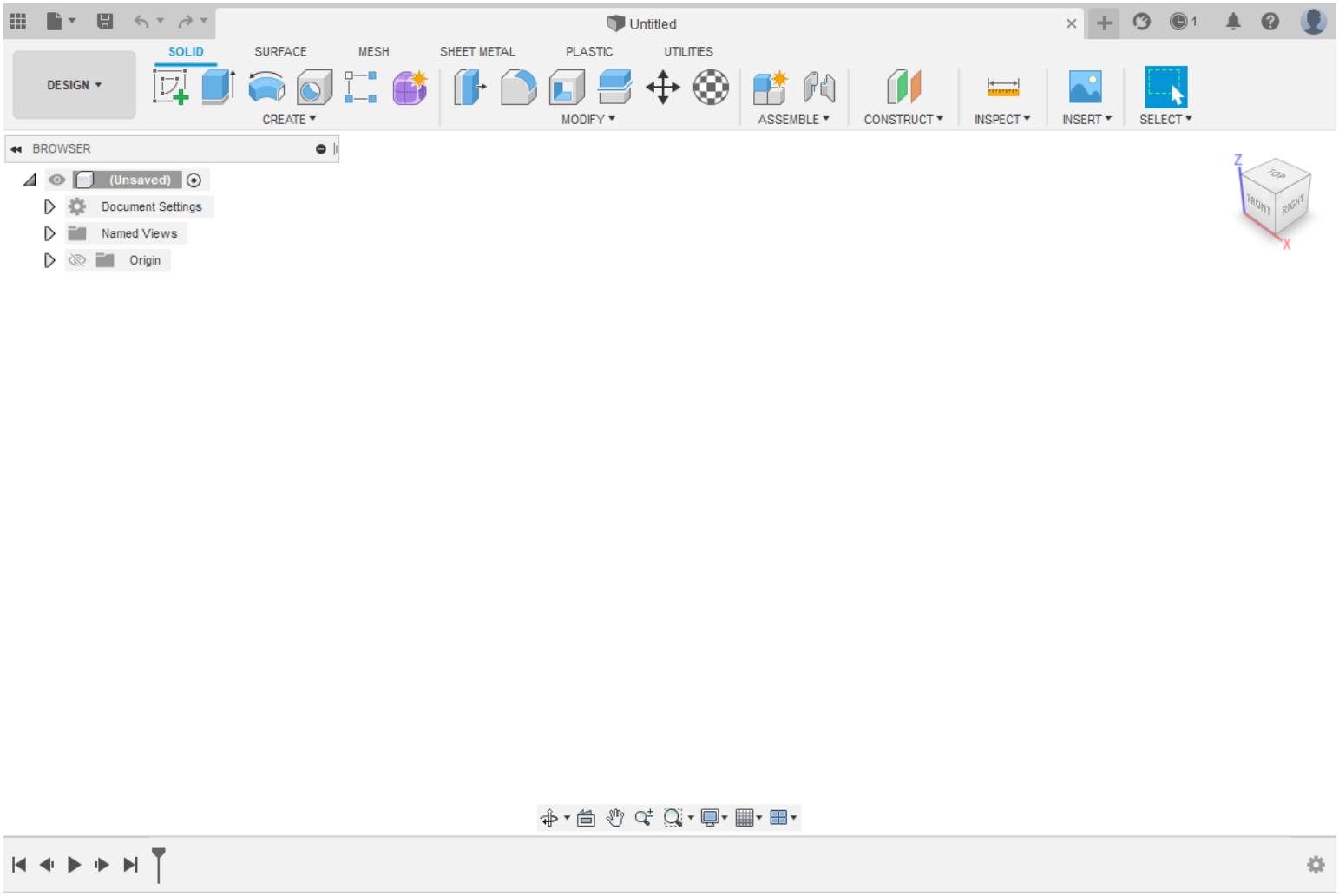
Fusion360 Starter

Open Fusion360 - You may be prompted to log in to your autodesk account using your email and password.
Once the software opens,

You may also be prompted to create your own team. Your team is just you. We will not be joining any other teams for the class.

