Starting to design with FreeCAD

Intended Audience

It's important to clarify that this document is not for the seasoned, or even intermediate user of FreeCAD who already has personal preferences for workbenches and design techniques. The intent is to provide new users guidance through what initially appears as a dizzying maze of workbenches. Along the way some general design techniques, as well as, FreeCAD specific techniques may also so be discussed.

Introduction

Once a new user gets past installing FreeCAD and exploring the FreeCAD interface as described in the FreeCAD wiki [Getting Started](), the list of work benches can seem daunting. Even seasoned CAD users can be a bit confused as to where to start. It doesn't take long to find out there are several ways to create a blob. And given that, for example, there is padding in the Part Design workbench and extruding in the Part workbench, which do I use for the design I want to work on. Not to mention there are sweeps, lofts, and primitives in both Part and Part Design. Then there are master sketches, unattached sketches used in Part workbench and attached sketches in Part Design.

There are plenty of tutorials and videos on Youtube that tell how do things with the various tools. But, that accomplishes the *"how"* of using the tool, but the *"why"* is rarer to find. Of course, there are also situations where the *"why"* is simply answered *"because that is the only way to accomplish that particular shape"*.

So where do you start to design a given idea? This question is what this document intends to discuss.

Some History

The Draft workbench is designed to draw on 2D planes. It can do some 3D work, but, it isn't the most user friendly way to do 3D. Still there are things where it really is the only choice. For example, drawing 3D lines that represent paths in space. It also has some lmited features for doing drafting like dimensioning of shapes. There are several operations available only on the Draft workbench, for example creating shapestrings (text string outlines) that can be used to extrude or cut out strings in solids.

Note: there are special functions in Draft workbench for specific use cases, for example processing profiles imported from a DXF file.

Part workbench came before Part Design and, as such, has a lot of features. What actually gets created in Part and Part Design have subtle differences that, in many cases, are more important to the

developers that to the end user. But, there are some things the end user needs to have a general understanding of to make life easier. We'll have a look at this later.

NOOB Guidelines in Brief

- Install the latest FreeCAD versions
  - On most platforms this is 0.18
  - The daily build version of 0.19 is very stable and used by many
  - Use the appimage version if necessary
- Skip tutorials about Draft workbench, at least in the beginning.
  - There are long time FreeCAD users who have never or seldom used it.
- Start learning Part Design workbench
  - Try to find tutorials and videos that use recent versions of FreeCAD
- Start by learning how to model basic parts.
  - Don't start modeling airplane hulls and screw threads
    - Yes, they can be done. But, they are less than trivial exercises
- Join the forum at freecadweb.org
  - Try to model parts
  - Ask questions
    - Make sure to include your system info as described in the pink bar seen in the forum
    - Make sure to include you FreeCAD file
      - If it's too large, try to simplify it, so it only shows the question/s
      - If there are other files, imported stl's, step files, include those
    - If you have lots of questions:
      - start with no more than 3
      - address more questions in other posts or as the forum thread progresses

New Install Recommendations

There are many ways to customize FreeCAD. By default there are many workbenches. Additional workbenches and macros can be installed to supplement FreeCAD base functionality. For those so motivated, macros or Python scripts can be written or acquired from various sources.

Initially, I'd recommend using the Tools > Customize > Workbenches tab to hide the workbenches that are not necessary for the new user. Reducing to Part Design, Sketcher, TechDraw, and Image initially will provide plenty of things for the new user to learn before adding more as the need arises.

Ground Zero

Designing things is not a simple task. Designing things with a CAD program entails a variety of skills. There are the skills of using the CAD tools provided by the software to develop the desired shaped object. The question of whether to start with a primitive (cylinder, cone, cube, etc.) or begin with an outline of a final shape.

When you have a "thing" you want to model you need to decide on your approach. A part that has a cylindrical cross section could be accomplished with a primitive in either Part or Part Design. A part with a more complex cross section may be better off using the sketch and pad in Part Design.

Note: to add to the confusion, you can, create a sketch with the Sketcher workbench and use it to extrude solids in the Part workbench. More on this later.

For a beginner it is best to stick to one workbench or the other. But, there are some things to watch out for and beware of.

If you create an object of a given shape and dimension in Part workbench and make an object of the same shape and dimensions Part Design workbench, they are the same visually and even would give the same center of gravity, moment of rotation, etc. But, they are different to FreeCAD. This is, and this is skimming a **LOT** details, because from a software point of view they reside in different wrappers. For the most part there are ways to change the wrappers.

Side Note: For example, an extrude object created in Part workbench can be moved into a Part Design wrapper by simply dragging and dropping the extrude object into a Part Design Body.

I think Part Design is the best place to start. That's my opinion.

In general the work flow in Part Design workbench is to draw a shape in a sketch, then pad the drawn shape. (In other packages it is called extrude. In fact in Part workbench it is called extrude as well.)

We'll detail the process more below. But, first let's look at the design thought process involved. Let's assume that we will be modeling a simple box with a counter bored hole:
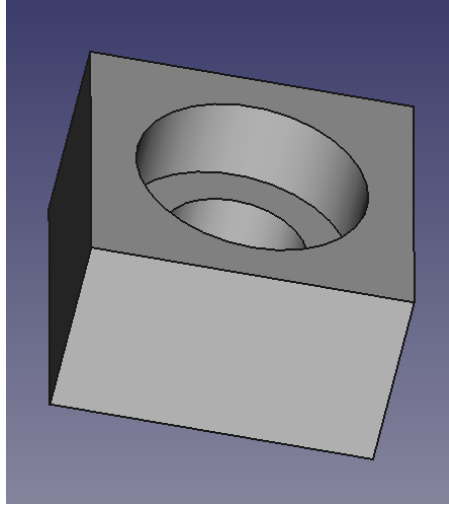
A machinist would be expecting to see something like below to actually machine this part. The image below shows a typical three view projection of the top, front, and right side.

This mechanical drawing contains all the dimensions required.

But how would we model something like this?

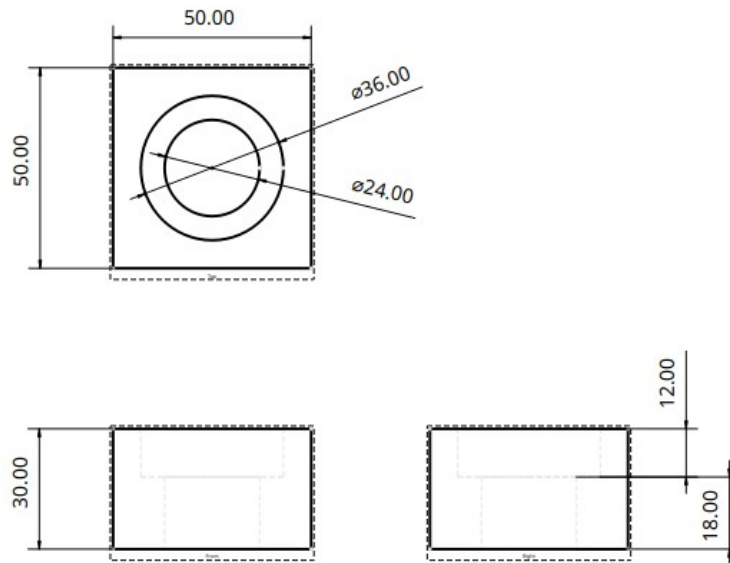As is typical with modeling software there is more than one way. Let's think about the the process.

