

3D printing Workshop Breakdown

Opening Lecture/Remarks (20 - 30 Minutes)

-Introduction to 3D modeling software

- Overview of what 3D modeling software is
- Introduction to 123D Design

Introduction to 123D Design (40 minutes - 1 hour)

-Basic 3D modeling

- A handout will be provide that walks the participants through some of features/tools of 123D Design
- At the end of this section of the workshop the participants will have created 3-4 “base” objects

Sandbox/ Introduction to 123D Design (1 hour)

-Making your own 3D object

- Using the “base” models created earlier in the workshop participants will create their own models
- Participants can create their own model if they already have something in mind.

3D printing time / More Sandbox (40 minutes – 2 hour)

-Making your own 3D object

- Participants not ready to print will be encouraged to continue working on their object
- If they don’t finish their object in time they will be provided an email address they can send it to once they are done.

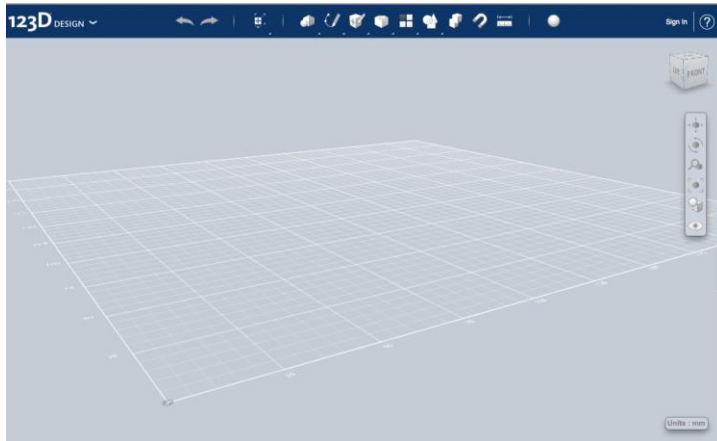
-Printing using the Makerbot

- Using the Makerbot desktop software participants will set-up and start a 3D print.
- Participants who do not finish their own object will be provided a dummy file so they can go through the process.

1. Introduction

The layout of 123D Design is comprised of three major parts: The workspace, the toolbar, and the viewing options.

1.1 The Workspace



The workspace is where sketches and objects can be created and manipulated. It is represented by a grid, which will scale in size in relation to your project.

1.2 The Toolbar



The toolbar is at the top of the window and contains various features/tools that will allow you to create 3D objects. This session you will be introduced to several, but not all, of these features.

1.3 Viewing Options



The viewing options can be found on the right hand side of the window. These will allow you to move the camera and change the property of the object in order to make modeling easier.

TIP: Right clicking while over the workspace will allow you to “orbit” objects

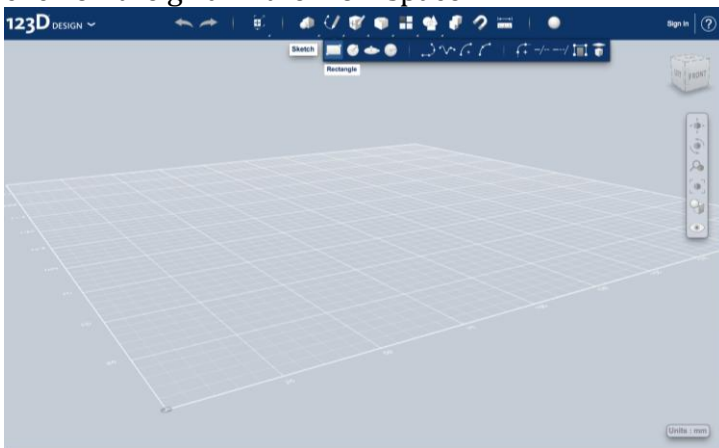
2. Object One – DICE

The first object we will be creating is a dice. The goal of this section is to introduce you to several features of the program (such as the extrude, cut, and fillet) as well as challenging you to think three dimensionally.

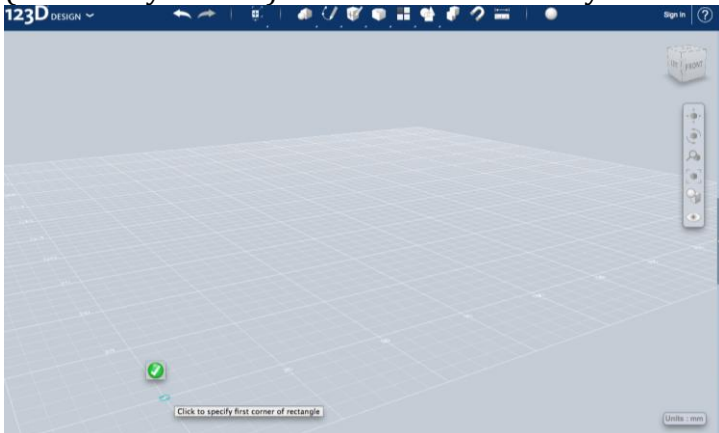
2.1 Creating a Base Solid

The first thing we need to do is to create a base solid for our dice.

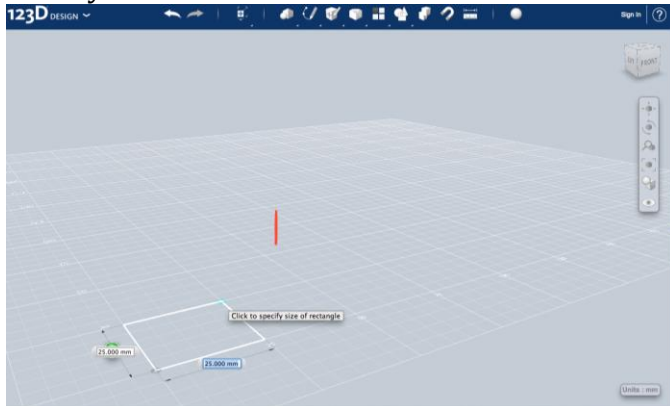
Select the **SKETCH** option in the toolbar and click on the **RECTANGLE** option. Then click on the grid in the workspace.



This will cause the grid to expand. To begin sketching click on the **ZERO POINT** (marked by a circle) which will now act as your base point for this object.

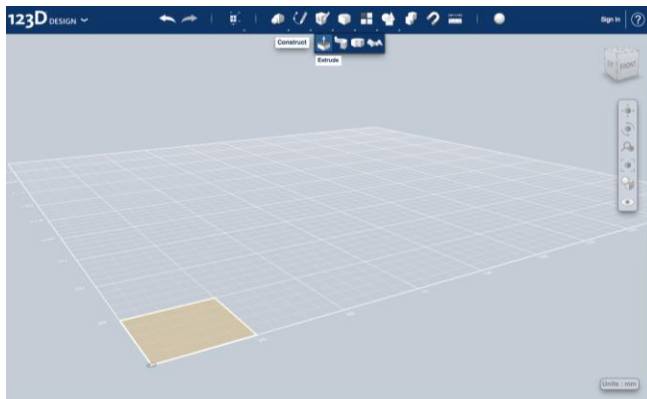


Move your mouse out a bit and the outlines of a rectangle will be project.

