Using CalcPlot3D to Create STL files for 3D Printing

Paul Seeburger – Monroe Community College   **pseeburger@monroecc.edu**

CalcPlot3D can save a surface or a thickened curve to an STL file. This STL file can then be used to print a 3D surface (or curve) model using a 3D printer. This includes solids of revolution and shells or washers.

3D-Printing Surfaces

In order for the STL file to work well with a 3D printer, the surface must have sufficient thickness (or must already be a closed region, like a sphere or a parametric surface knot). Most surfaces and curves do not have inherent thickness and are not composed of the kind of closed region needed to 3D print an object. To address this issue, CalcPlot3D can automatically generate the thickness you specify for a surface before creating the STL file.

1. Open CalcPlot3D from the link on the project webpage: https://sites.monroecc.edu/multivariablecalculus/.
2. Graph a surface in CalcPlot3D (using the **Function**, **Implicit Surface**, or **Parametric Surface** option on the **Add to graph** drop-down menu).
3. For nicer 3D-printed surfaces, I recommend adjusting the **Number of Gridlines** for a **Function** of two variables to about 80, but you can test this feature out with the default number of gridlines.
4. From the **File** menu, select **Create an STL file from the plot**. A dialog will appear like the one to the right.
5. Enter the name you wish to give to the solid. This will be the generated name of the STL file when you save it.
6. Next enter the approximate size at which you expect to print it and the desired relative thickness. These will be used together with the dimensions of the 3D plot window to determine the approximate relative thickness to use with the surface. You may need to experiment a little with these values and with your 3D printer software to get the thickness just right.
7. If the surface you wish to print is already a closed 3D surface like a sphere or a torus, you can specify that no thickness be given to the surface before printing, or you can still use a thickness, depending on how you desire the printer to fill the void inside the closed surface.

> **STL File Creation Options:**
>
> Solid name: `mySolid`
>
> *xy*-dimensions of the printed solid:
> `4` in × `4` in
>
> Printed solid thickness: `0.0625` in
>
> **Create surface thickness to make the surface solid?**
> ● Yes ○ No
> (Thickness should always be created for non-solids.)
>
> **Direction to move to create thickness:**
> ● Opposite Normals
> ○ Normal Direction
> ○ Half-and-half
> ○ z-Shift Down
>
> **File type:**
> ● Binary file ○ ASCII file
>
> Scale factor (in. to mm): `25.4`
>
> [View Solid] [Create File] [Cancel]

If you choose to give a surface a thickness, you will need to choose between using the **Opposite Normals** direction (generally inside a closed solid or below an ordinary surface), the **Normal Direction** (generally outside a closed surface or above an ordinary surface), or the **Half-and-Half** option, which goes both directions half the thickness and creates two new shells with the same basic shape. A closed surface cannot use the **z-Shift Down** option, but it can be a good option for

ordinary surfaces, as long as they are not too steep (where it may yield very thin walls). It will preserve the surface shape on both top and bottom. This is not true for the other options that keep the walls equally thick, but deform the surface to some degree in the selected normal direction(s). The **Half-and-Half** option is the best to choose for surfaces that flip orientation in the middle of the surface like a Möbius Strip or a Klein Bottle.

As mentioned above, for ordinary surfaces given by a function of two variables, like the graph of $z = x^2 + y^2$, the **z-shift down** option is nice since it preserves the exact shape of the surface on the bottom as well as the top. But you need to compensate for the fact that the thickness will vary over the surface and be least thick where the surface is steepest in the *z*-direction.

Advanced tip: If you'd like to see the surface normal vectors displayed for every triangular face on the surface, you can click on the plot and then type the **n**-key. This may be helpful to check, as it's generally required that the normal vectors of all faces in an STL file point outwards. Sometimes, for example, you may need to adjust the parameterization of a parameterized surface so its normal vectors point outwards rather than inwards. Note that if you wish to view the normal vectors on the thickened solid generated for the STL file, you'll need to have them visible before clicking the **View Solid** button.