Figure 1: Mechanical Drawing

There is a lump of material that,, from the top view, is 50 mm X 50 mm. From the side view we can see it is 30 mm high or thick. So, the first thing we need to do is make our lump.

It could be made in a few different ways:

1. Create a sketch of the top view and pad.

2. Create a sketch of the front view and pad.

3. Create a sketch of the right side view and pad.

There may reasons to choose any of the three ways. Since, in this case the counterbored hole is centered in the lump, and is a simple shape, we'll choose #1.

The process to create this will be similar to virtually every model you make.

With FreeCAD open and Part Design workbench selected:

1. Create a new document. (use the icon, or Ctrl-n keyboard shortcut)

2. Create a new Body and make it active. (left click the blue block stair)

   1. Active will be the default state, as indicated by a gray background for the Body in the Model tab of the Combo View. (typically on the left of the FreeCAD window)

3. Create a new sketch. Click the Create new sketch icon.

1. Select the working plane of this sketch.

2. Note: If you are in Part Design workbench and create a new sketch, the Body will be added automatically.

Now let's step through actually creating our lump of material. I will add some notes for things you should do as you proceed through these steps.

With FreeCAD open and Part Design workbench selected:

1. Create a new document. (use the icon [image], or Ctrl-n keyboard shortcut)

   1. Save the document and name it appropriately. How about Exercise1.

   2. My motto: save, save often.

   3. Note: FreeCAD does have periodic backup and can in many cases use the backup to recover in the case of a crash.

   4. Recommendation: select View > Toggle axis cross (or keyboard shortcut A C)

2. Create a new Body and make it active. (left click the blue block stair [image])

   1. Note in the tree that the name of the body, currently *Body* is on a gray background indicating it is the active body.

   2. Right click on *Body* notice there is a choice to Rename, select this (you can also hit F2) and give it an appropriate name. Let's call it *Lump*.

   3. Save the document if you like.

When you have done this take a moment to see what is in the Model tree. The body we just created and renamed *Lump*. Under the body there is a branch called *Origin*. By default it is grayed out. We can explore this a bit by clicking it and pressing the space bar to un-hide it. This an import piece in Part Design. This *Origin* is a sort of framework that anchors the body and subsequently all branches (referred to as features) in space. By default the origin is located at X=0, Y=0, and Z=0. (Note: The object *Origin* can be moved from 0,0,0 it is typically left alone.) Also note, there are six objects under the *Origin* that represent the *x/y/z* axes and the *XY_Plane* (top), *XZ_Plane* (front), and *YZ_Plane* (right) views of our 3D virtual space.

3. Create a new sketch [image] .

   1. Select the working plane of this sketch. For our example, select the XY_Plane since we want to sketch in the top view. Click Ok.

2. The sketcher will now open. For the moment click Close.

3. Right click on the Sketch in the tree (or hit F2) and rename the sketch, TopSketch

4. Save the document.

Take a look at the tree in the Model tab of the Combo View. You have this:
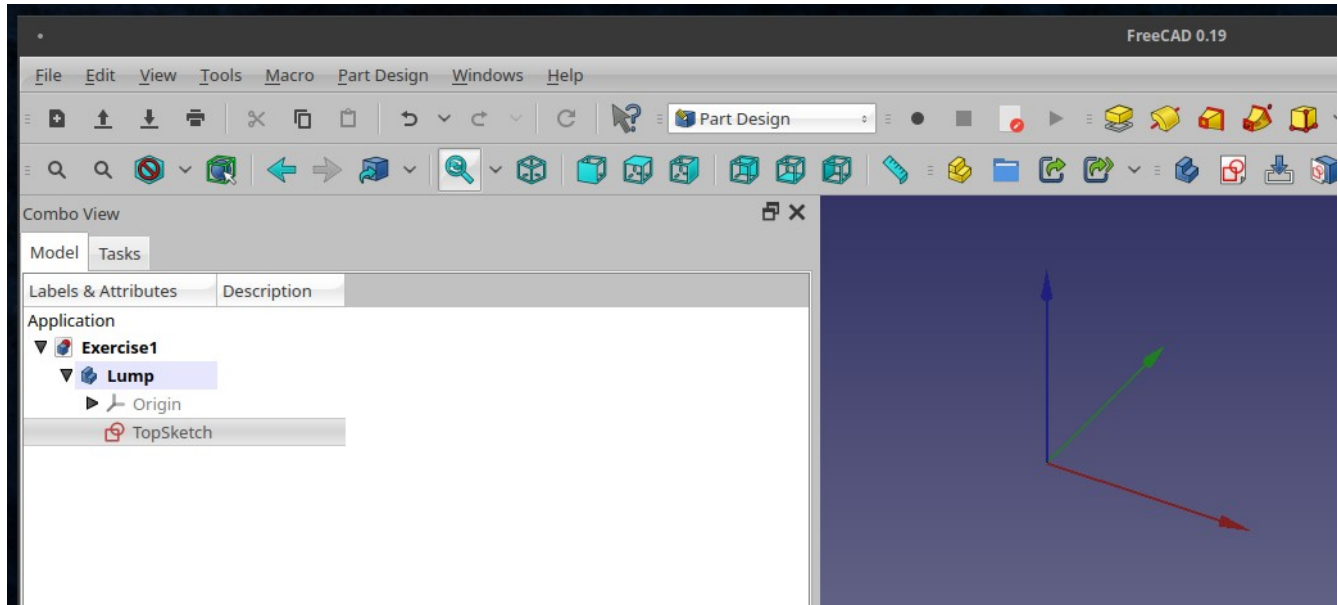
Let's continue.

4. Double click on the sketch to open it.

Note the sketcher is now open and the *Axis cross* is in the center of the grid (assuming the grid is turned on). (If it is not, select the Tasks tab in the Combo view find the Edit Controls and click the Show grid check box.) You can rotate the view to see the axis cross. The grid will resize as the geometry of the sketch changes. (If you rotated the view go back to top view by clicking on TOP in the rotation cube (the rotation cube is in the upper right corner of the 3D view)

(Note: before the next steps, close the sketch and toggle the axis cross off, it'll make it easier for the moment. Then re-open the sketch.)

5. Select the *Create rectangle* tool (or R on the keyboard).

6. In the Edit controls, make sure Auto constraints is unchecked.

7. Click somewhere on the grid to indicate the lower left of our rectangle, then somewhere else indicating the upper right of the rectangle.

8. Move the mouse over the upper horizontal line. When it turns yellow, click to select it.

9. Move the mouse over the left vertical line. When it turns yellow, click to select it.

10. You should have two green lines. Click the equal constraint icon ![equal] to create an equality constraint between these two lines.