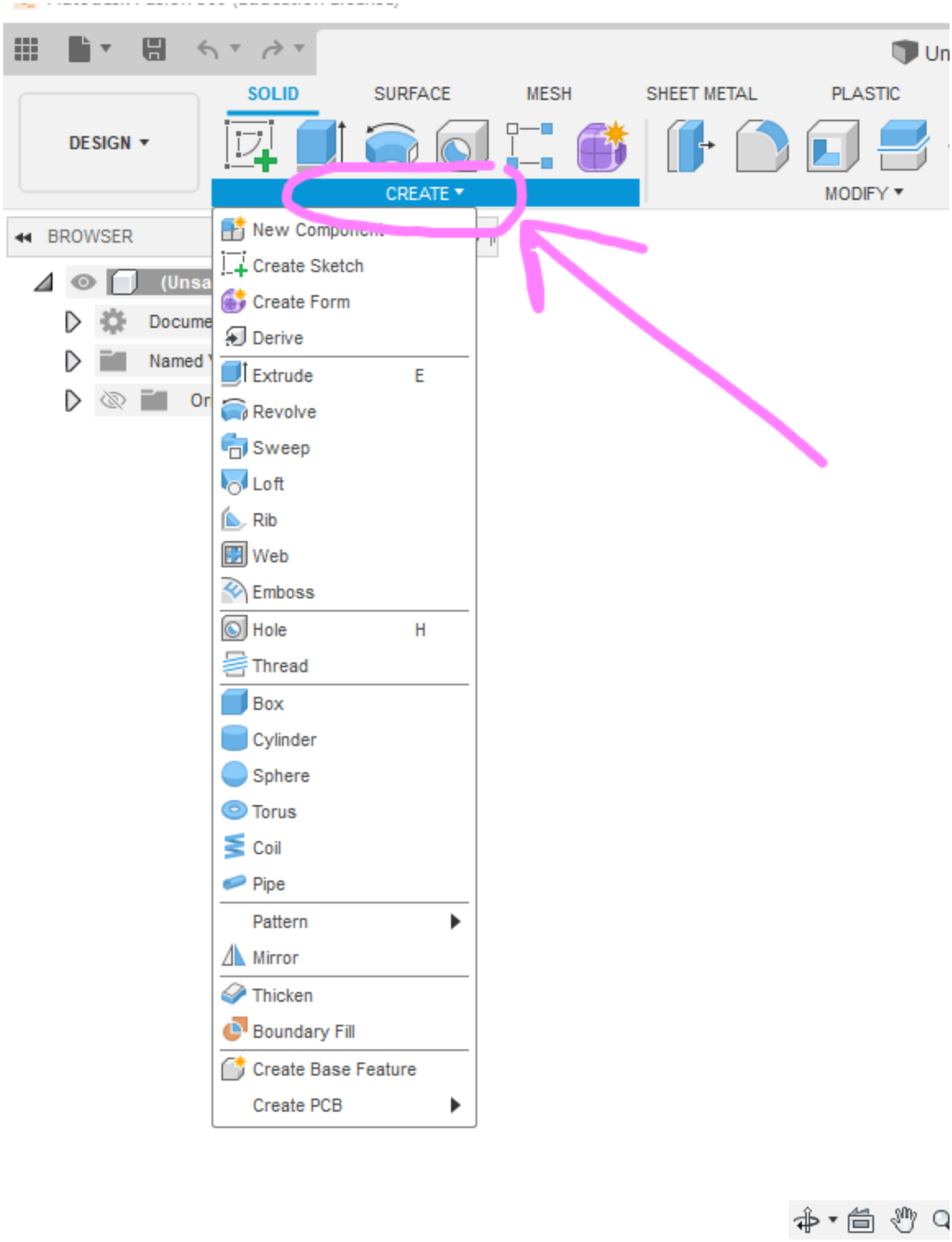
The software interface looks like this



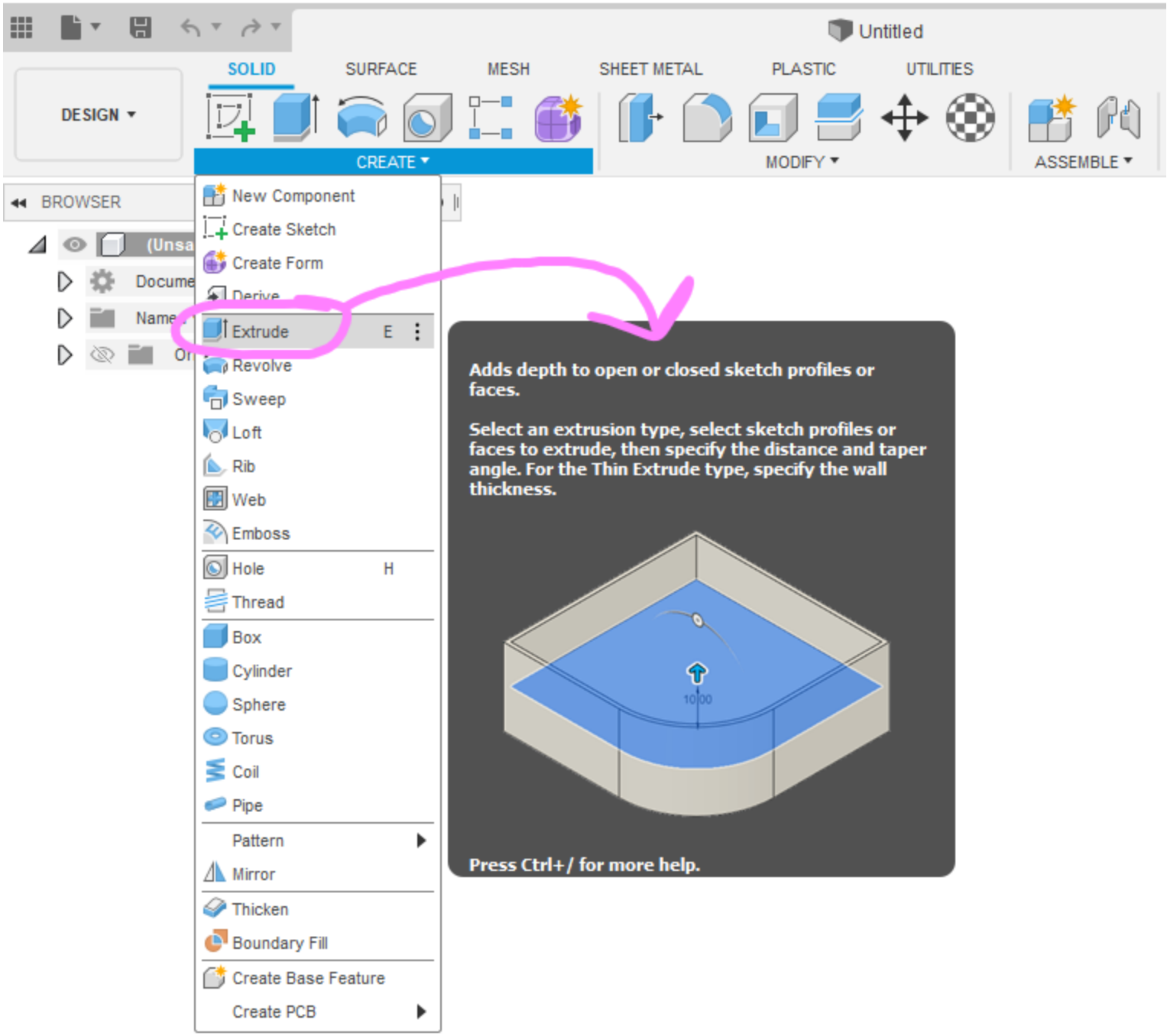
Click on the squares in the top left to open the projects menu.

The project menu will open on the left side of the screen.
Click the grid again to close the menu.

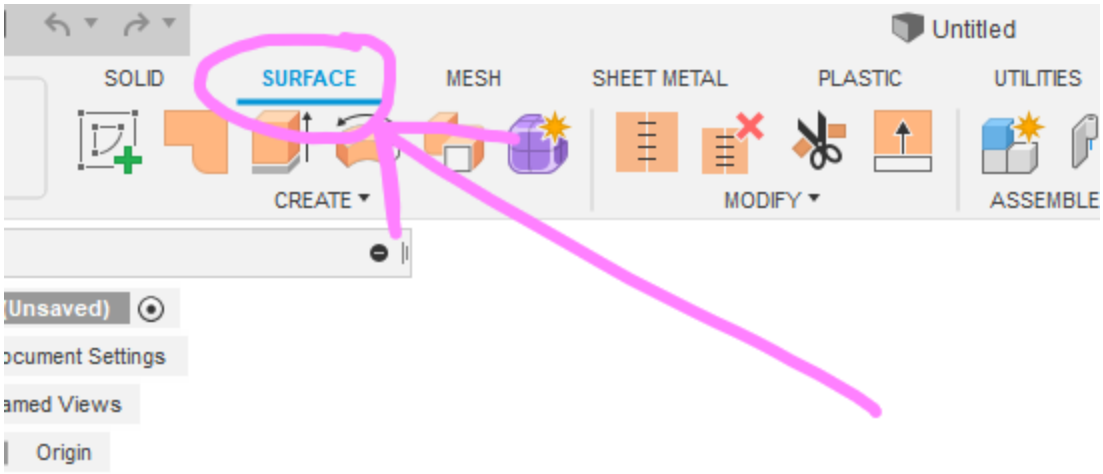
Note: All fusion projects are saved in the cloud rather than on your computer, therefore this menu is different that what we are used to with traditional software.
Back in the design interface, lets explore the various tabs and dropdowns available to us.
Click the “Solid” tab, and brose the options.
Hover your mouse over an icon to see the pop-up description.
Click the “Create” dropdown to see the full list of commands/options



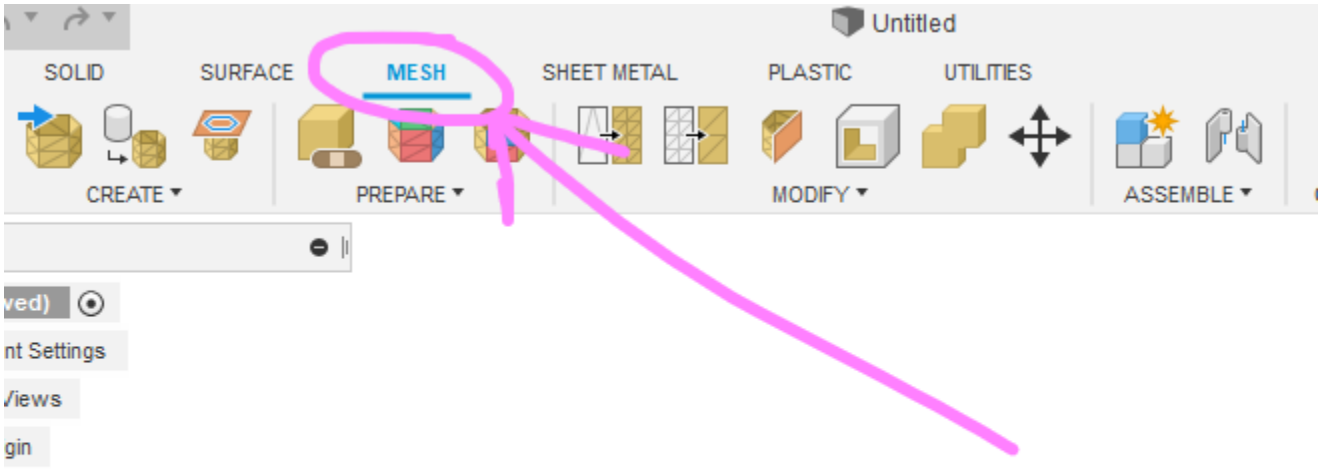
Hover over the extrude command/option in the list to see a detailed description of what it does.



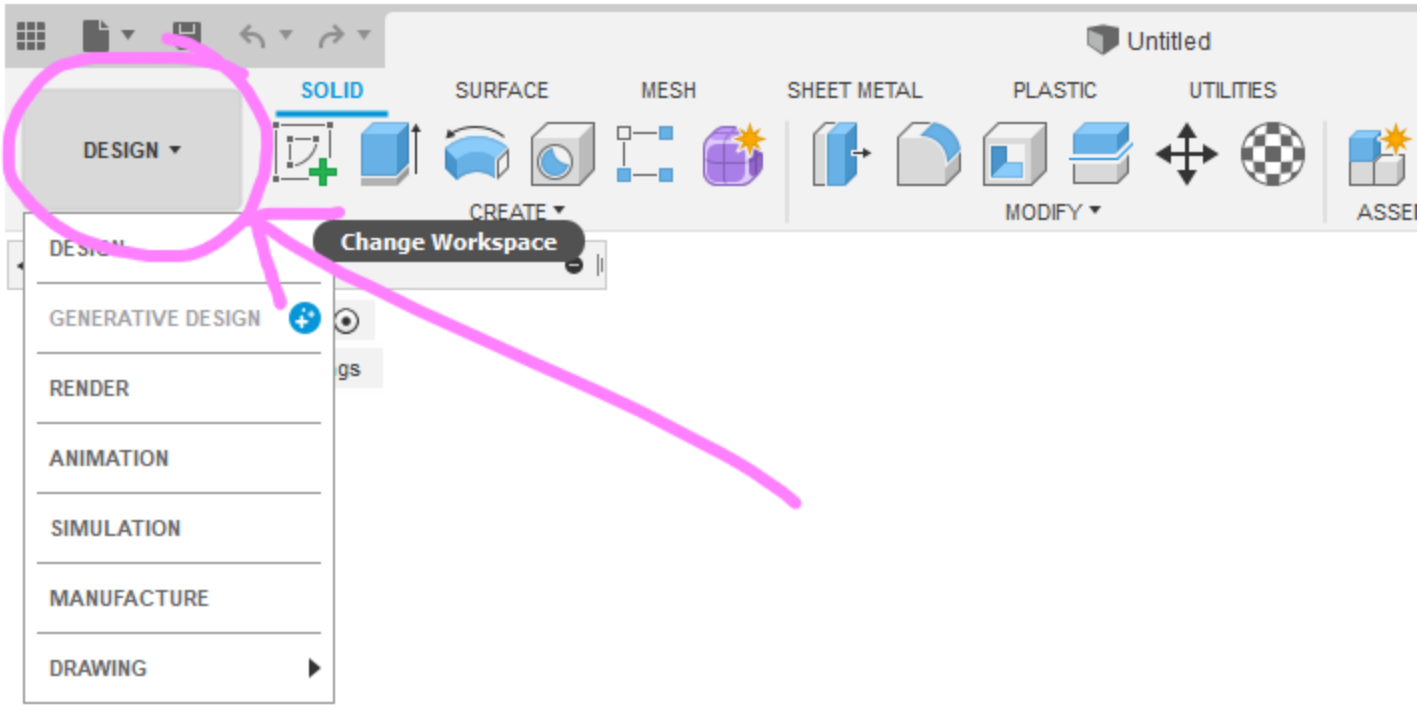
We will get more into this in a moment.
Notice that there is a “Create” dropdown menu and a “Modify” dropdown menu. These menus correspond to the “Solid” tab and are only good for solid objects.
Let's switch to the “Surface” tab.