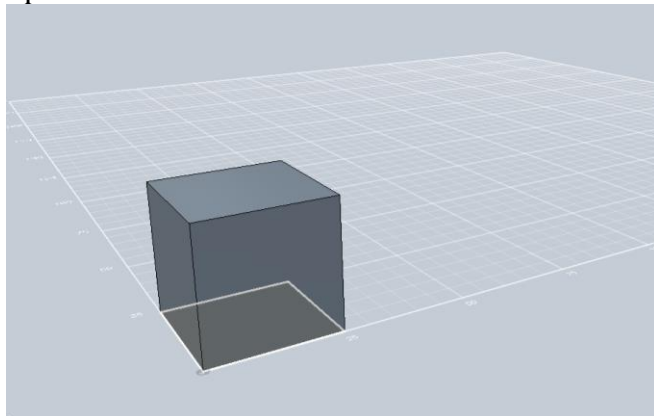


Enter a value of 25mm for both the length and width (using tab to move between the value boxes) and press enter. Your newly created square will change colours indicating you have **CLOSED** the sketch.

Next, go back to the toolbar and select the **CONSTRUCT** option and click on the **EXTRUDE** tool.



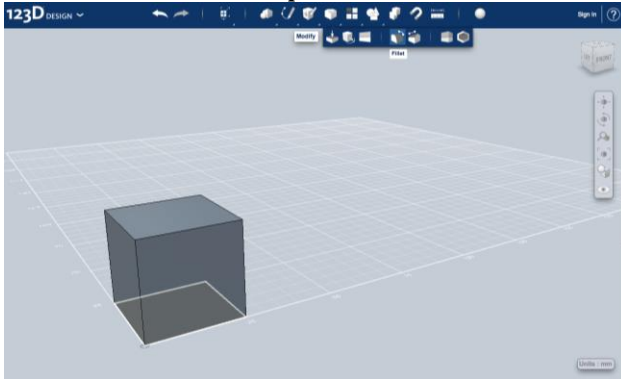
Click on your square, input a value of 25mm, and then press enter. You should end up with cube that looks like this:



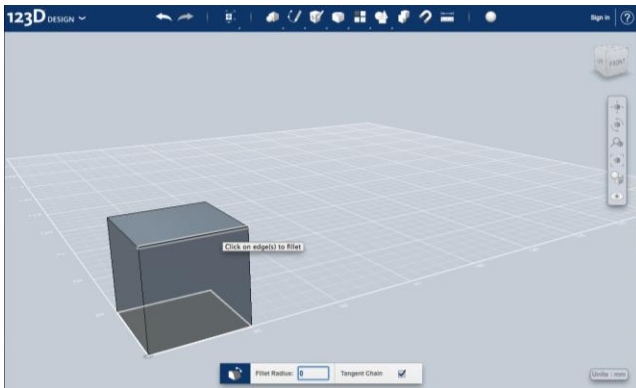
2.2 Modifying the Base Solid

With our base solid now in place we can start to make it more “dice” like in appearance.

Select the **MODIFY** option and click on the **FILLET** tool



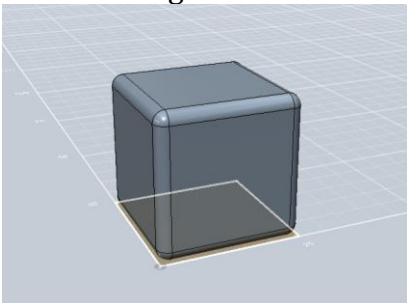
Begin to select the **EDGES** of the square. They will change in colour once they have been selected.



Once all the edges have been selected input a value of 2mm for the fillet radius and press enter.

TIP: The edges can be fillet individually if you wish.

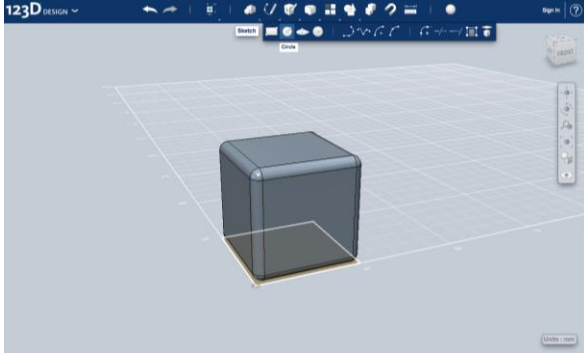
The resulting cube should look like this:



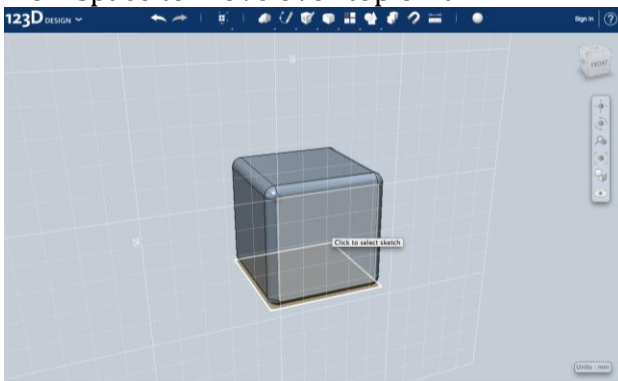
2.3 Making Holes

The final step in making the dice is to assign values to each face.

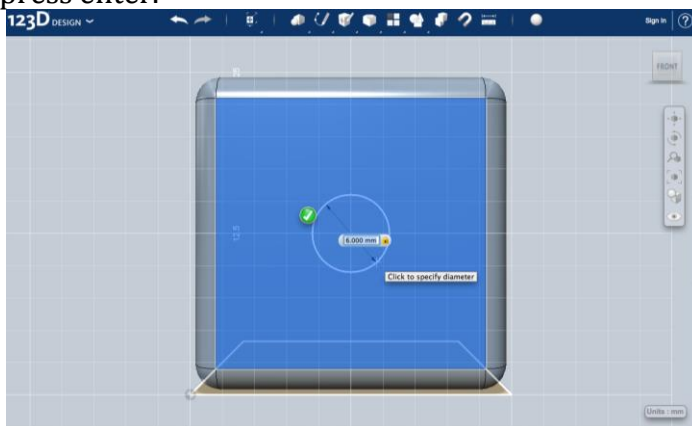
Selected the **SKETCH** option in the toolbar and click on the **CIRCLE TOOL**



To begin your sketch, click one of the faces the dice. This will cause the grid in the workspace to move over top of it.