Note: this equality, combined with the horizontal and vertical constraints added by the rectangle tool force our geometry to be a square.

11. Now select he upper horizontal line again and click *Fix horizontal constraint* icon ![fix] . In the popup dialog enter 50 and click ok.

We now have a square that represents the top view of our lump of material. The dimensions are constrained to the required 50 mm and the sides are horizontally and vertically constrained. But, the square itself is not constrained with respect to the origin. It is not required that it be constrained to the origin, it will just make locating later features easier. There are many ways we could constrain the square to the origin. We will choose to minimize dimensional constraints and go with a geometrical constraint. Specifically, a symmetry constraint around the origin.

12. Hover the mouse over the lower left vertex and click to select. Do the same for the upper right vertex. And then the same for the *RootPoint* (this is the virtual vertex where the red X axis line and the green Y axis line cross and is the Origin discussed previously). Now click the symmetry constraint icon ![sym] . All the geometry will turn green, indicating the sketch is now fully constrained.

Note: The order of selection was critical. The last point selected, in this case the *RootPoint* is the point, around which symmetry is desired. The first two vertices, are what is desired to be symmetrical.

Note: Fully constrained implies it is constrained for the current state of the geometry. This means there may be other mathematical solutions possible. (This can be of import more complicated models.)

Note: We choose to minimize dimensional constraints, preferring geometrical constraints, because it produces more stable and less computational requirements. (The later becoming more important as the model gets more complicated.)
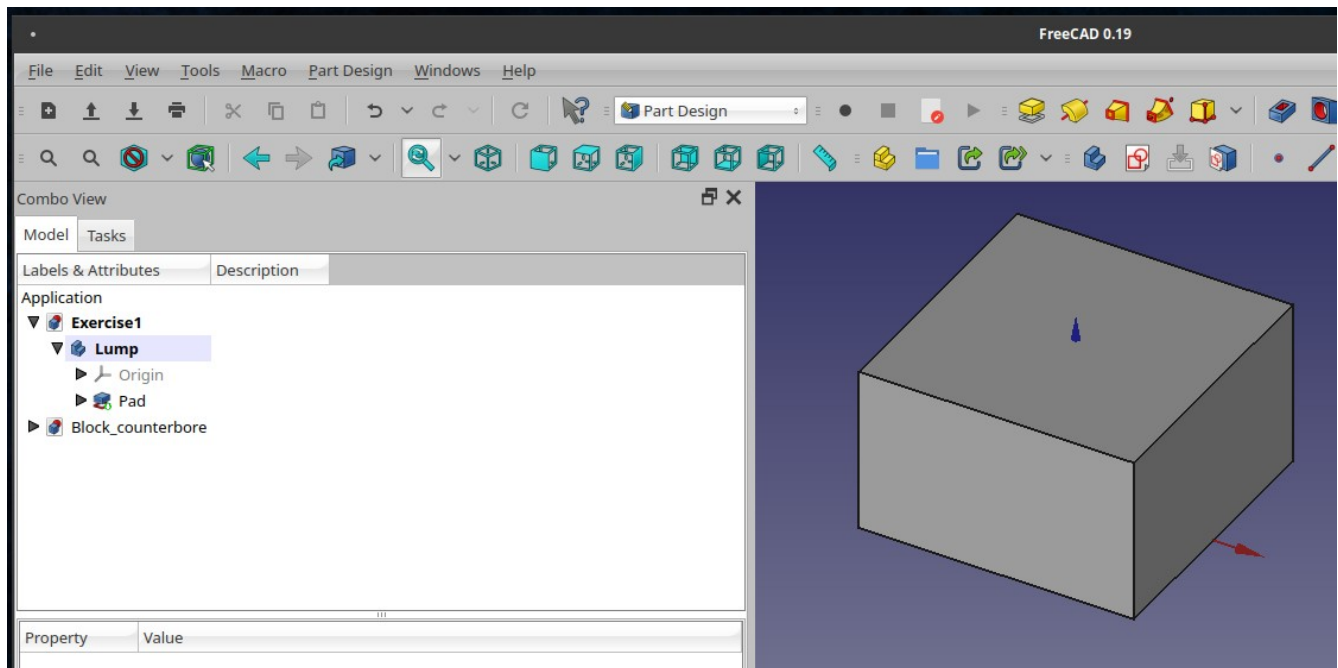
At his point, with the sketcher still open we have:
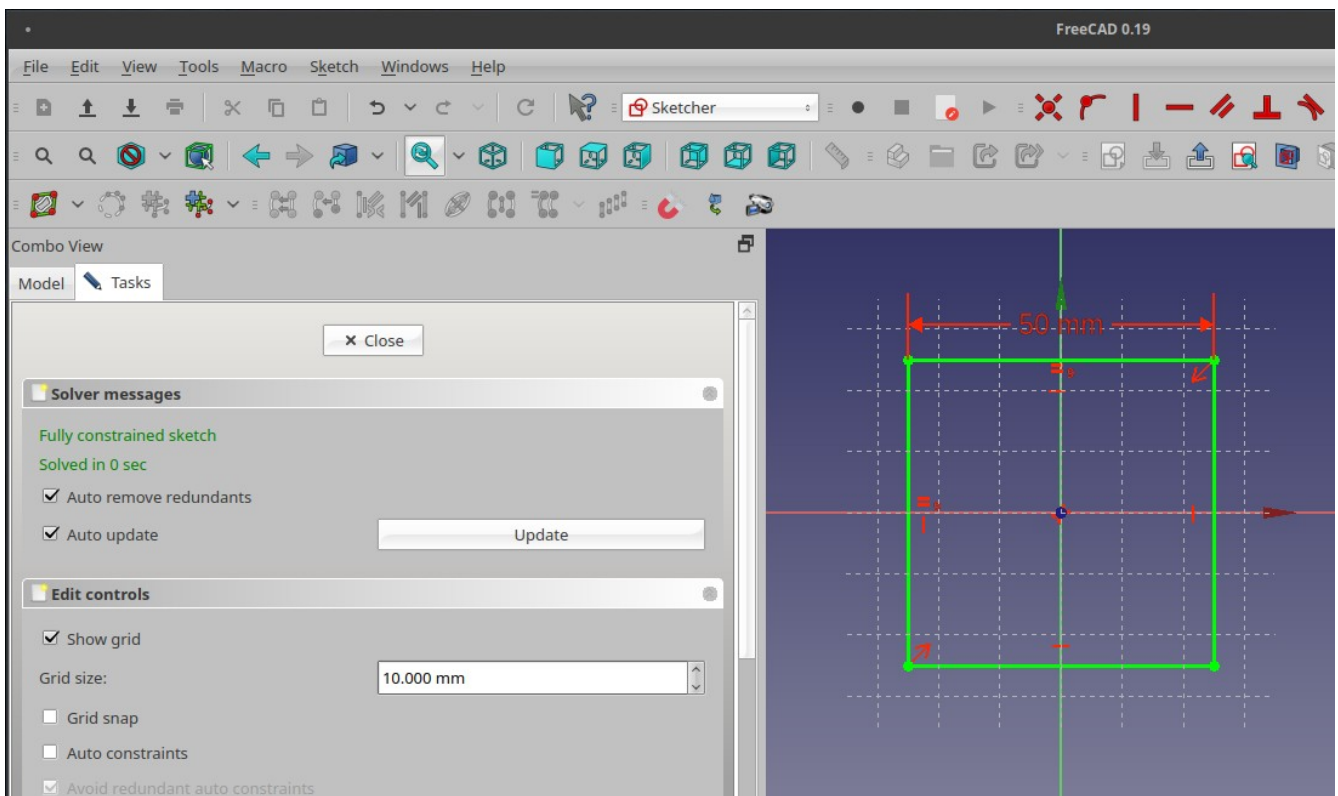
13. Now click Close and we have this (I toggled the axis cross back on):
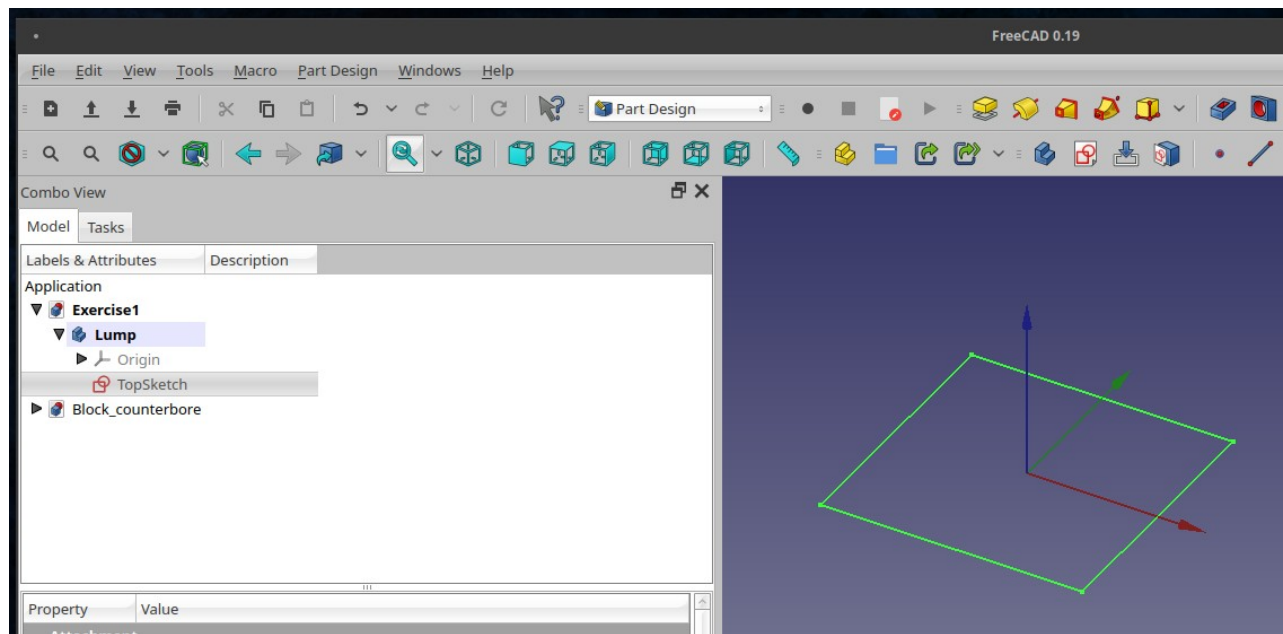
14. With TopSketch selected click the pad icon ⬛ and for the length enter 30 and click OK. We now have a 50 mm x 50 mm x 30mm lump of material.