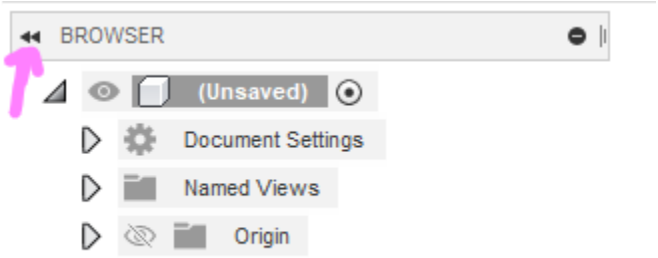
These tools are for surface type of objects (ie not solid).
Next click the “Mesh” tab



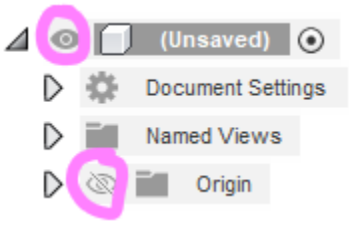
These tools are only for mesh objects.
Next, quickly browse through the “Sheet Metal” “Plastic” and “Utilities” tabs to give yourself an idea of what they contain.
Let’s return to the “Solid” tab as out home base
Click the large DESIGN button on the left to see the different workspaces available in Fusion360



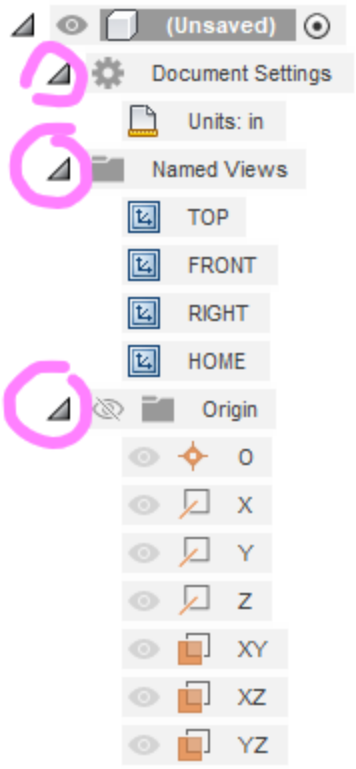
We will stay in the DESIGN workspace for now.
Below in the open working area, is the browser. Click the small arrows to hide/expand the browser.



Most of the work that we do will involve the browser.
Browser items can be shown or hidden by pressing the eye icons



Click the arrows in the browser to expand all of the sub menus



Now hide them again.
At the bottom of the screen is the timeline. This will record every single move we make for later referencing.

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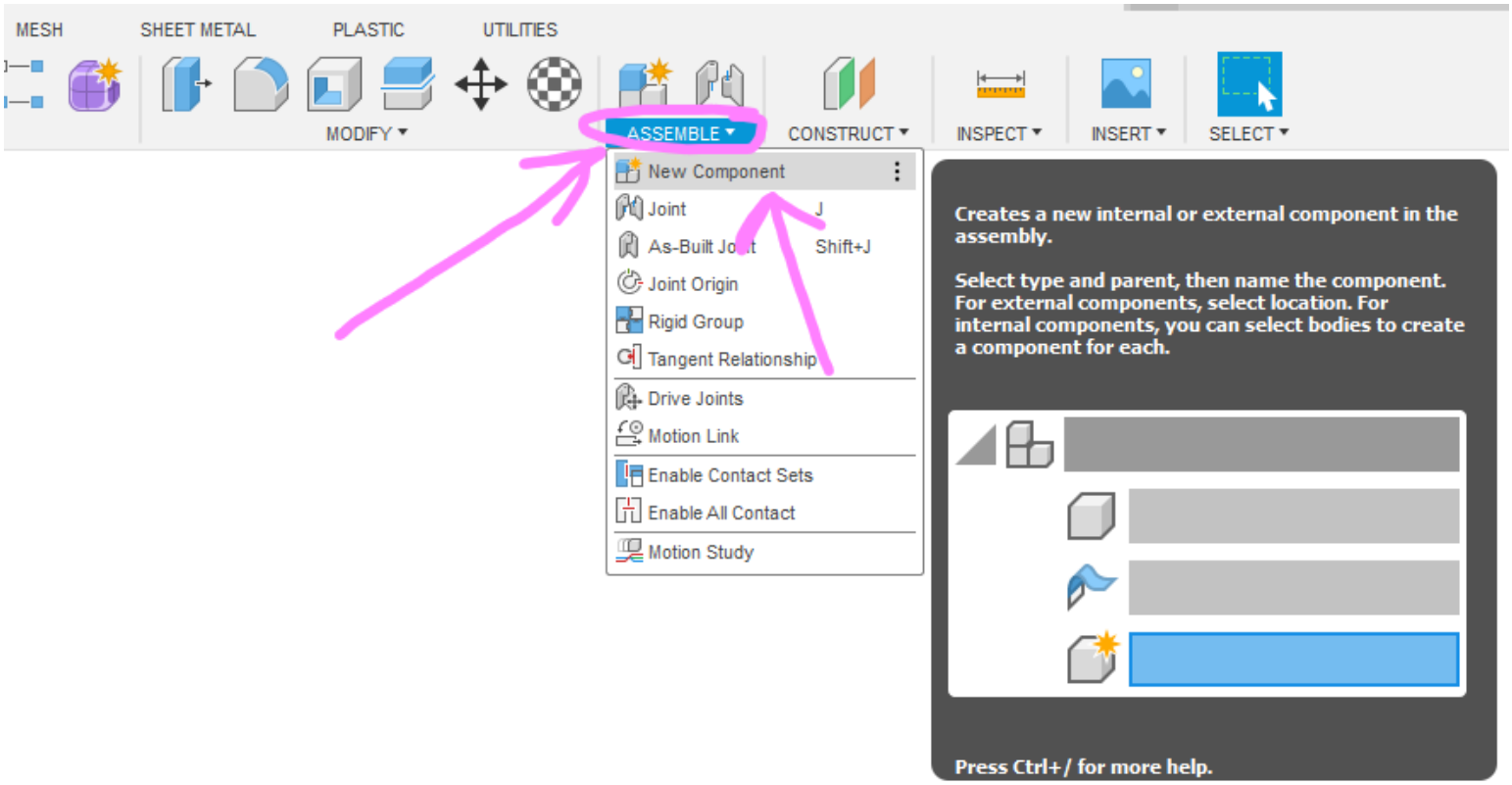
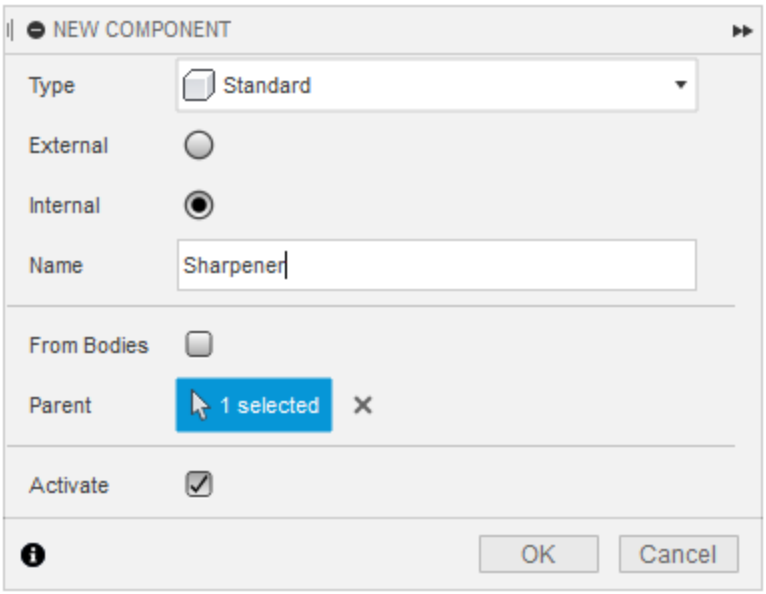
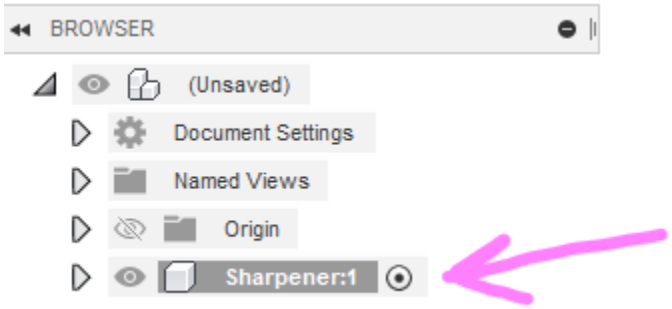
Let’s make our first component
Click the “Assemble” dropdown and click on “New Component”