8. Click the **View Solid** button to see what the surface/solid will look like before saving the STL file. Note that it can take a while to render, during which the cursor will show the app is busy (until you move the cursor off the app or it completes the task and displays the solid region). The more faces you have in the plot (for a more accurate printed solid), the longer this process will take.

9. Before creating the STL file, note that you can select either a **Binary STL file** (the default) or an **ASCII STL file.** The benefit of the Binary STL file is that it is much smaller than the ASCII STL file (about 1/5 the size). The benefit of the ASCII STL file is that you can actually read it to view the normal vector and vertices for each face.

10. Note that there is also a scale factor that you may wish to adjust. The default scale factor of 25.4 essentially assumes that the units in the plot are inches and converts the coordinates from inches to millimeters. For a plot over -2 to 2 in the *x*- and *y*-directions, this should correspond fairly well to a 4 inch by 4 inch object in Cura (or another slicing application). We can scale the object further in that application, but this scale factor is important to understand so you can generate STL files with a reasonable size for printing.

11. Once you are ready, click on the **Create File** button. You should see the fill downloading to your default download folder (in Chrome) or you may need to choose where to download it in some browsers.

12. Once you have created the file, you will need to check it out in the 3D printer software (slicer) you plan to use (Cura, for example).   Most computers now have a default STL file viewer that you can use to check the resulting object. On Macs, I use **Preview**. It does a great job of giving a clear view of the 3D object, so you can check for missing closure or other issues. On Windows, STL files can be viewed in **Paint 3D** or **3D Viewer**. I prefer **Paint 3D**. You may have other options.

13. Now you should be ready to print to the 3D printer using the 3D printer's slicing software. There is still some room to play with the surface in this software, scaling it appropriately and choosing how you want to fill the surface and how much support material to use. Ask your local experts (and YouTube, online tutorials, etc.) to help you with this step. By the way, it can take a long time to print one of these surfaces, depending on the solid's size and the printer and material being used to print it. My 4 in. x 5 in. mountain surface took over 8 hours to print. My 6 in. by 5 in. surface took over 14 hours (It's taller). But 3D printers are getting faster (and less expensive) all the time.

If you'd like to share your favorite 3D print tested STL files along with an image of the solid/surface, please send them to me at pseeburger@monroecc.edu.

3D-Printing Curves

1. To 3D print a curve, just plot the curve in CalcPlot3D (using the **Space Curve** option on the **Add to graph** drop-down menu).
2. Now select the option to **Create an STL file from the plot** from the **File** menu. The STL File Creation dialog will appear.
3. Typically, you will want to select **No** for **Create surface thickness** for a curve.
4. Click the **View Solid** button. A solid form of the curve will be displayed, encasing the curve in a tubular sheath. Default values are used for the radius and resolution of the tubes. See below for more information on adjusting parameters of the tube sheath for 3D-printing space curves.
5. Click **Create File** to download the STL file.
6. Note that working out how to 3D print curves often requires some careful attention to support structures when previewing the curve in the 3D printer slicing software. Only 2D curves that can be printed directly on the build plate will not require support.

Adjusting Parameters for 3D-Printing Space Curves

If you click on the settings button (⚙) for the curve object itself, you can adjust a variety of parameters that affect how the tube sheath looks for the curve. Going to the bottom of this Settings dialog, you will see the following options, labeled **Tube Sheath Settings**:

Selecting the first checkbox will display the tube sheath for the curve in the 3D plot, without even going to the STL File dialog.

The next two options allow you to adjust the radius of the tube and the number of circular steps used to make the sheath look round. The default values will work, but you may wish to adjust these. These are the values that will be used when you create an STL file containing this curve.

Next, we can choose to **Optimize joint angles**. This means that the number of sections/joints will be dynamically adjusted at sharp curves in order to get a better rendering of the curve. Experiment with the joint angle tolerance for some curves with sharp turns, and you'll see how nicely it adjusts the rendering.

You can see in this figure how there are more segments taken at the sharp corners than on the portions that are not as sharply curving.