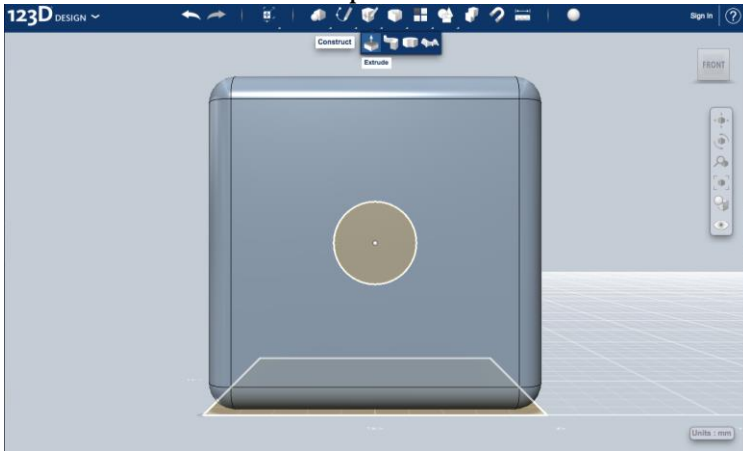


Move your mouse to the center of the face and click. Move your mouse slightly and the outline of a circle will be project. Input a value of 6mm for the diameter and press enter.

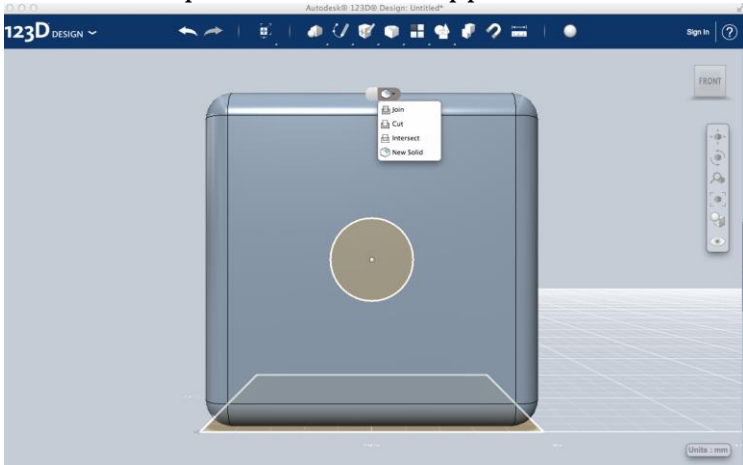


TIP: Rotating the camera so it is directly pointed at the face will make this process easier.

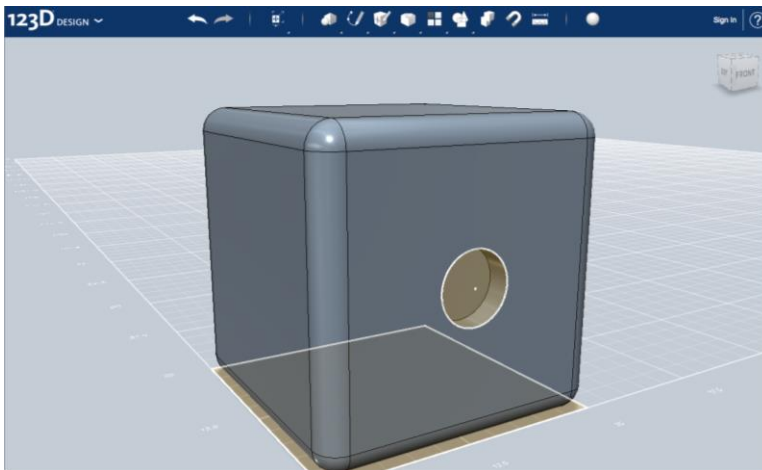
Select the **CONSTRUCT** option from the toolbar and click on the **EXTRUDE** tool.



From the dropdown menu that appears select the **SUBTRACT** option.

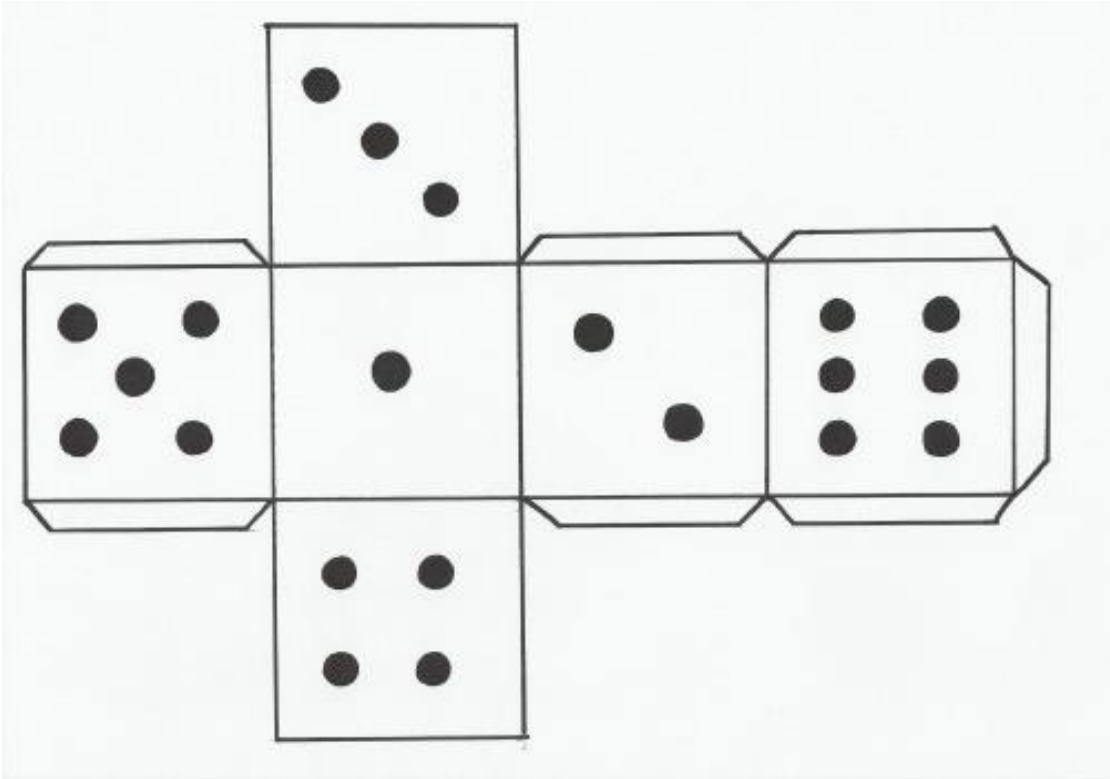


Then select the circle you just sketched, input a value of -3mm, and press enter. This will result in a cube that looks like this:

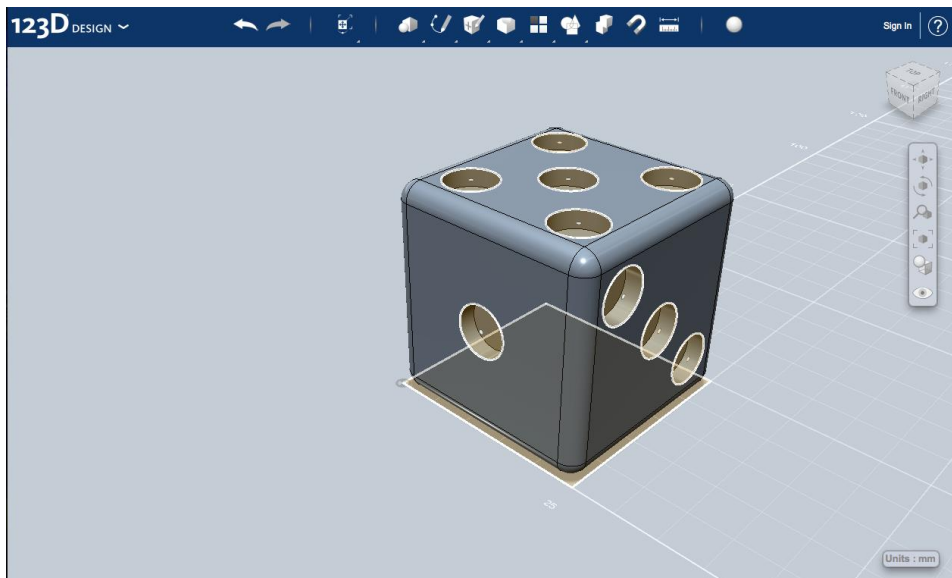


2.4 Finishing Up

Using the below template assign the proper values to each of the dice's face.



The end result should look similar to this:



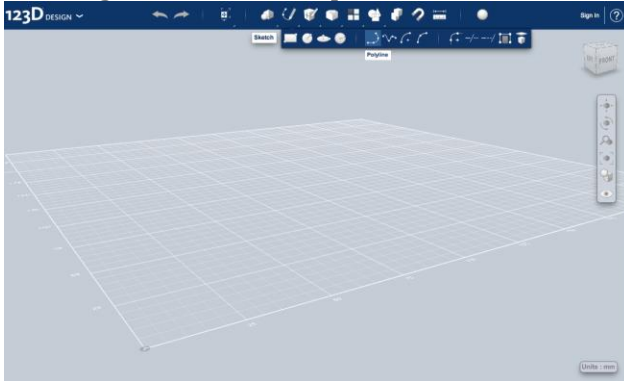
TIP: When subtracting multiple circles can be selected

3. Object Two – Cup

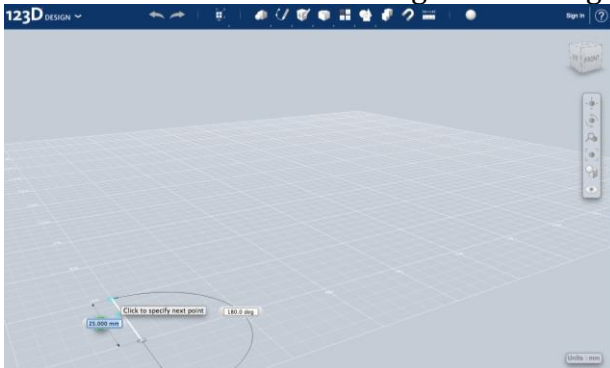
In this section of the tutorial line construction, line fillet, and revolving will be taught

3.1 Creating a Base Sketch

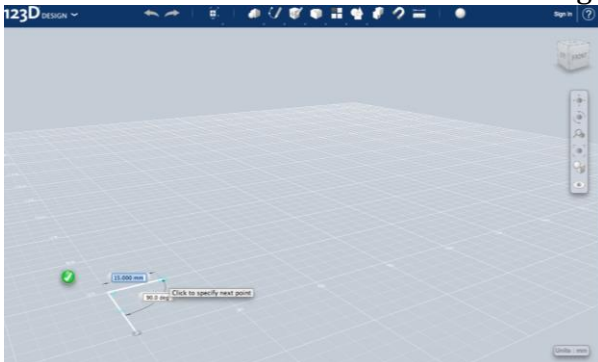
Select the **SKETCH** option in the toolbar and click on the **POLYLINE** tool. Then click on the grid in the workspace.



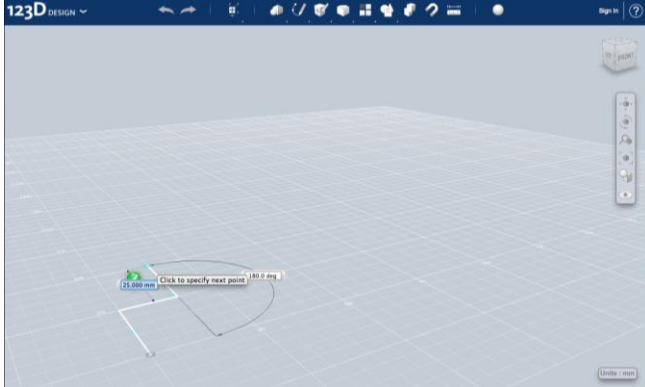
Click on the **ZERO POINT** to create a starting point, input a value of 25mm for the distance of the line with an angle of 180 degree, and then press enter



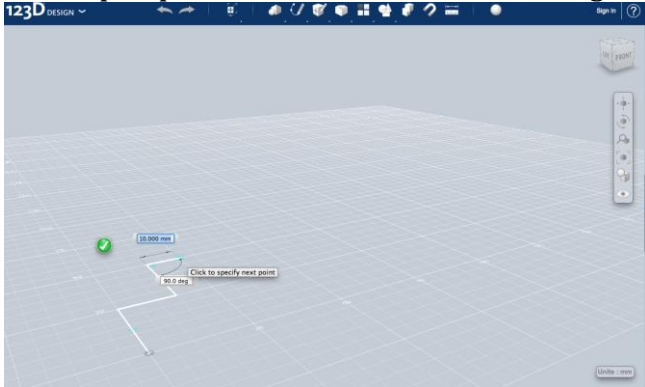
Re-select the **POLYLINE** tool and click on the end of your newly created line. Input a value of 15mm for the distance with an angle of 90 degrees and then press enter.