Making holes is called pocketing. In more technical terms, it is a subtractive process. Bottom line, we remove material to create the voids where we need them.

In the real world, if we want a round hole we would find a drill the correct size mark the center of the hole and drill. For the counterbore we might drill with a larger drill to the desired depth. In our model we could create another sketch with the smaller hole and pocket the hole through the material. We could also include the circle describing this hole in our current sketch. This would create a round void it the lump. Then create an additional sketch for the larger hole, position the sketch appropriately, and pocket to the desired depth.

In the virtual world of CAD we have additional options. If we revolve the profile of the counterbored hole, we can accomplish the counterbore with a single sketch.

There are other possibilities as well and there may be good reasons to choose one route over another. The two round pocket process is good if you have only round holes or if you might be making non-cylindrical holes. For example something that would require a pocket milling operation. See Figure 2: Complex Pocket.
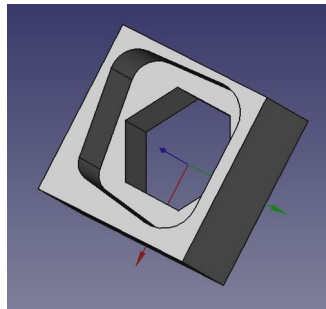
Figure 2: Complex Pocket

The revolution process will allow complex profiles, that in general aren't doable on a milling machine, but are easily done on a 3D printer. See Figure 3: Complex Bore.
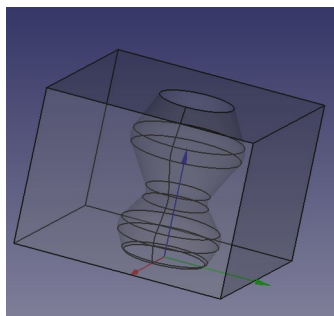

Figure 3: Complex Bore

Now we'll proceed to make the counterbored hole. We could drill them (or pocket in FreeCAD terms), but that would require adding two more sketches, one for the center hole (yes, this could be added to the sketch we just did…) and one for the counterbore. But, we can do it in one sketch if we use a revolution. The revolve will also allow the introduction of techniques that will prove useful for future projects.