By default the actual curve is hidden when its 3D tube sheath is rendered, but you can deselect the checkbox labeled **Hide Actual Curve**, to render the actual curve inside the tubes.

By default, CalcPlot3D will also close the ends of any given curve that's not already closed. But if you want to create a hollow tube around a curve (like a thickened straw), you can deselect **Close ends** and then select **Yes** in the STL File dialog to create surface thickness.
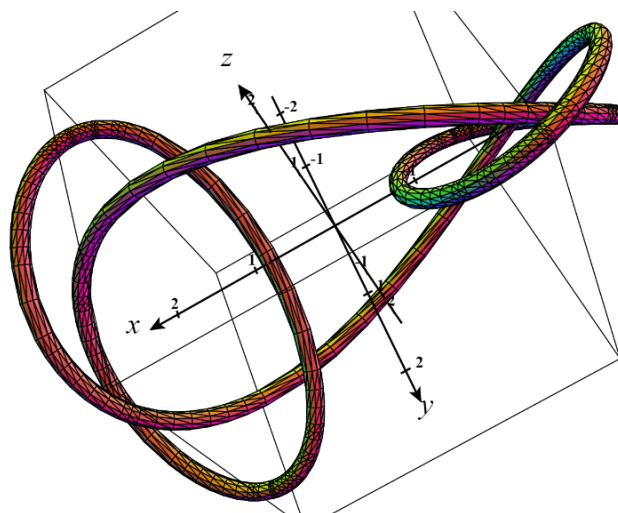
3D-Printing Contours

At this time, there are three different methods of printing contours plots for surfaces. If you have access to a 3D printer that can print in two different colors at the same time, you can 3D print the contours embedded in the surface in a different color, either as bands of equal width or as tubes. Or, whether you have a two-color 3D printer or not, you can 3D print the contours separately from the surface (flat on the build plate), creating a puzzle to have students place the contours correctly on the surface.

Method 1: A contour plot can be 3D printed as bands of equal width on the surface.

Contour Plot Options:
○ z-Layers (variable thickness for bands)
○ Wrap each contour with a tube
● Bands of equal width on the surface

Uniform width of bands on the surface:
0.05

1. Use CalcPlot3D to create the contour plot of a surface you want to 3D print and be sure to click on the 3D plot window to display the contours on the 3D plot of the surface.
2. On the **File** menu, select the option to **Create an STL file from the plot**. The STL File Creation dialog will appear.
3. Because you have a contour plot visible, an additional section will be visible in this dialog labeled **Contour Plot Options**. Select the option titled, **Bands of equal width on the surface**. This happens to be the default option.
4. Enter a width for the contour bands (in units from the grid). The default of 0.05 units works well for a typical surface that is 4 x 4 units or so (from the axes in CalcPlot3D).
5. Select **Yes** to create surface thickness and select the desired surface thickness above the **Contour Plot Options** box.
6. I recommend the **Opposite Normals** option, especially in this use case, as this causes the top of the surface to 3D print most accurately. This is also where the contours will be most accurate.
7. Now click the **View Solid** button. A dialog will appear to ask if you want to **View Contours Only** or **View Surface with Contours Bands Missing**. You can select either one. We'll need to come back and select the other one later. For the purposes of this tutorial, let's select the second option to **View Surface with Contours Bands Missing.**
8. A waiting cursor should show you the app is working on your request for a while. Note that this step is much faster for Method 1 than it is for Method 2.
9. If you are happy with what you see, click the **Create File** button to save the surface (with bands missing) to an STL file.
10. Now, without making any other changes, click the **View Solid** button again and this time select **View Contours Only**. Once the contours are displayed, click the **Create File** button to save these contours to an STL file.
11. You should now have two STL files that can be used to 3D print the surface with its contours in a different color. For instructions on how to 3D print this, see the subsection below titled **Combining Contours with the Corresponding Surface in Cura.**

Method 2: The surface can be 3D printed with its contours embedded in it as tubes (like the curves described in the previous section). These tubes can be made to extend only out of the top of the surface or to extend out from both the top and bottom. We'll explain how to accomplish these two options below.