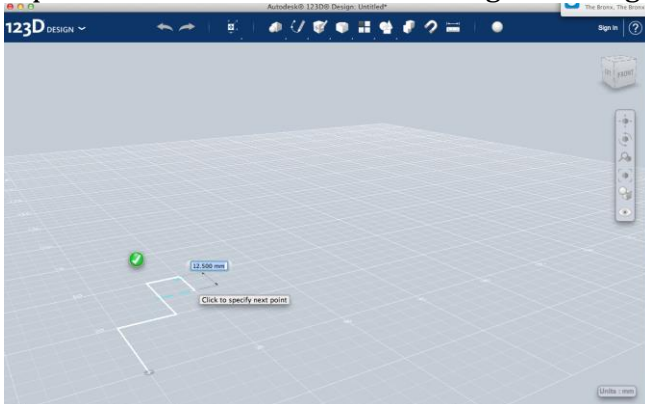
Re-select the **POLYLINE** tool again as click on the end of the line you created in the previous step. Input a value of 25mm with an angle of 180 degrees and then press enter.



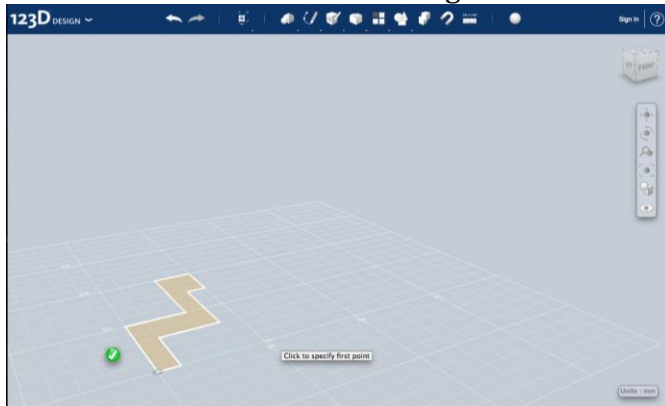
Once again select the **POLYLINE** tool and click the end of the line you created in the last step. Input a value of 10mm with an angle of 90 degrees and then press enter.



Select the **POLYLINE** tool again and click the end of the previously created line. Input a value of 12.5mm with an angle of 0 degrees and then press enter.

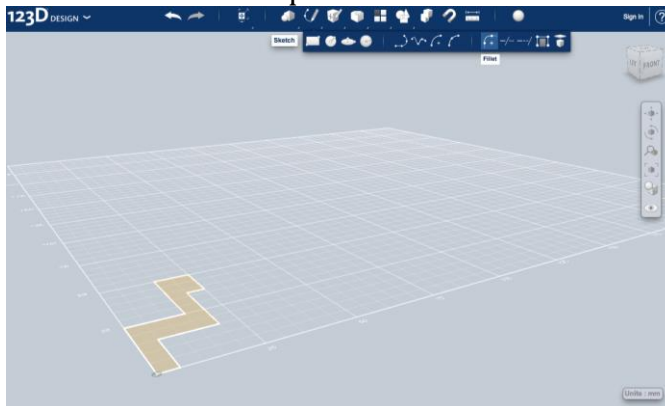


Using the **POLYLINE** tool sketch out enough lines to enclose the base sketch. The end result should look something like this:

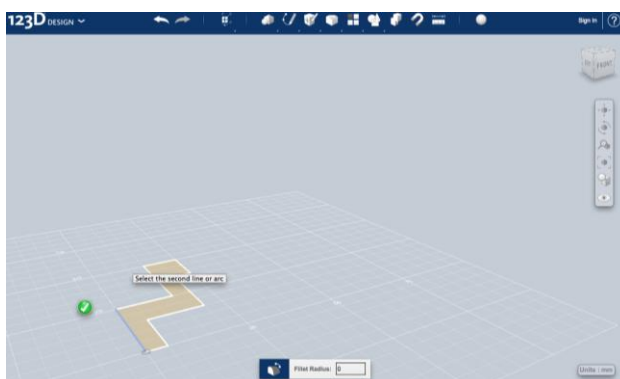


3.2 Modifying the Base Sketch

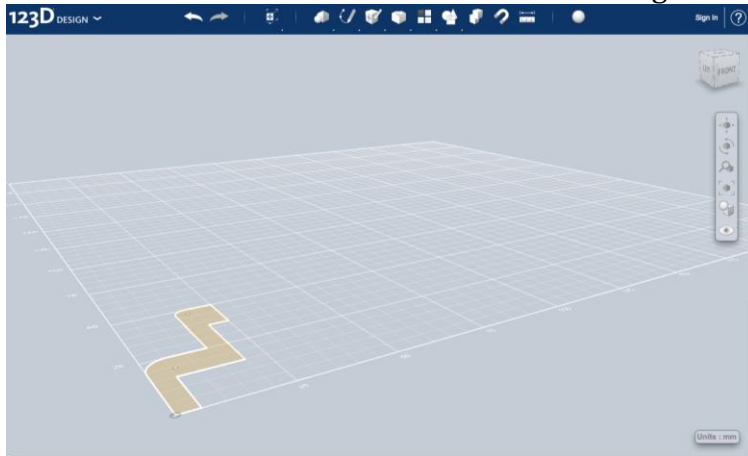
Select the **SKETCH** option in the toolbar and click on the **FILLET** tool.



Click out the outer edges and input a value of 2mm for both. (A red outline will be projected to give you a preview of the fillet)

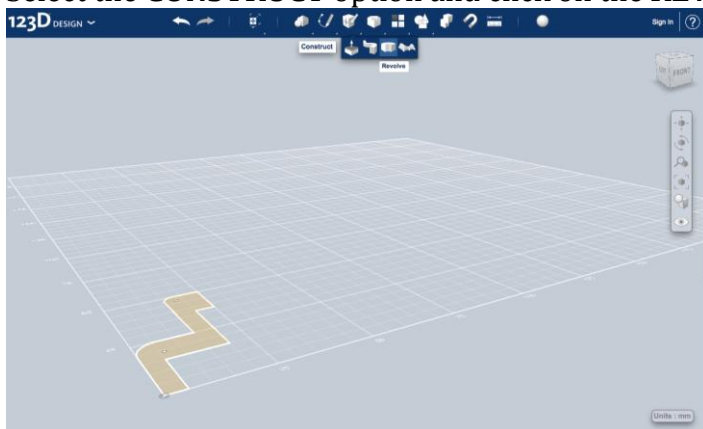


When finished the sketch should look something similar to this:

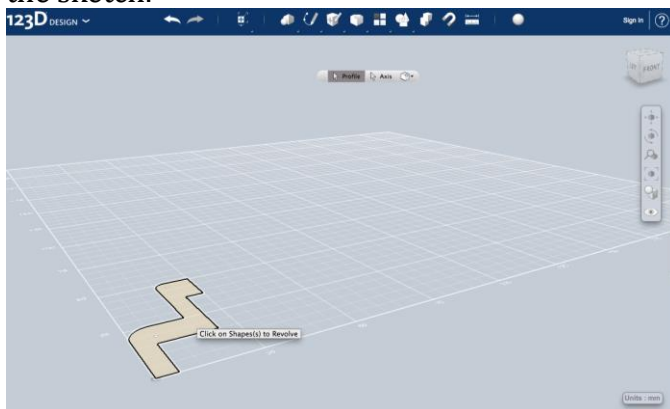


3.3 Revolving

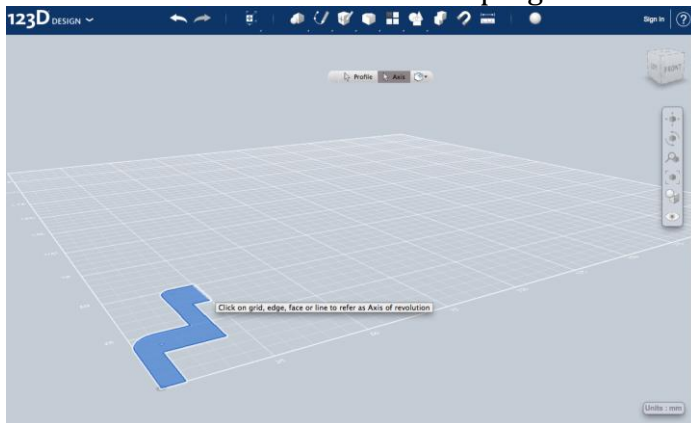
Select the **CONSTRUCT** option and click on the **REVOLVE** tool



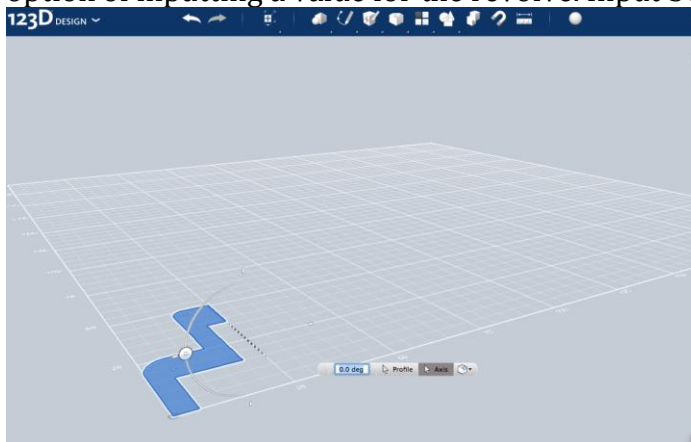
For a revolve you will need to select both a profile and an axis. For the profile select the sketch.



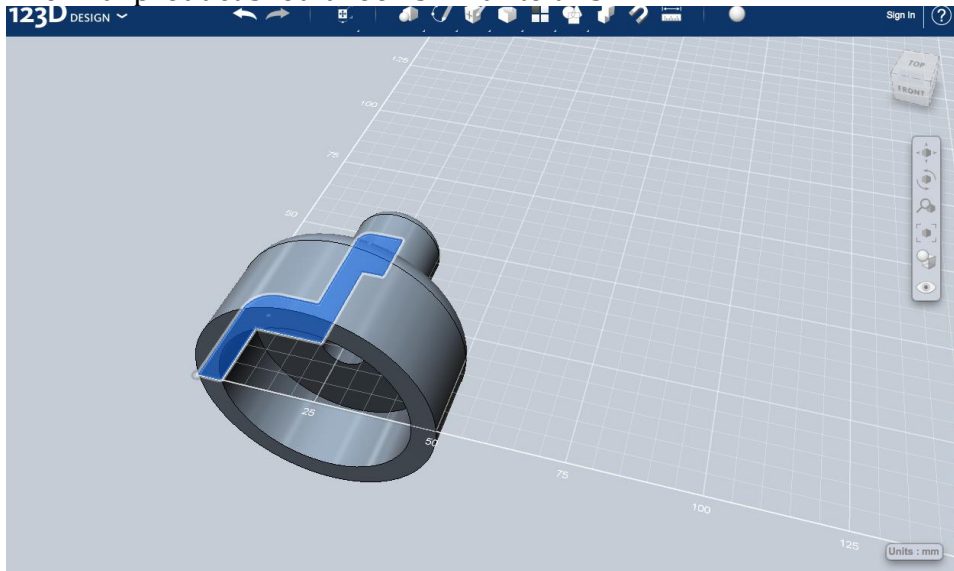
For the axis select the line in the top right corner



Once both the profile and axis have been properly selected, you will be given the option of inputting a value for the revolve. Input 360 degrees and press enter.



The final product should look similar to this:



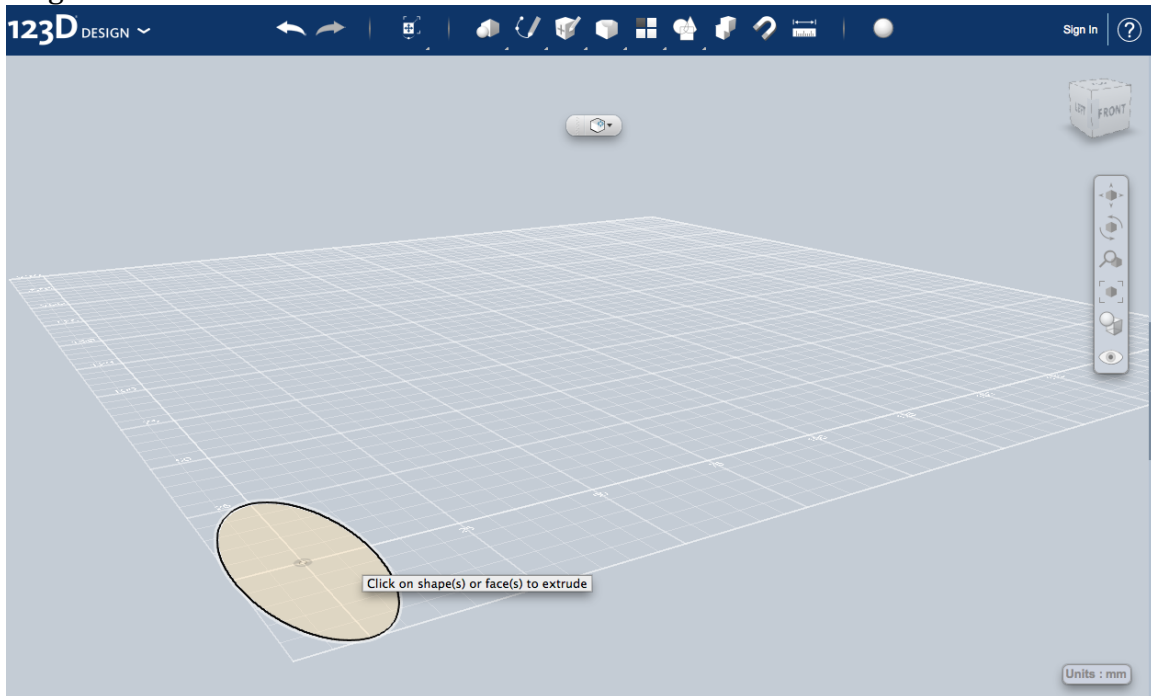
4. Object Three – Dish

In this section of the tutorial the push and shell function will be taught

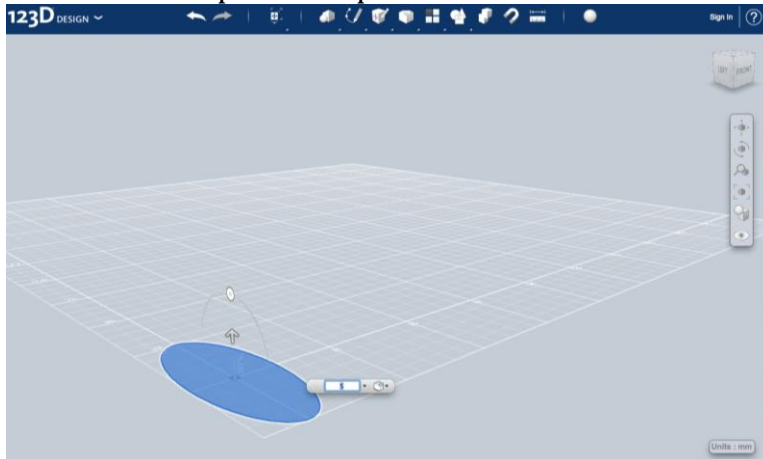
4.1 Creating a Base Sketch

Select the **SKETCH** option in the toolbar and click on the **ELLIPSE** option. Then click on the grid in the workspace.

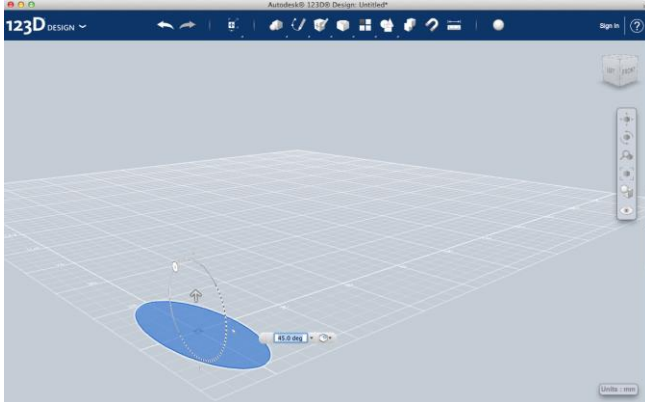
Using the **Zero Point** as your starting point create an input a width of 10 mm and a length of 25 mm



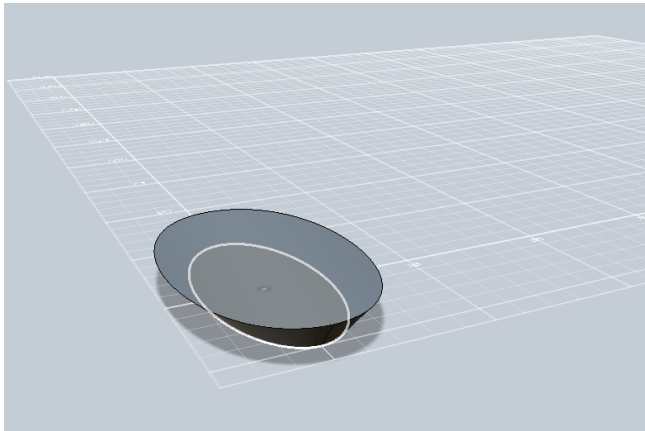
Select the **CONSTRUCT** option from the toolbar and click on the **EXTRUDE** tool. Click on the Ellipse and input a value of 10mm. **DO NOT PRESS ENTER YET**



Above the preview of the extrude there is a dial that can be moved. This is the push function. Begin to move the wheel and then input a value of 45 degrees and press enter.

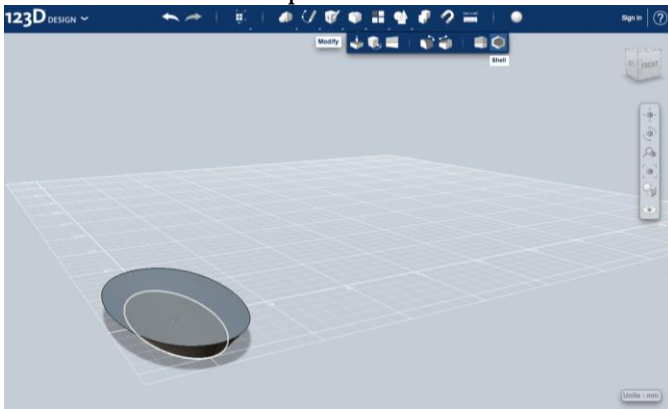


The end result should look like this:

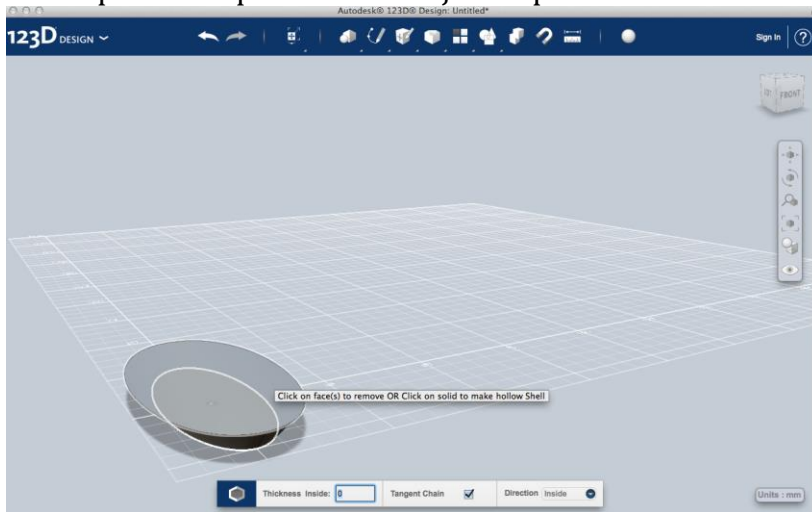


4.2 Shelling

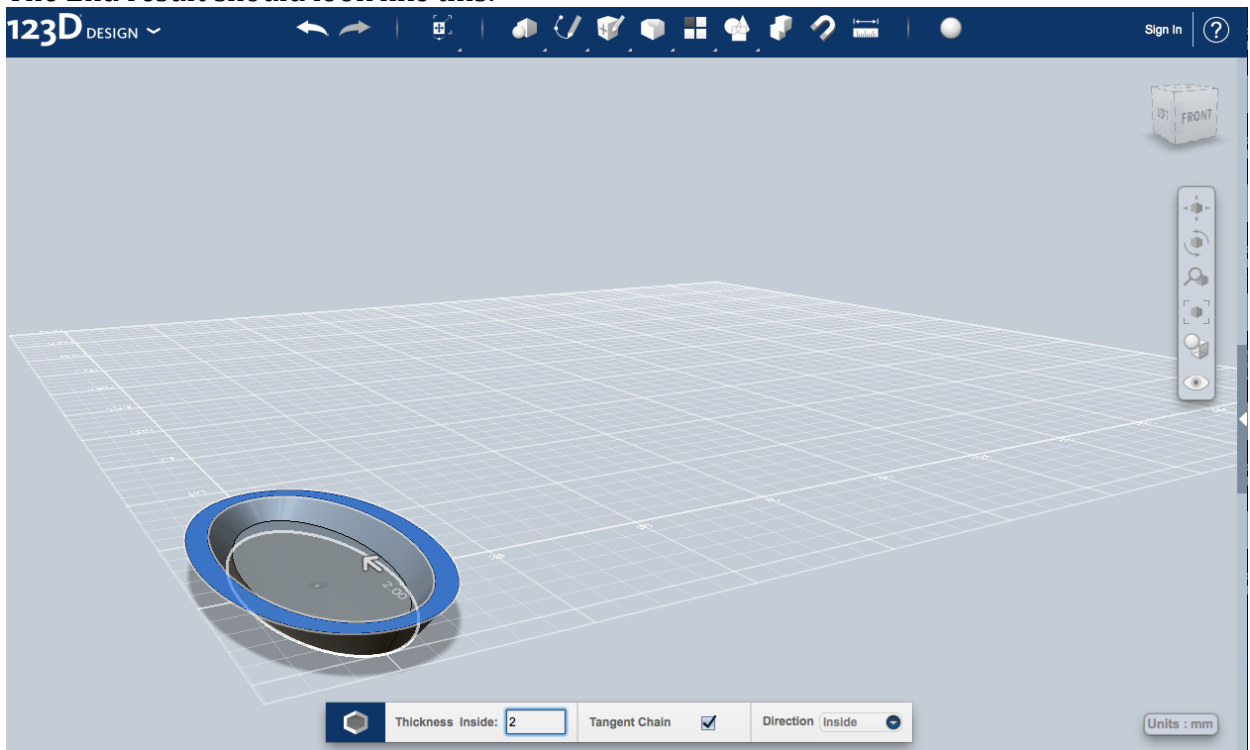
Select the **MODIFY** option in the toolbar and click on the **SHELL** tool.



Click upon the top face of the object. Input a value of 2mm and press enter.

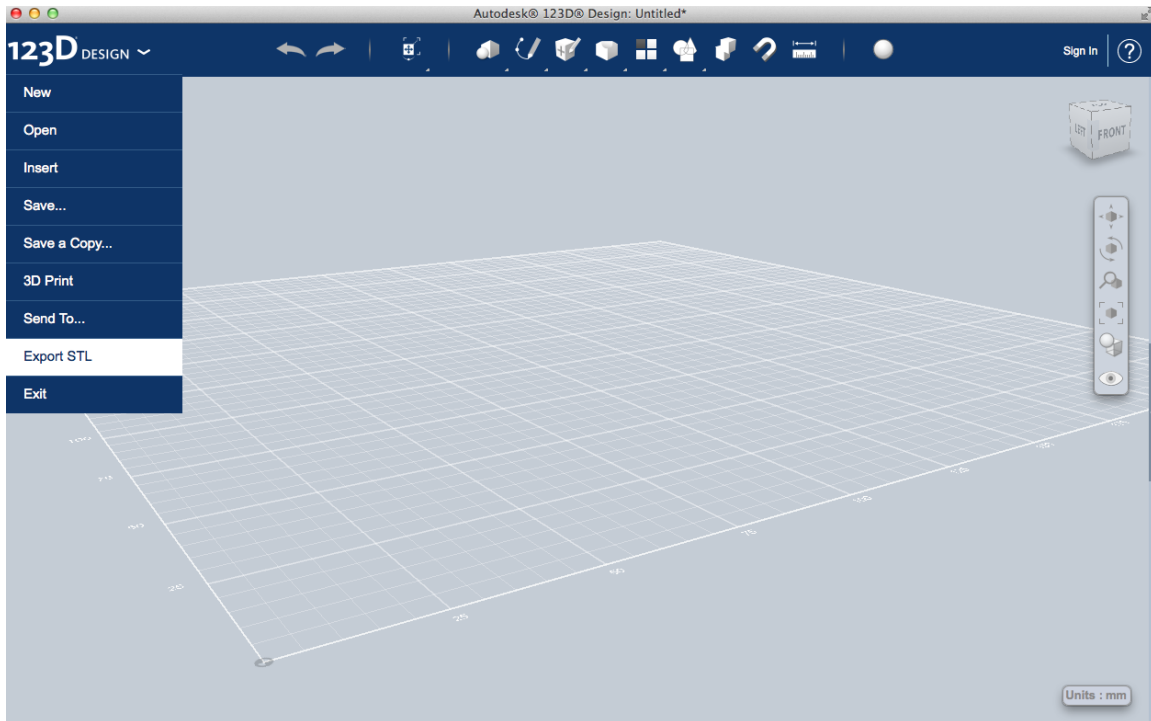


The End result should look like this:



5. Finish your 3D Model

Once you've finished your 3D model it needs to be exported as an STL if you want to 3D print it. To do this select the 123D Design drop down in the toolbar and click on export STL option.



Save your file and you're ready to 3D print!

For the rest of the workshop you are free to design whatever you'd like. If you cannot finish the model in our allotted time you have until Tomorrow at 9AM to email the file to me at daniel.southwick@mail.utoronto.ca