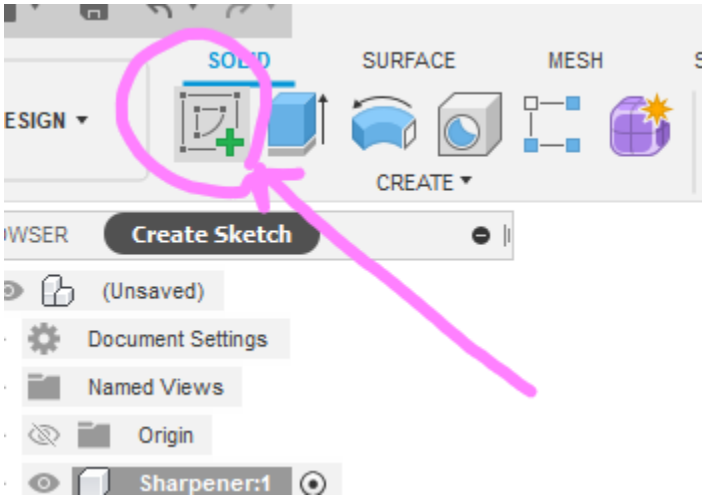
In the new component window that appears, give the component the name “Shareper”



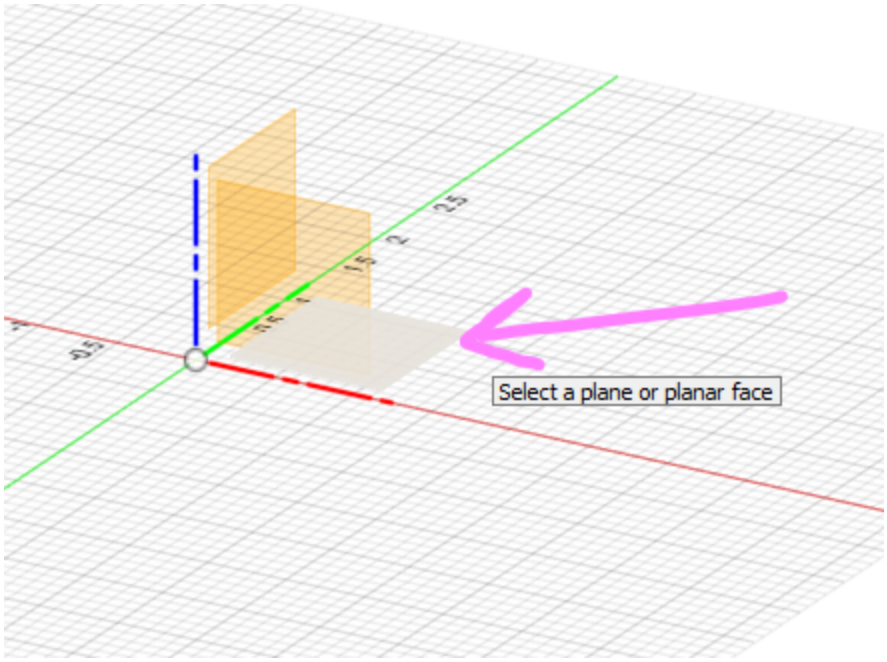
Your dialogue box should look like the above. Then click OK.
Our new component is now visible in the browser and it is actively selected



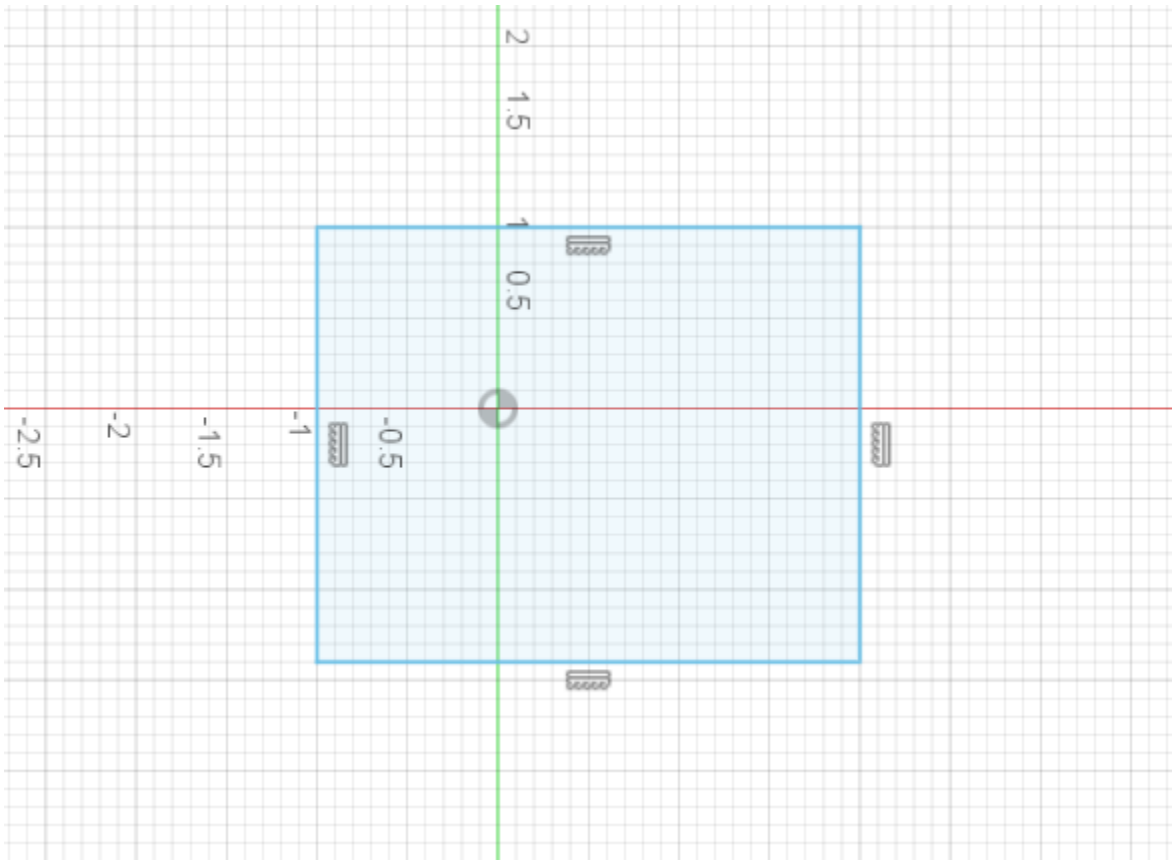
Next, click the “Create Sketch” button in the menu above. This should be under the “Solid” tab