15. Make sure the sketch is closed. Click the new sketch icon and select the XZ_plane (front) to continue.

16. Click OK and the sketcher will open. Now click the Create polyline icon and sketch out the general shape of half the cross section of the counter bore.
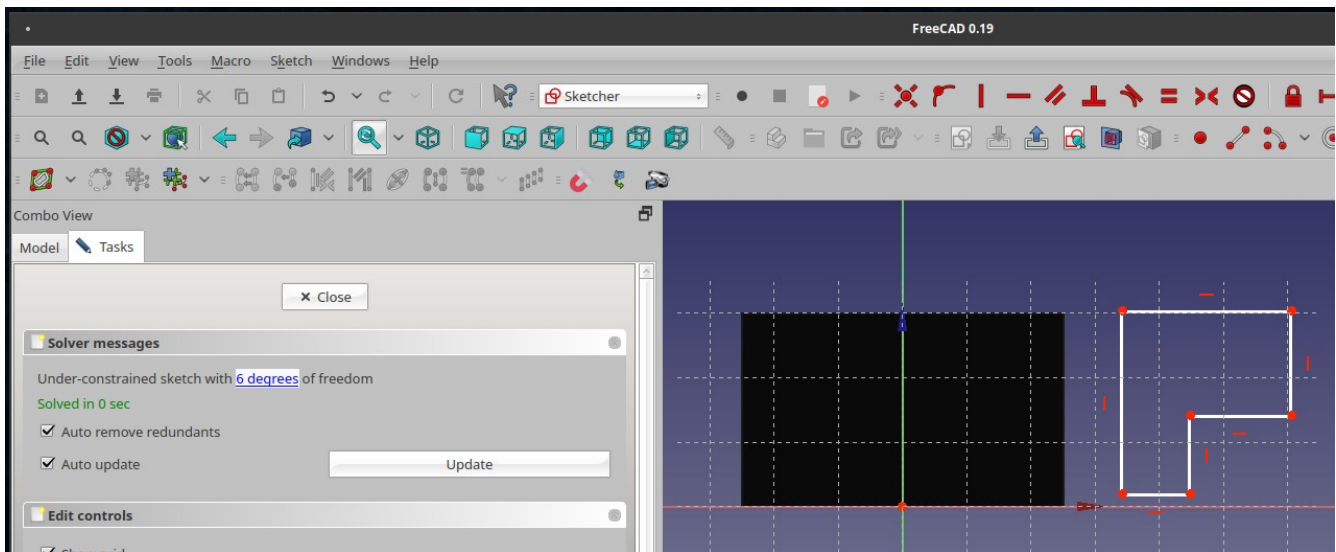
Let the tool add the horizontal and vertical constraints by clicking when the constraint indicator shows up on the cursor. Make sure on the last point (that will close the shape) that the coincident constraint indicator is shown. (If the last line is not vertical/horizontal, select the line and add the appropriate horizontal/vertical constraint.)

17. Now we need to locate, dimension, and constrain the profile within our lump. From our drawing we know the counterbore is 36 mm diameter and the through hole is 24 mm. Select the top horizontal line and click the Fix horizontal icon and enter the 18 (we want the radius). Do the same for the lower horizontal line and enter the radius of the through hole, 12. Put vertical constraints on the left and right vertical lines with the overall height of the lump and the depth of the counterbore from the drawing.
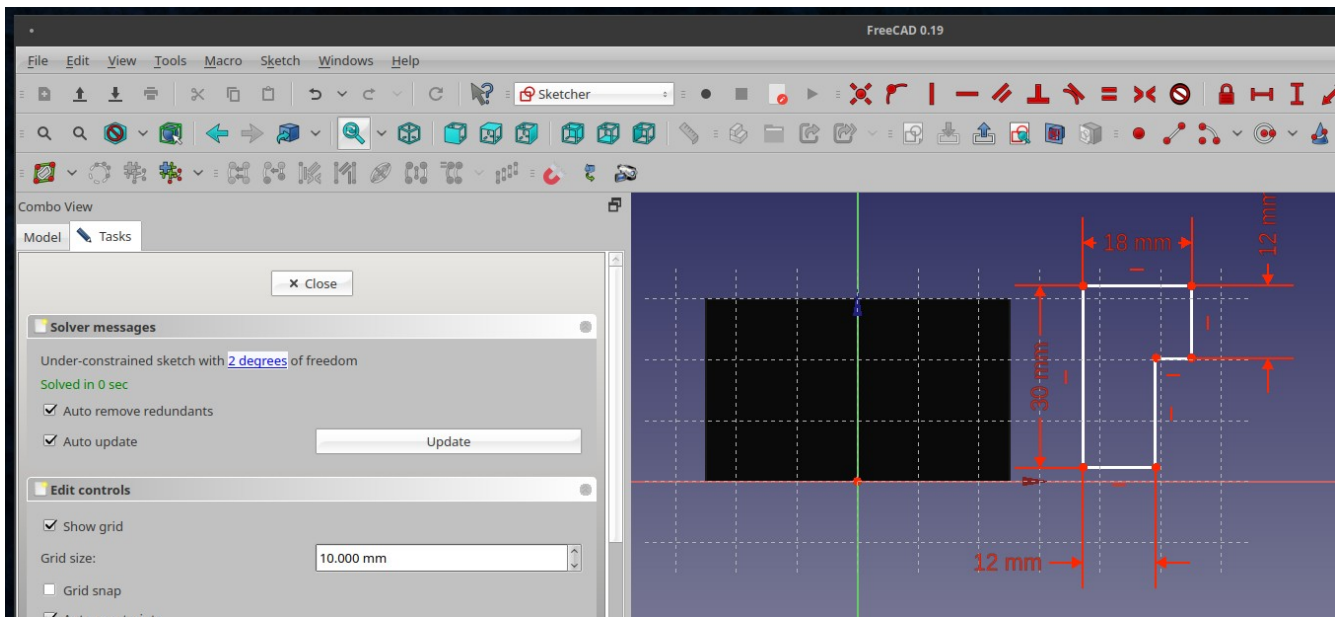
At this point, if you attempt to move any of the lines (by clicking on them and dragging) the entire shape will move, but not deform. You'll not there are still 2 degrees of freedom shown though. And, indeed the shape can still move up/down and left/right. (And in my example, the shape is not within the lump we already created (shown as a black rectangle...thanks to my PC's render engine.)

18. To finish this shape we will constrain the lower left vertex to the origin. Select the lower left vertex and the *RootPoint*. Then click the Create coincidence icon ✖ . And we get the all green shape and the Fully constrained message.
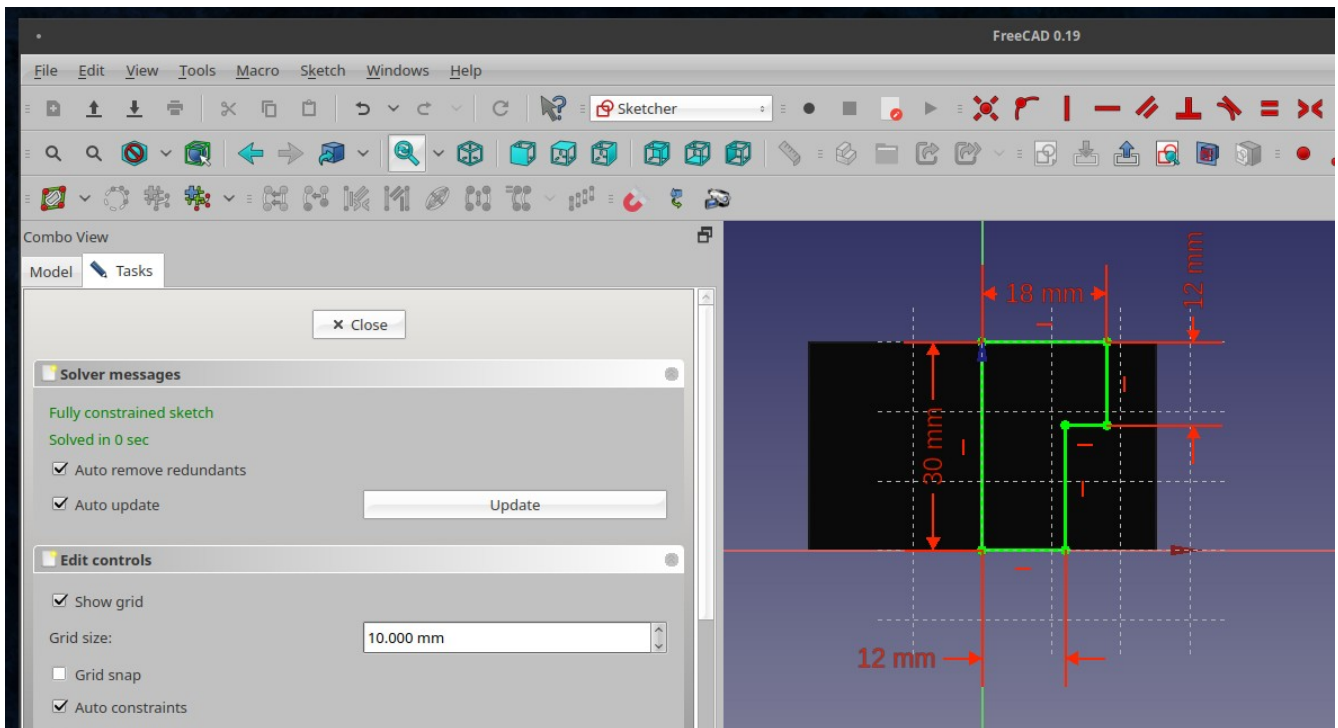
19. Click Close.