1. As in Method 1, first use CalcPlot3D to create the contour plot of a surface you want to 3D print and click on the 3D plot window to display the contours on the 3D plot of the surface.
2. This method takes quite a bit more time to render than Method 1. I recommend that you decrease the number of contours and/or the **# of grid pts each row** to perhaps 27 (46 is the default) for your first experiment as this process can take a while to complete, as it produces a lot of faces. Once you understand how this works, you can increase these numbers to what you want, and you can be patient while it works.
3. Now select the option to **Create an STL file from the plot** from the **File** menu. The STL File Creation dialog will appear.
4. Because you have a contour plot visible, an additional section will be visible in this dialog labeled **Contour Plot Options**. We're going to use the option **Wrap each contour with a tube**. You can decide how large you want the contour tubes (in units from the grid) and how many facets should be used to create the "circular" tubes.
5. CalcPlot3D knows not to add thickness to contour tubes, but, I tend to choose to create the STL file for the surface in a separate step anyway, so I would typically choose **No**.
6. Click the **View Solid** button. A dialog will appear to ask if you want to **View Contours Only** or **View Surface & Contours**. Note that it can take a while to render them together, but you'll want to try that with at least a few contours to see how it will look, and whether the contours will be visible on the top and bottom of the 3D-printed surface or not. But when I'm ready to create the final STL file for the contours only, I usually just view the contours by themselves.
7. A waiting cursor should show you the app is working on your request for a while. For this method, you may sometimes get a message from the browser asking if you want to wait for it to continue working.
8. Once they do render, inspect the contours by themselves to be sure there are no gaps. If you see one or more gaps, try to adjust the number of grid points used in creating the contour plot that we talked about in step 2 above and then click **View Solid** again.
9. If they look good, click the **Create File** button and the STL file for the contours will be saved!
10. Being sure not to change the current settings for the surface plot, clear the contour plot by clicking the contour plot button again. Then use the steps described on pages 1-2 to create the STL file for the corresponding surface (with no contour plot on it).

> **Contour Plot Options:**
> ○ z-Layers (variable thickness for bands)
> ● Wrap each contour with a tube
> ○ Bands of equal width on the surface
> ___
> Radius of tube around contours:
> `0.05`
>
> # of steps in "circles" forming tubes: `8`

NOTE: If you want the contour tubes to extend only on top of the surface, select the **Opposite Normals** option to create thickness on the surface.

If you want the contour tubes to extend out from both top and bottom of the surface, select the **Half-and-half** option. You will also need to consider the relative thickness of the surface and the radius of the tubes around the contours. Note that if the contour tubes are too small, they may not extend out from the surface at all.
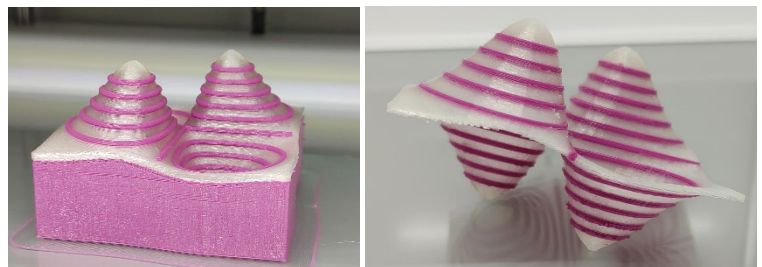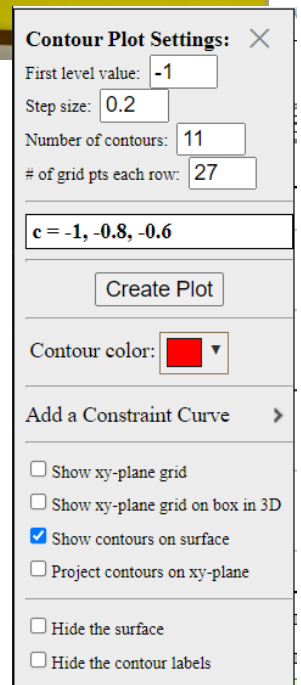

Left: Contours on top of surface
Right: Contours half-and-half in surface

11. See the section at the bottom of this page for instructions on how to recombine the contours with the surface (but with different colors) in Cura (a slicing application needed to 3D print surfaces).