Click the “Floor” grid for our new sketch to live on

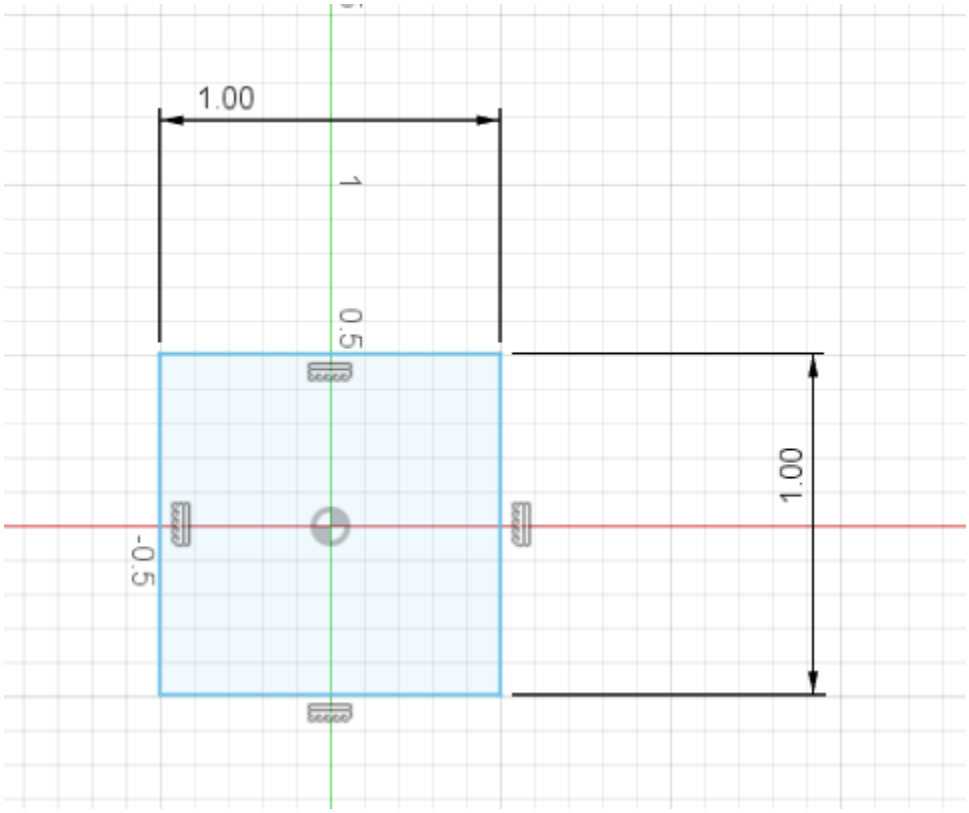


We are now inside of “Sketch” view (or mode). Notice the icons and menus change.
Sketch mode is a 2-dimensional mode only and we draw the lines here that we want to use to affect our final geometry.
Let's draw a small rectangle that will serve as the body for our sharpener.
Click the rectangle button in the menu above.
Click the starting point of the rectangle and then click the ending point of the rectangle. It does not need to be centered, and the size does not fully matter yet.

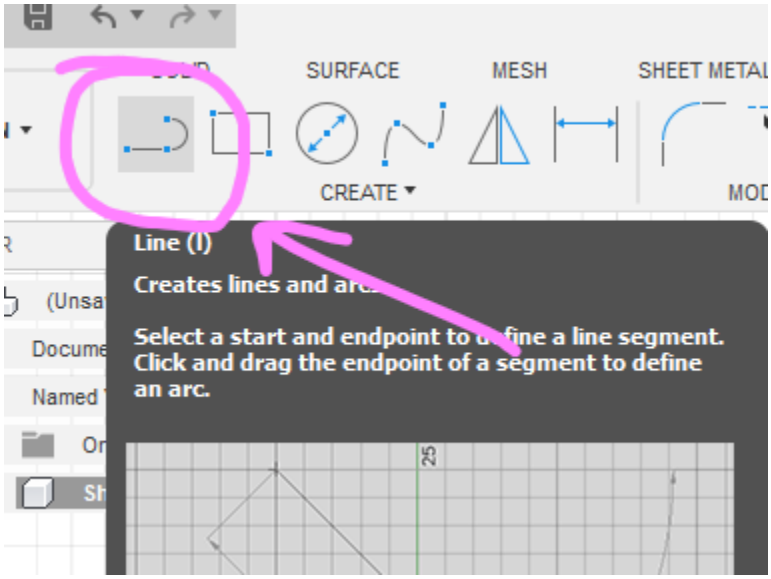


With The rectangle drawn, it looks blue. Next let's define the actual dimensions

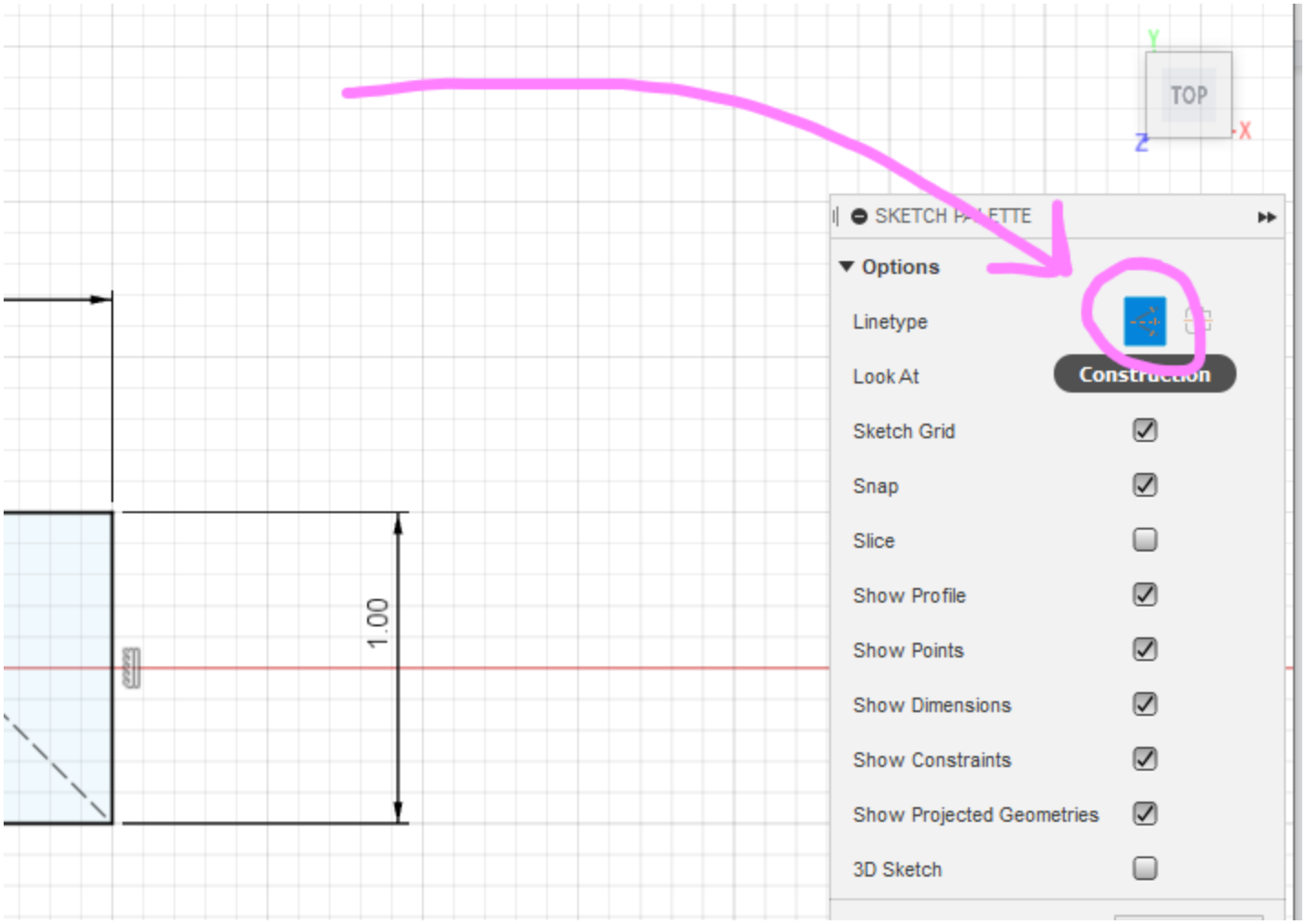
Press the “d” key on your keyboard and then select one of the edges. Draw out the dimension values away from the rectangle and click to place the dimension lines. Type in 1” for the dimensions of two sides of the square. Like this:



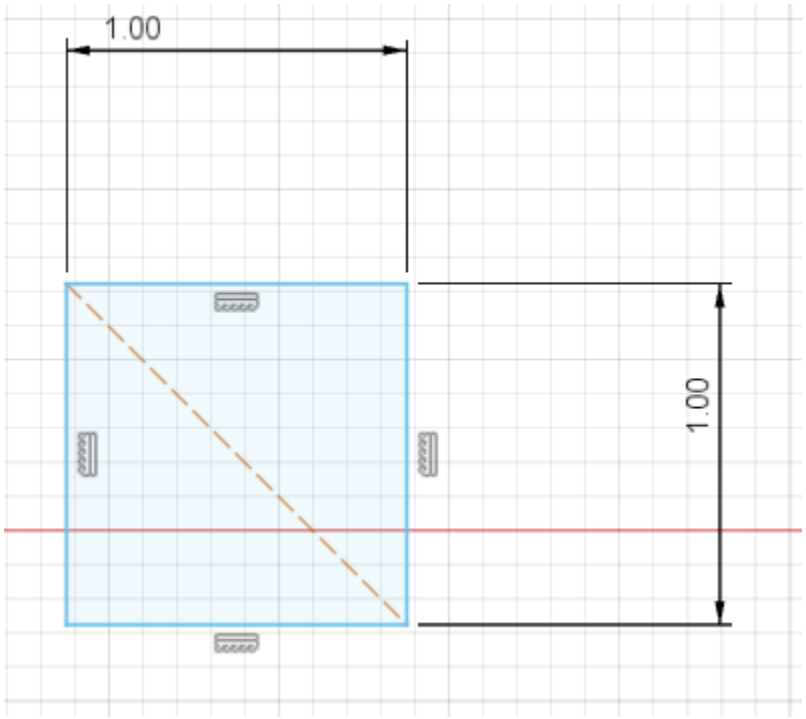
Try to keep the square somewhat central over the origin of the grid. To move the square, click the edge and drag it over to the center.
Lets center the square and constrain it to the origin.
Click the “Line” button



