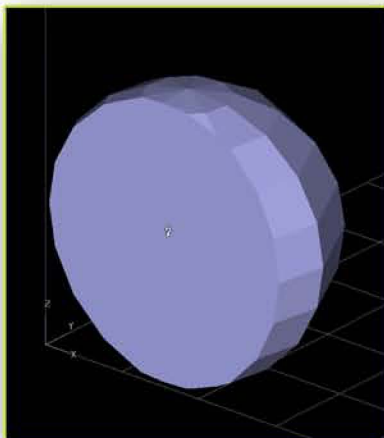
CatalystEX will automatically bridge gaps between naked edges. This only happens when the gap between naked edges are very small. If they are large, Catalyst can't determine the gap fix. Leaving messy tool paths that cannot be printed. When this happens, and we press the "Auto Orient" button, the part will not process. However, a good way to find out where your problem lies, is by processing the part, an looking at the tool paths. See the following figures.

Orient Selected Surface:

- Bottom
- Top
- Front

Orient Selected Surface

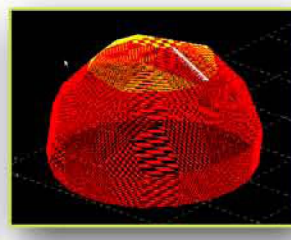
By using the 'Orient Selected Surface' Command, we can select a surface on our part, and distinguish it as either; bottom, top, or front. This allows us to orient our piece, (rather than "Auto Orient") even with reversed normals, or missing facets. By doing this, we can now hit 'Process STL' at the bottom of the screen, and see where our missing facets or curves may be located. If we look at the screens hots below, we can see we have Oriented the half sphere with its flat side as the bottom; when we process the .stl, the problematic curves and tool paths appear in yellow.



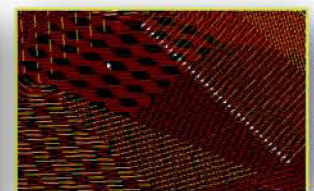
'Orient Selected Surface' Command



STL appears to have a missing facet



Problematic Tool paths highlighted in yellow.



Problematic Curves up close.

Steps for Printing on the Dimension 768 SST:

4. 'Orientation' Tab Cont.

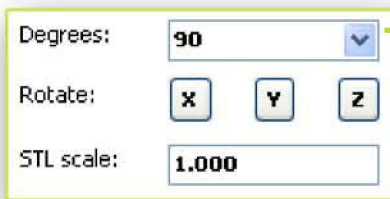
Other Orientation Options

If you choose to try out other orientation options, you may use the 'Orient By Selected Surface' (Figure 1.21) Command, or the 'Degree Rotation' Option Bar. (Figure 1.22)



Orient Selected Surface (Figure 1.21)

This option allows you to select a surface on your model and assign it as the Bottom, Top or Front.



Degree Rotation Options (Figure 1.22)

If you decide to rotate your part for whatever reason by Degrees, this option is available. Simply Select the degree of rotation, and choose which axis to rotate your part on.



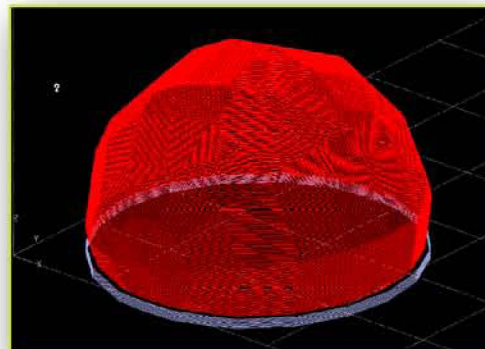
STL Orientation and View Bar

If you would like to see your part in different views, this bar provides you with the option to do so. Additionally, we can undo any orientation changes we have made by selecting the 'Undo STL Orientation' Button. If you would like to see your part in its original imported orientation, click the 'Restore STL Orientation' Button.

Process STL



Once we have our STL object properly oriented, we can hit the 'Process STL' button at the bottom of your screen. This will determine your part's tool paths, and where support is needed. Any ABS printed will appear as red or green, any support printed will appear in a light gray color.