Method 3: 3D printing the contours flat on the build plate while 3D printing the corresponding surface separately. See the image below. These contours were 3D-printed in flexible TPU plastic, while the surface was 3D-printed in PLA.
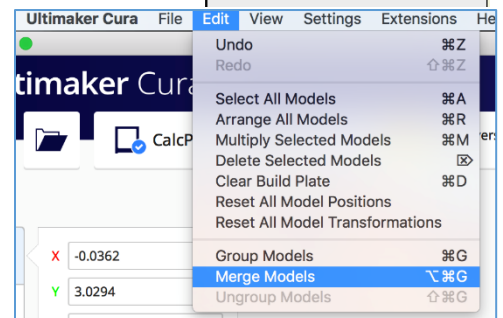
1. Create the STL file for the surface as described on pages 1-2 above.
2. Without changing the other surface plot settings, create the contour plot you want to 3D print and click on the 3D plot to make it display the contours on the surface in 3D.
3. Now we need to return to the contour plot dialog to make some changes.
4. **Deselect** the option to **Show contours on surface** and **select** the option **Project contours on xy-plane**.
5. Click on the **Create Plot** button and click on the plot to make it display in 3D. You should see the contours now only displayed in the *xy*-plane, along with the surface.
6. Now select the option to **Create an STL file from the plot** from the **File** menu. I recommend making the contours thinner than we did for Method 1, especially if you are printing in TPU or NinjaFlex filament.
7. Complete steps 4 – 10 from Method 1 to create the STL file for the contours. These should now 3D print on the base plate without support in 20 minutes to an hour or so, depending on the radius you used and the speed of your 3D printer.

Contour Plot Settings: ✕

First level value: -1
Step size: 0.2
Number of contours: 11
# of grid pts each row: 27

c = -1, -0.8, -0.6

[Create Plot]

Contour color: [red] ▼

Add a Constraint Curve ❯

☐ Show xy-plane grid
☐ Show xy-plane grid on box in 3D
☑ Show contours on surface
☐ Project contours on xy-plane

☐ Hide the surface
☐ Hide the contour labels

Combining Contours with the Corresponding Surface in Cura
If you wish to use Methods 1 or 2 above, you will need to combine the two separate STL files into a single surface using a slicing application like Cura.
1. To do this in Cura, open a new project and load the two STL files representing the surface and its contours into Cura.
2. Select each model and choose which extruder to use, selecting a different one for each.
3. Then select both models and select **Merge Models** from the **Edit** menu. They should now appear merged and with the contours at the right heights, but with different colors.
4. You will now need to figure out how to set up appropriate support (using one of the two extruder colors) and 3D print!

| Ultimaker Cura | File | Edit | View | Settings | Extensions | He |

Undo ⌘Z
Redo ⇧⌘Z

Select All Models ⌘A
Arrange All Models ⌘R
Multiply Selected Models ⌘M
Delete Selected Models ⌫
Clear Build Plate ⌘D
Reset All Model Positions
Reset All Model Transformations

Group Models ⌘G
Merge Models ⌥⌘G
Ungroup Models ⇧⌘G

X -0.0362
Y 3.0294

3D-Printing Constraint Curves
You can use a similar process to Method 2 above to create a surface with a constraint curve projected onto the surface as a tubular curve embedded in the surface. You could also try printing out the constraint curve separately and laying it on the surface as we described doing in Method 3 above. This time the curve would likely not be able to be printed flat on the build plate, but would need significant support.

To create the STL file for a constraint curve take the following steps:
1. Click the button to create a contour plot for the function you are interested in printing.
2. Select **Add a Constraint Curve**, and select a method to generate the constraint curve in the *xy*-plane.