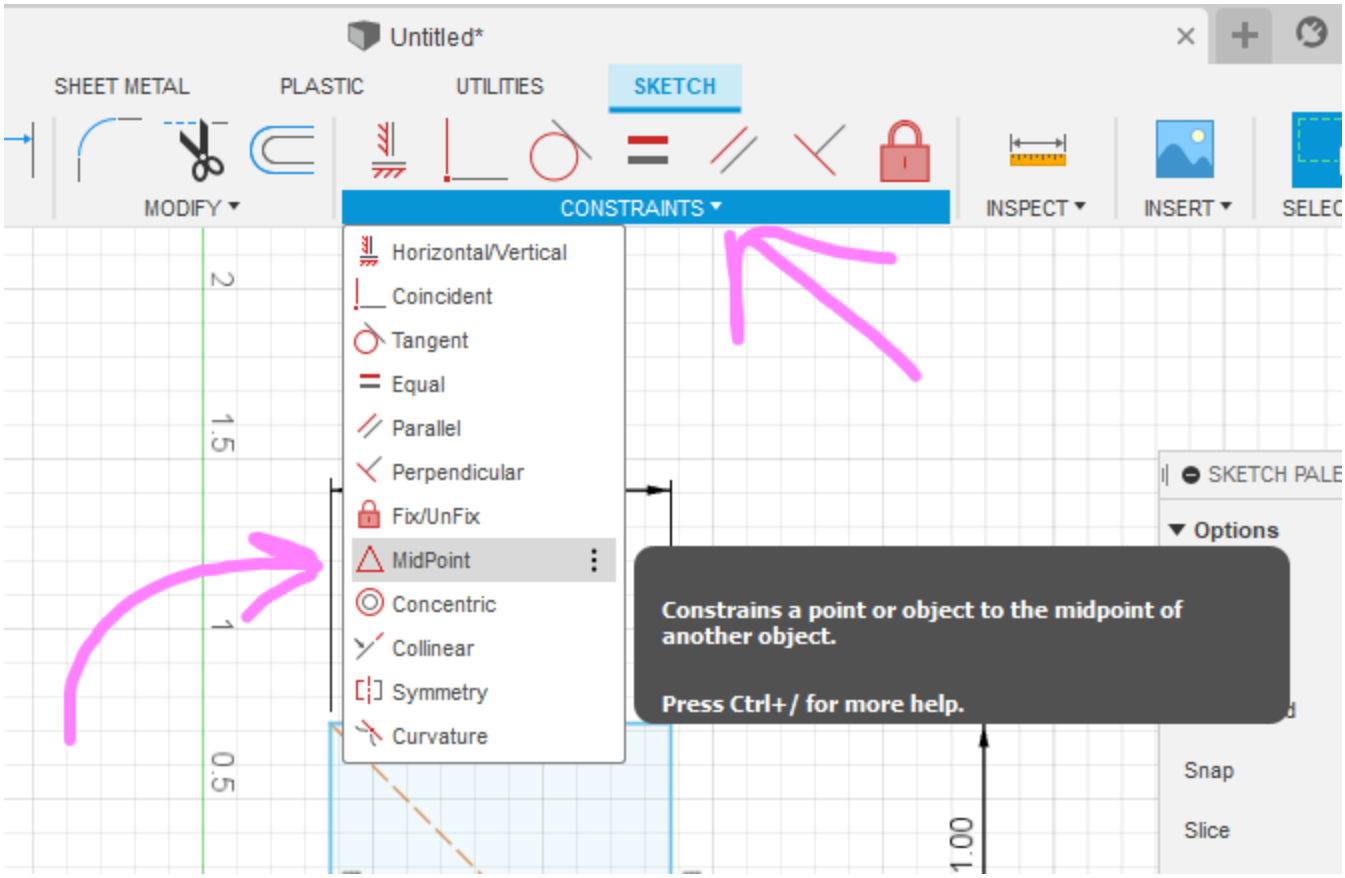
With the line command active, change the line type to “Construction Line” by clicking the small button in the menu on the right side of the work area:



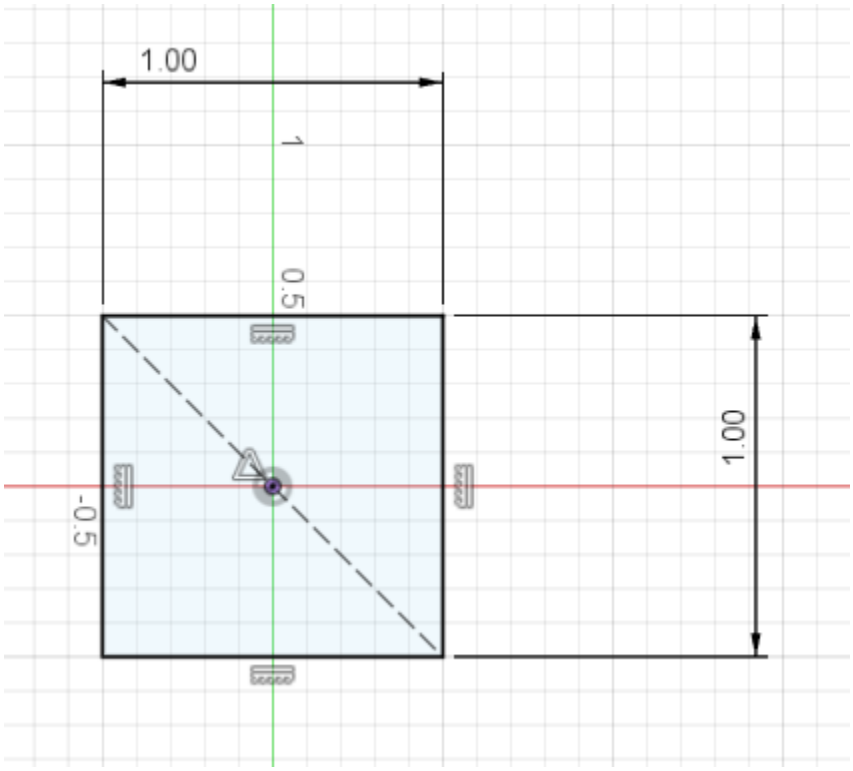
Construction lines are just guidelines for our designs.
Now draw a line from one corner ot the square to the other:



Notice the construction line is a dashed line with a different color.
Now lets constrain the midpoint of the diagonal line to the grid origin. Click the Constraints drop down menu and select the “Midpoint” option