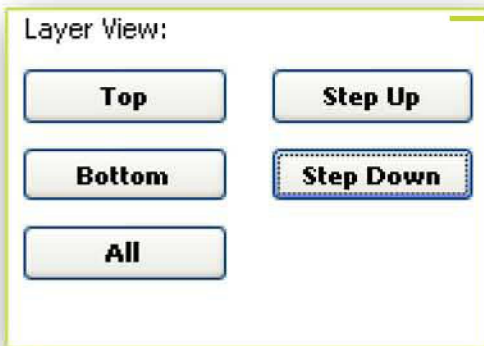
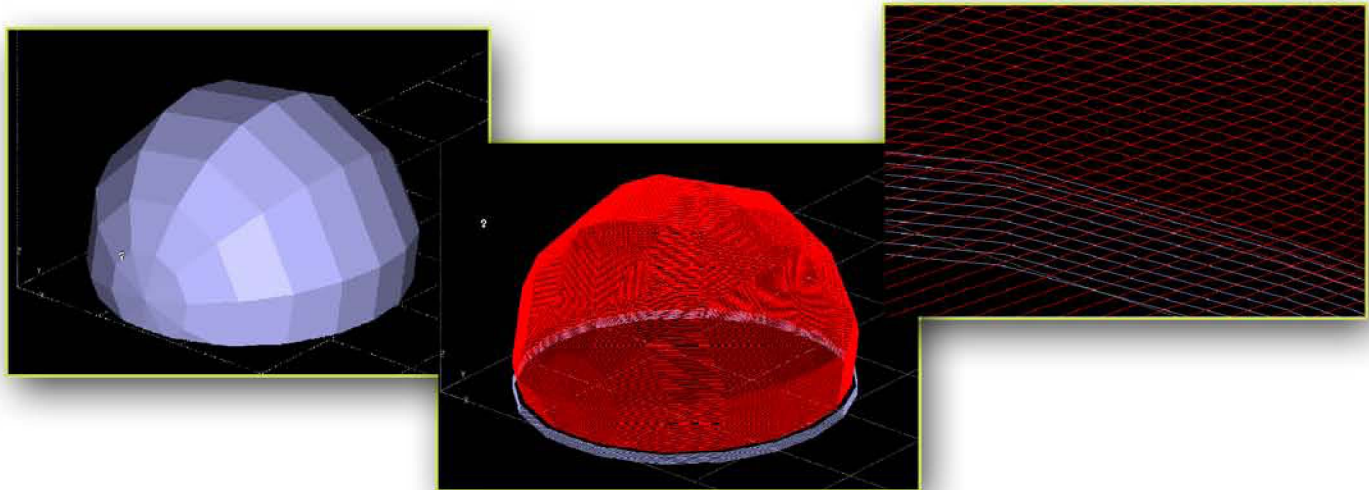


Steps for Printing on the Dimension 768 SST:

4. 'Orientation' Tab Cont.

Understanding Toolpaths

Where you may look at the big red blob on your screen as useless; if we were to zoom in on this object, we can see much more. After processing our STL, Catalyst generated tool paths in which it follows to extrude ABS and Support material in certain locations on our tray.



Layer View

After Processing, we can view each layer of the print process by utilizing the 'Layer View' Bar now viewable in the top right of your screen. If we look at Figure 1.23 and 1.24, we see two different types of tool paths. Figure 1.23 shows us a print layer with 'Sparse' selected as our model interior option. (Less Material!) Figure 1.24 displays our 'Solid' interior option. We can visibly see the difference.

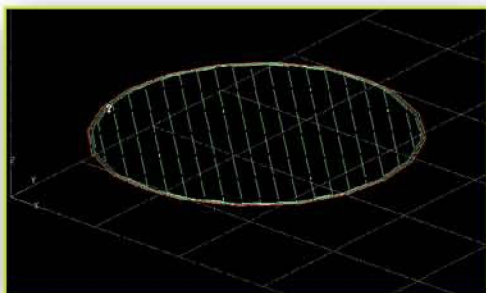


Figure 1.23
With 'Sparse' Selected as our 'Model Interior' option from the 'General' Tab; we can see it will be using less material.