20. While we're here let's rename the Pad and Sketch001. Rename Pad to LumpPad and Sketch001 to CounterboreSketch

21. Now we will make a subtractive revolution from the CounterboreSketch. Click the CounterboreSketch to select it, then click the groove icon .
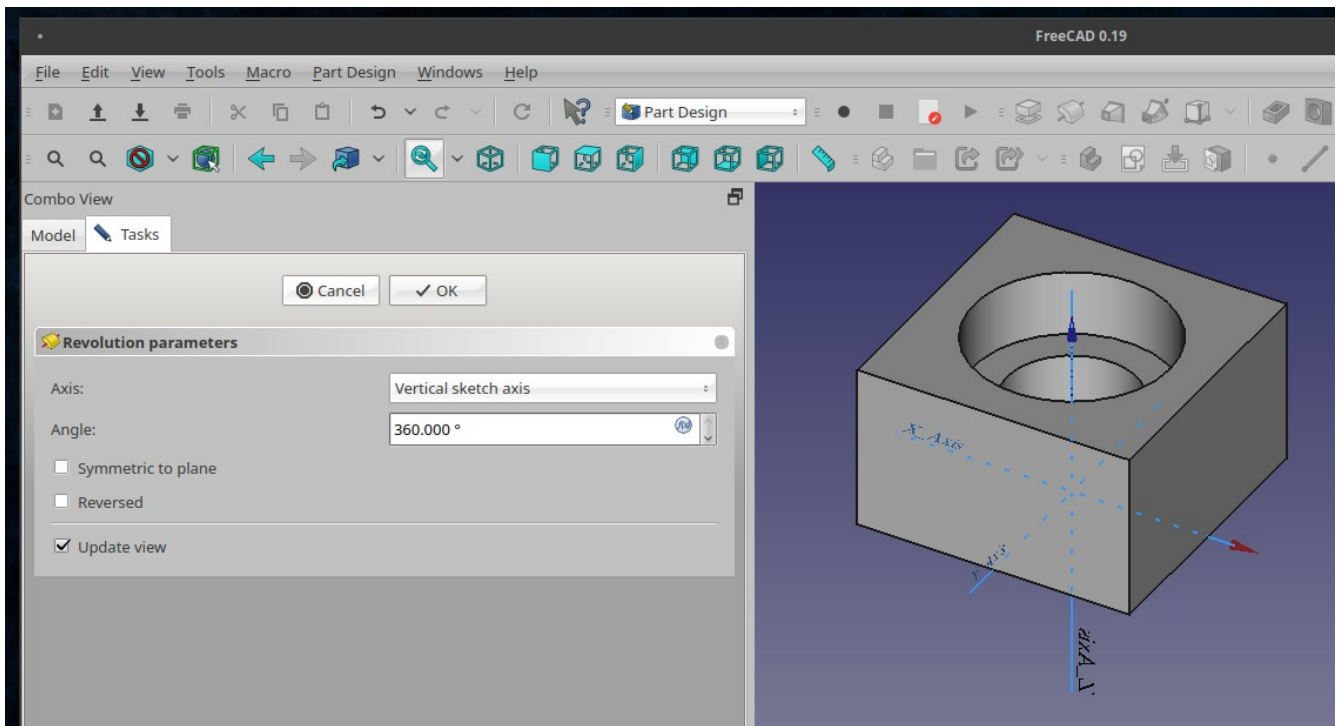
The defaults have provided exactly what we want. So we need not change any settings or values. So click OK.

22. Just to be tidy, we will name the groove feature CounterboredLump. And save the document.
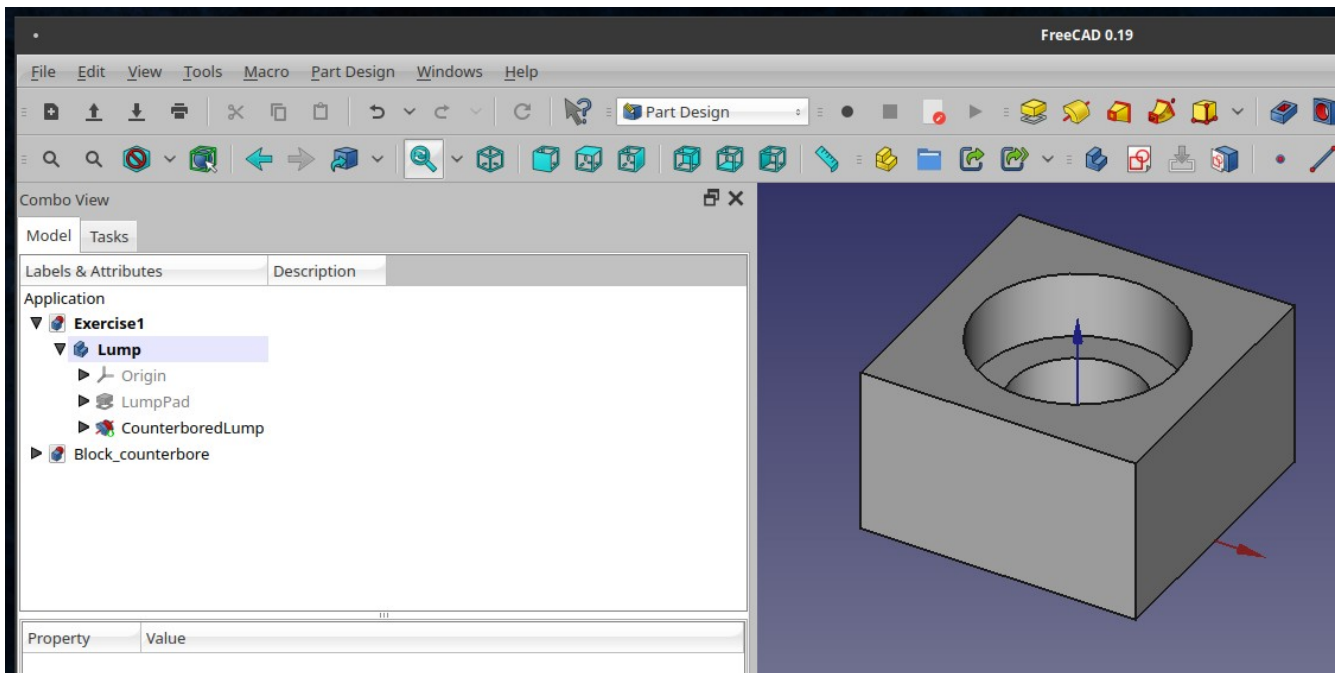
As we mentioned earlier, this is not the only way to accomplish this model.