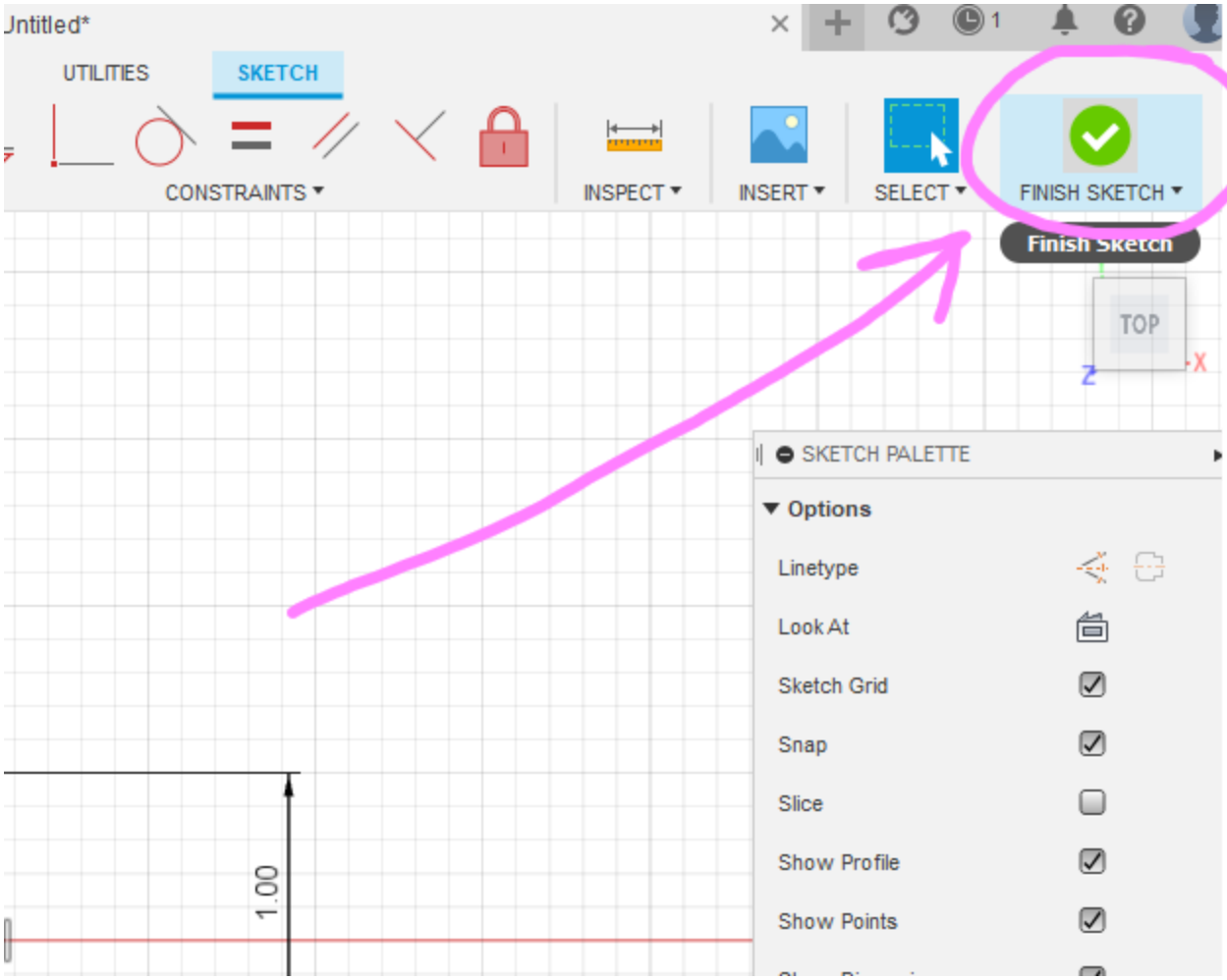


Next, with the midpoint option command active, first select the diagonal construction line, then select the origin of the grid (point 0,0). The square will snap to the grid origin.

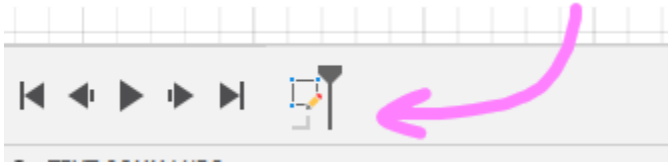


Notice the square edges are now black (rather than blue) which means that our square is fully constrained. More about this later.

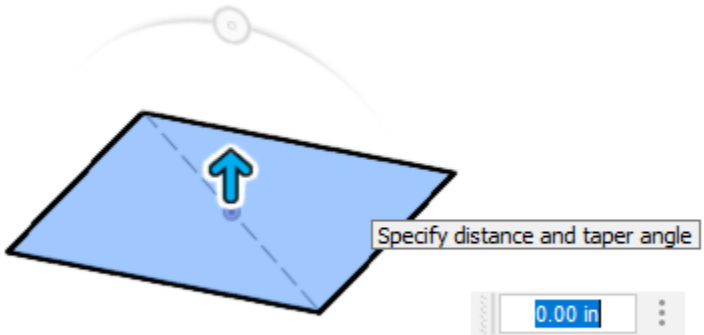
Click the “Finish Sketch” button at the top right of the workspace



The mode now exits Sketch mode and returns to the default Design mode view. Notice the icons change.
We have now created our first sketch!
The history of this action is reflected in the timeline below:

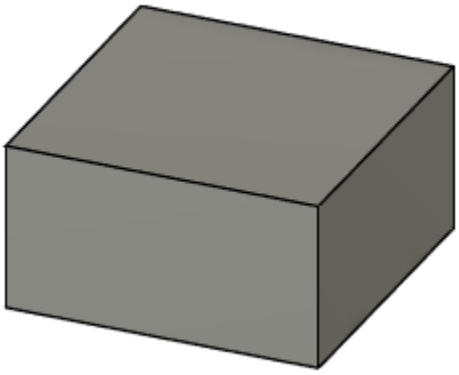


Now let's extrude our square to form the geometry of the sharpener object.
Press the “e” key on the keyboard to extrude. (select the square if prompted)

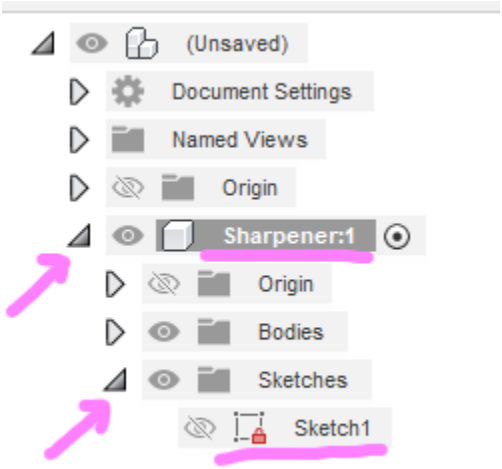


In the input field, type on the value 0.53”

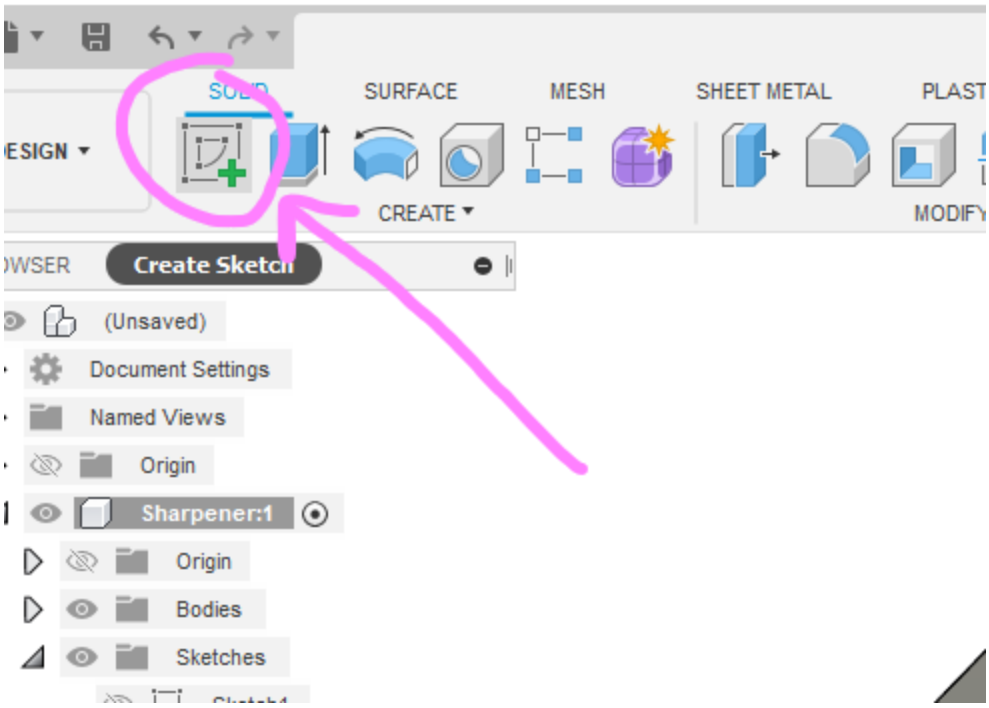
Our square is now 3d geometry



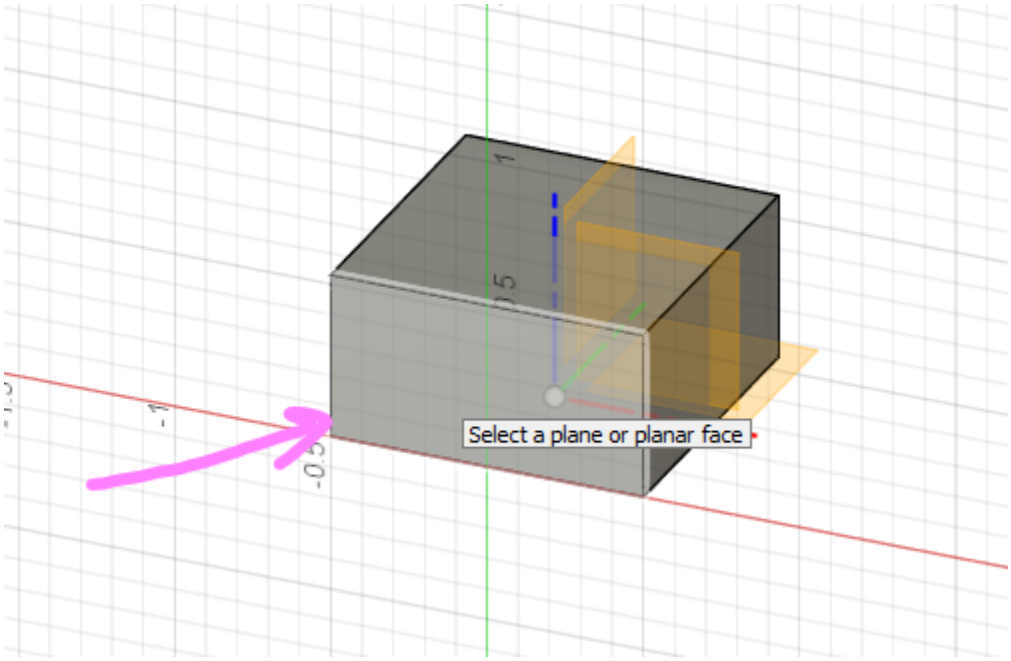
Note: The sketch we were working on is now hidden from view! It can become visible again by enabling it in the browser if needed. The browser contains all of our sketches, bodies (geometry), and other elements, and is constantly changing to reflect new actions.