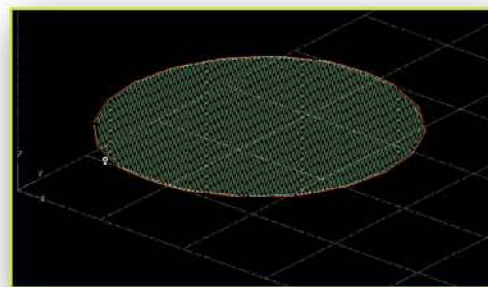


Figure 1.24
With 'Solid' Selected as our 'Model Interior' option from the 'General' Tab; we can see many more tool paths-

Steps for Printing on the Dimension 768 SST:

5. 'Pack' Tab

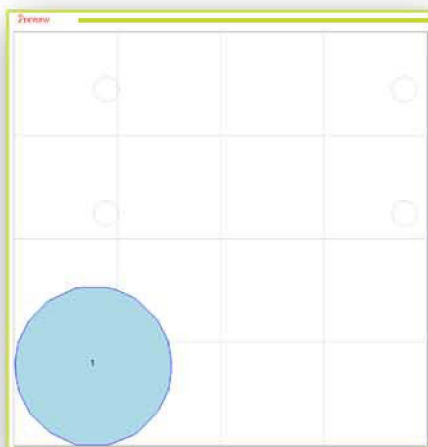
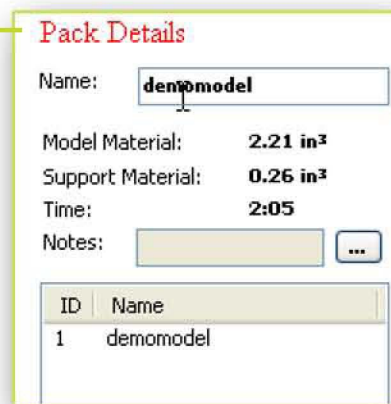
Add to Pack



Once we are confident that we have achieved a desirable orientation and set our necessary print options; we can click 'Add to Pack' at the bottom of your screen. We may now continue to the next tab labeled, 'Pack'.

Pack Details

The 'Pack Details' on the far right hand side of your screen will display your model name (as imported) the amount of model material it will use, (cu. in.) the amount of support material you will use, (cu. in) the estimated time in which it will finish, (accurate) any notes you would like to add to the print job, and lastly, the name of the model added to the pack, and number i.d. assigned to it. You can now add the Model and Support totals, and multiply it by the Lab's pricing/ cu. in., and discover your total price/print.

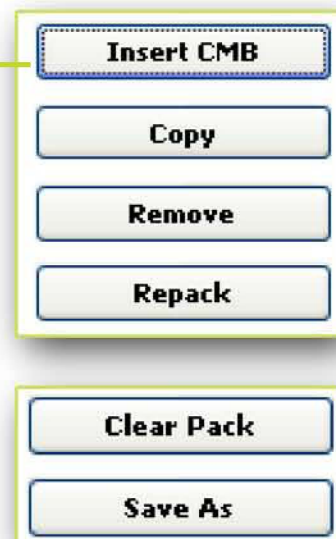


Pack Preview

The 'Pack Preview' window has now taken place of your modeling window. This Preview is a top view of what will be printed. Your 'Pack' is your print setup for the tray that we will insert into the machine for printing. for Options on how to change your pack, please review the following paragraphs.

Pack Options

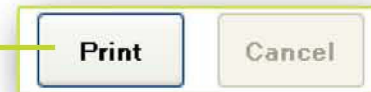
In order of importance: There are a few other options for your Pack. It is first important that you save any Pack that you are finished with and ready to print. We can do this by clicking the 'Save As' button. This will save your file as a .cmb file type. This directly corresponds with the 'Insert CMB' button. We can now insert your pack into Catalyst later on. If you need to, you can click 'Clear Pack' to remove any parts you've added. You may also remove them one by one with the 'Remove' button. If you decide to use the 'Copy' Tool, or even add different parts to the pack, you may want to hit the 'Repack' button, to arrange all your parts inside of the Pack in a way that Catalyst determines is best.



Steps for Printing on the Dimension 768 SST:

6. Print

Print



Congratulations on completing this tutorial. You should by now understand how to edit your model with Rhinoceros, export in .stl format, take it into Catalyst EX and set up for printing. Please read the 3D Printing Policies and Procedures to discover how you might drop your file for printing and printing pricing. The last step is to hit print, but you must fill out a form for 3d printing before hand. Make sure you save as a .CMB file, and drop it on the dFab Server!

Notes