   You can choose between a function $y = f(x)$, a circle centered at the origin, an ellipse centered at the origin and a parametric curve. Once you have selected a curve, go back to the Add a Constraint Curve panel. The type of constraint curve you selected should be highlighted in blue.

   Note that what will be generated is the corresponding curve on the surface with (x, y) inputs on the constraint curve you've specified.
3. Now create the contour plot. It does not matter what contours you include, because we are not going to generate an STL for them this time. Note that you should see the constraint curve on the contour plot in blue. Click on the plot to make it display in 3D.
4. Now that you can see the blue constraint curve on the surface, select the option to **Create an STL file from the plot** from the **File** menu.
5. This time, since it sees you have a constraint curve on the plot there is a new option on this dialog. By default the contours are deselected and the constraint checkbox is selected. This will just generate the STL file for the constraint by itself. If you'd like to create an STL file that combines the contours and the constraint you can select both options here.
6. Click the **View Solid** button. A dialog will appear to ask if you want to **View Contours Only** or **View Surface & Contours**. Click the **View Contours Only** button, as it will give us whatever we selected above in step 5, including the constraint.
7. If it looks satisfactory, click the **Create File** button to save the STL file. You can either print it by itself to set on top of the 3D-printed surface or you can combine it with the surface and print them together like we did in Methods 1 and 2 for the contours above.

Creating Corner Posts for Surfaces: You can use Regions to create small corner posts to help surfaces lie flat. The hardest part of this process is determining the lowest point on your surface.

1. Select Region from the Add to graph menu.
2. Find the lowest point on the surface and use this value as **Bottom** $f(x,y)$.
3. Use the Function as Top $f(x,y)$.
4. Set the x and y-bounds to mark off small squares.
5. Deselect surface so only posts are visible and save these to an STL file.
6. Save surface to an STL file.
7. Import both STL files into Cura, select them both, **Merge Models** to align and then **Ungroup Models** to drop posts to the build plate.
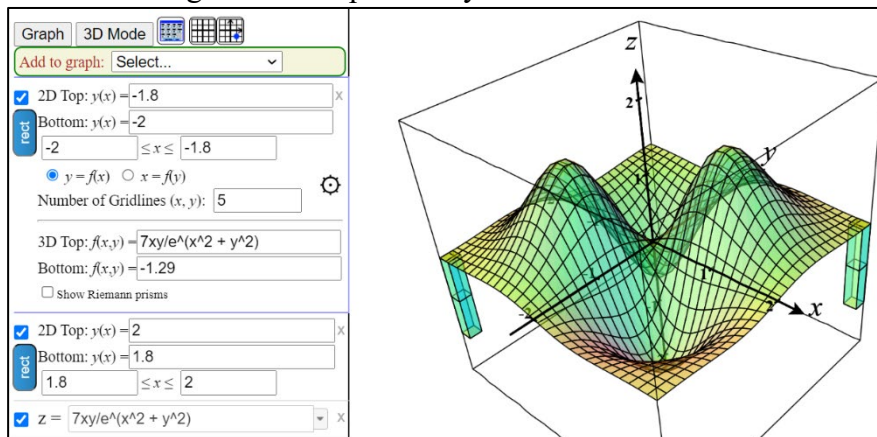
3D-Printing Solids of Revolution (Shells and Washers)

It is also possible to use CalcPlot3D to generate STL files to 3D print solids of revolution or the collection of shells or washers that approximate them.

To 3D print a solid of revolution (not the shells or washers):