Let's look at another possibility.

Open the Exercise1.FCStd document and save it as Exercise1-2.

We can delete features in the model tree by simply clicking on them and hit the delete key. (Or right click on the feature and select Delete.) Delete the CounterboredLump and the CounterboreSketch so we are back to only the LumpPad (and below it TopSketch).
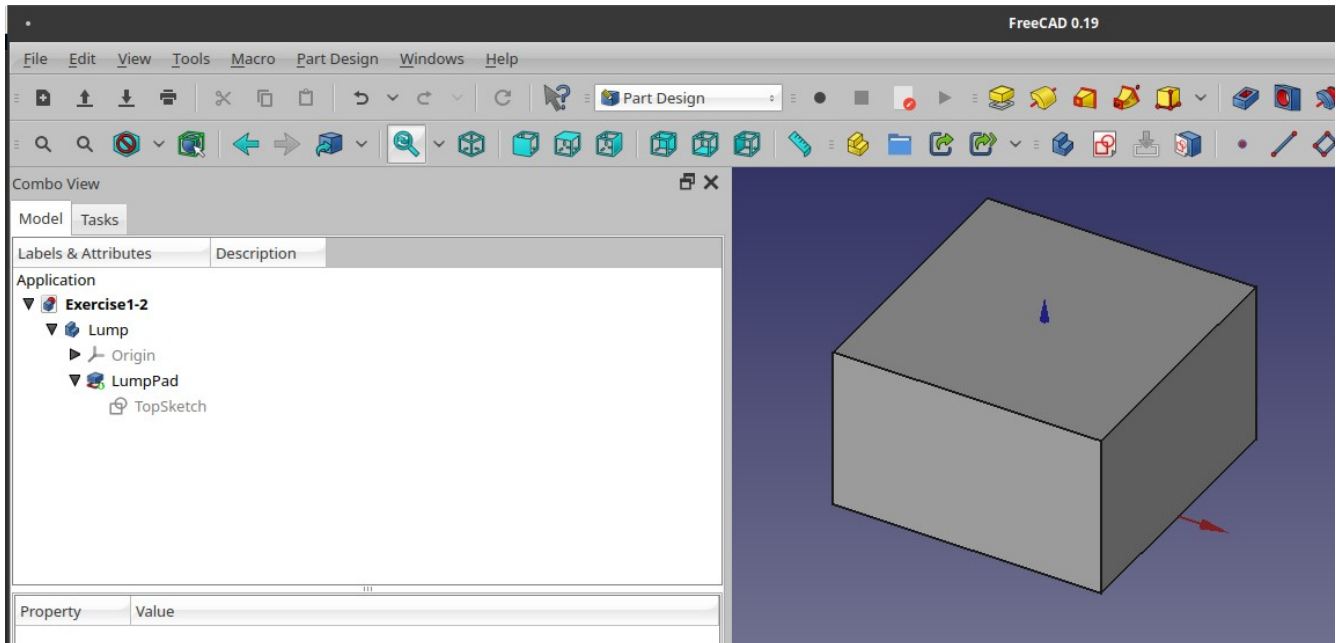

Now double click on the TopSketch to open it in Sketcher.

1. Click on the Create circle icon ⊙ and click somewhere inside the square we sketched before. This sets the center point of the circle. Move the mouse and drag a circle and click to set the circumference. Don't worry about the size we'll fix that shortly.

    1. Note: the green lines have disappeared since the sketch is no longer constrained.

    2. If we click on the circumference and drag the circle will change size. Click on the center point and drag and the circle will move.

2. Now select the center point of the circle and the *RootPoint.*

3. Click the Create coincidence icon ⨯. The circle is now constrained to the *RootPoint*.

4. Now click the circumference

5. Click the the down arrow next to the Fix radius icon ⊘. In the resulting drop down menu select the Fix diameter icon. Enter 24 and click OK.

1. Note: we used the diameter because we know from the Figure 1: Mechanical Drawing that the hole through the center of our lump is to be 24mm.

2. Note: the sketch is once again fully constrained.

6. Click Close to exit the sketcher.

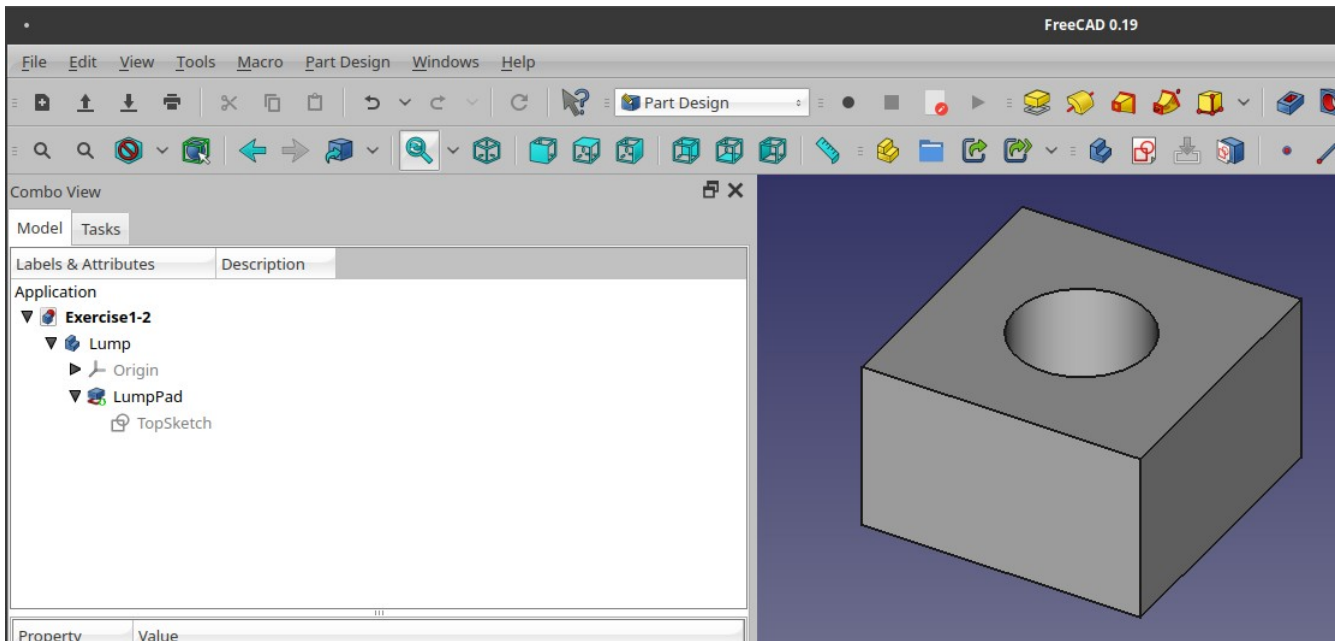1. Note: Don't be temped to add a circle representing the counterbore diameter. You can only have shapes nested one level deep.
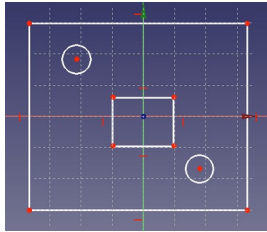
2. This is ok: This is not:
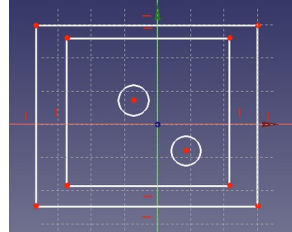
3. When we close the sketch we can see the Lump has been updated with a hole:

7. Now let's do the counter bore. Click on the top face of the LumpPad to highlight it and then click the Create new sketch icon

1. Note: no prompt to select the plane for the sketch. That's because the sketch is being attached to the top face of LumpPad. This sort of attachment is fine for our simple case, but, can cause problems in more complex models when we want to go back to a previous feature in the model tree and change it. (We'll discuss how to avoid this in more detail later.)

8. Now draw a circle outside the LumpPad shape and constrain it's diameter.
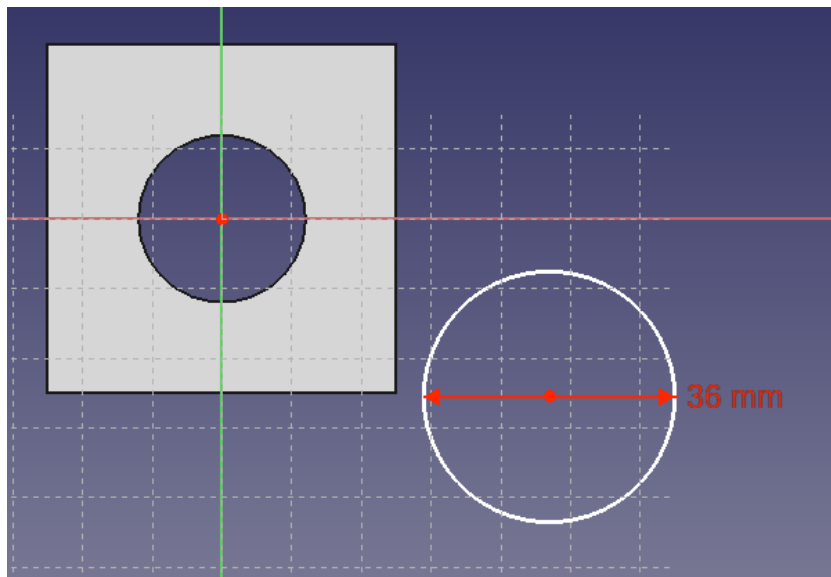
9. Now select the center point of the circle and the *RootPoint* and create a coincidence constraint.

10. The sketch is now constrained, so Close the sketch.

    1. Note the circle is on the top face of LumpPad.

11. Click the Pocket icon and enter 12 for the Length.

    1. Note: the pocketing took place in downward direction. Pocketing acts in a negative direction along the normal of the sketch plane. If this sketch was on the XY_Plane, then we'd have to check the Reverse checkbox in the Pocket dialog. This would cut the counterbore in the bottom of LumpPad.

12. We now have created the same model:
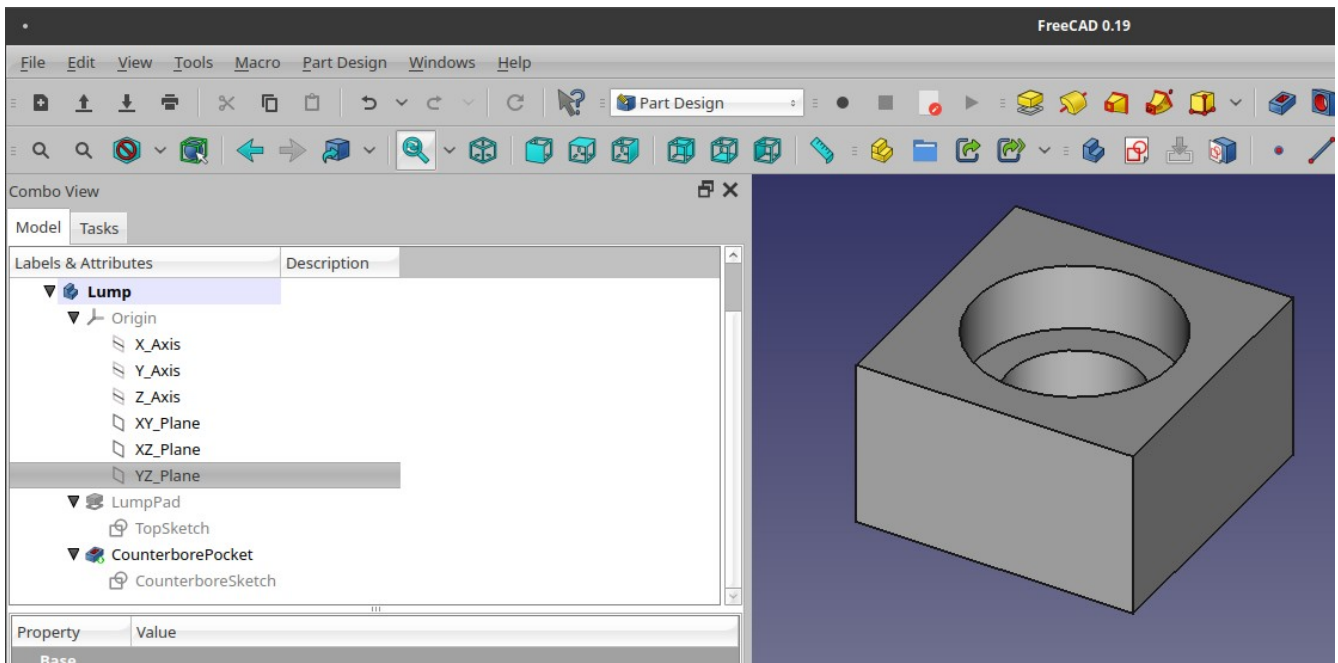
As mentioned above attaching sketches to model geometry such as faces and edges can cause problems in complex models. If a change to a previous feature causes the face or edge to be renumbered subsequent attached sketches can lose their attachment. This is known as the topological naming issue and is a well documented problem. Attempts are in the works to address this in FreeCAD, but the issue comes from the OpenCascade engine and not FreeCAD.

There techniques to address this and we'll see those used later in this series.

In the next installment we will create the drawing from Figure 1: Mechanical Drawing.