1. Use the **X** in the upper-right corner of the function object element to delete the default surface in CalcPlot3D.
2. Use the **Add to graph** dropdown menu to select the **Surface of Revolution** object.
3. To display it, either select the display checkbox on the object definition element or press enter in one of the function input boxes.
4. Adjust the region and axis of revolution to the example you would like to 3D print.
5. Rotate the region to the position you prefer for the 3D print.
6. Now select the option to **Create an STL file from the plot** from the **File** menu.
7. A well-defined solid of revolution should be a closed region, so you should be able to select **No** for creating surface thickness.
8. As nothing should change when you render the solid of revolution in the plot, there is not much benefit from clicking the View Solid button this time. Go ahead and click the **Create File** button to save the STL file.
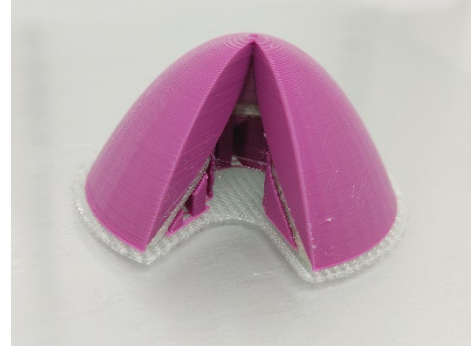
3D-Printing Washers or Shells

CalcPlot3D can also facilitate 3D printing a collection of washers or shells representing a solid of revolution. However there are some special things we need to consider as we create the STL file used to 3D print these objects. First, we need to consider whether we want the washers/shells to come apart (like a puzzle) once we 3D print them, or if we want them to 3D print as a single demonstration piece. We can accomplish either goal by adjusting the gap between the shells/washers and by choosing whether to create separate STL files for each shell/washer so they can be 3D printed separately.

Note that we also need to choose whether we wish to include the region rotated about the axis of revolution in the 3D print. If we do include it, the shells/washers will be connected making a single inseparable model, even if we include a large enough gap that they would normally come apart.

1. With your Solid of Revolution from above, select the checkbox labeled, **Show *n* washers/shells**. Depending on whether your independent variable in your bounding functions is the same as the variable in your axis of revolution equation, you will see a series of washers or shells representing the solid.
2. Adjust the number of washers/shells using either the corresponding input box or the slider.
3. If you wish the shells/washers to be able to come apart after you print them (or if you wish to 3D print the region in a different color), you'll want to select the checkbox labeled **Hide region in *xy*-plane**.

4. Revolve the washers/shells with the long slider above that we used earlier to revolve the region. Decide on whether you wish to 3D print complete washers/shells or a partially revolved model.
5. Now select the option to **Create an STL file from the plot** from the **File** menu.
6. Notice that a new section is visible titled, **Region Settings**. Here we can adjust the size of the gap we'd like to have between the washers/shells in the 3D print. I've found that a gap of about 0.65 mm works well for separating concentric shells. But note that we need to consider how large the gaps will be in Cura (or another slicer) once we possibly resize the solid there. Currently the gap is adjusted only for the scale factor shown at the bottom of the dialog (25.4 mm per inch, where inches are the assumed scale on the axes in the plot).

   For washers 3D printed on top of each other, you will likely need to use a very small gap, but note that a gap of 0 mm will likely not show any clear indication of the separate washers (or shells), particularly if you 3D print a partially revolved model. For this reason, I recommend using a gap of 0.1 or 0.2 mm to show the separate shells/washers more clearly in this case. But you will need to experiment with your 3D printer to see what size gaps produce the results that you prefer.
7. If you wish to 3D print the shells or washers separately, select the checkbox labeled **Save washers/shells in separate files**. When you click the Create File button, a series of files will be saved, one for each of the shells/washers and one for these shells/washers together. This option makes it much easier to 3D print shells that fit more tightly together while not being stuck together. It also allows us to 3D print separate washers that can be stacked to show the washer method!

Another issue that you will need to consider, if the shells or washers do separate, is how to display them in their appropriate positions in the object once they are 3D printed.

Shells that naturally extend down to the *xy*-plane (and build plate) are particularly easy to 3D print and display, as gravity helps them to stay in the correct positions.

But it's more of a challenge to display shells (or washers) for solids whose lower surface is curved.

A feature is being considered that may allow you to create a small ledge around a lower shell at the correct height to support an adjacent higher shell. Settings to select/adjust this feature will appear under the checkbox described in step 7 above.

This feature is envisioned to work for shells, but it may not help with washers. We are working to develop an approach to facilitate this, but we'd welcome any suggestions you may have!