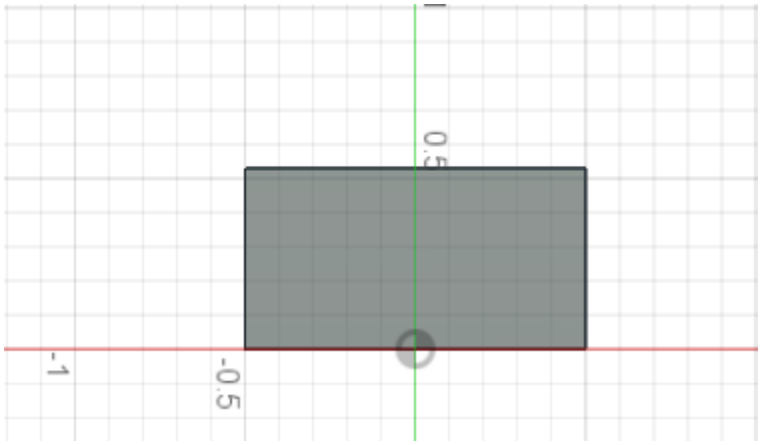
Leave it hidden for now.
Also notice that our sharpener is still the selected object because it is highlighted in gray and has the radio button “dot” filled in.
Next let's add the slant for our sharpener.
Click the Create Sketch button again.



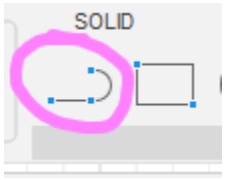
This time, we are asked where we would like to draw the sketch (ie on the grid or on one of the surfaces of the object.) Select the front surface of the cube like this:



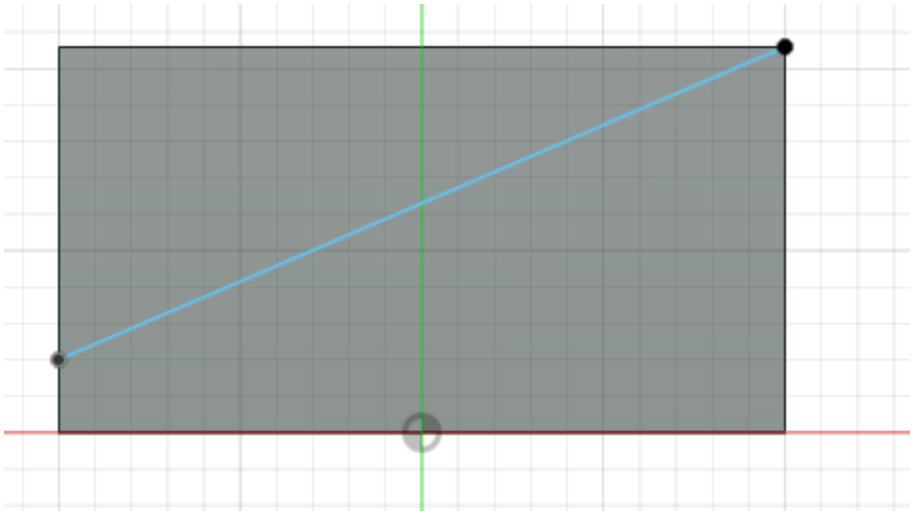
And now we are in Sketch Mode again and the camera changes to the side view of the cube. We are drawing our new sketch on the side of the cube!



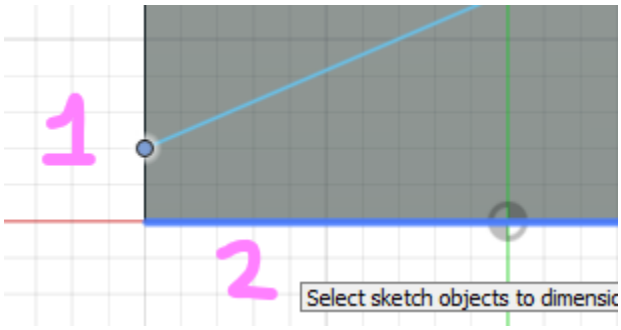
Lets add the slanted shape of the sharpener.
Click the Line tool



And draw a line diagonally from the left to the top right of the cube like this. Leave some room on the left side. The line should be blue.

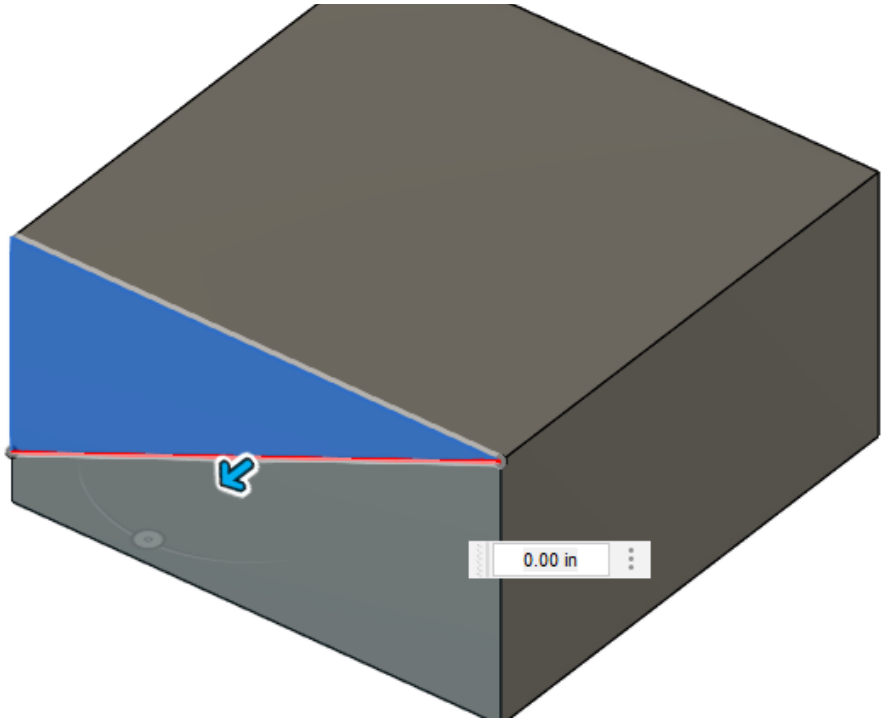


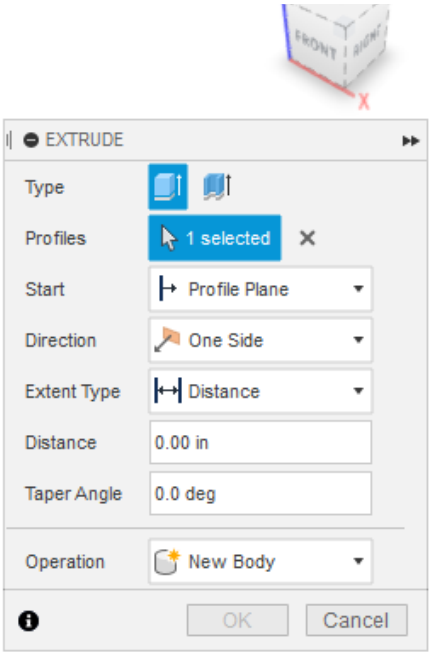
Next, lets dive the line a dimension from the bottom left so that we know the distance.
Press the “d” key to create a dimension. Click the left point of the line, followed by the bottom edges of the cube.



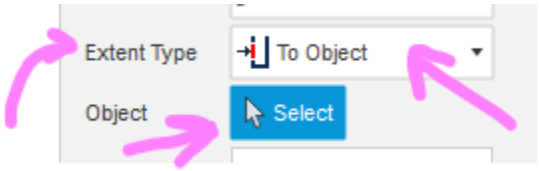
Drag out the dimension line and then put 0.1” into the input field.

Our line now turns black because there are no ambiguous dimensions about it. It is fully constrained. The left point will always be 0.1” above the bottom of the cube, no matter what size we decide the make the cube at a later stage.
Lets use this new diagonal line to extrude away the top of our sharpener.
Click “Finish Sketch” to return to exit sketch mode and return to design mode.
Next, press the “e” key to extrude.
Select the top portion of the sharpener. You will see this:

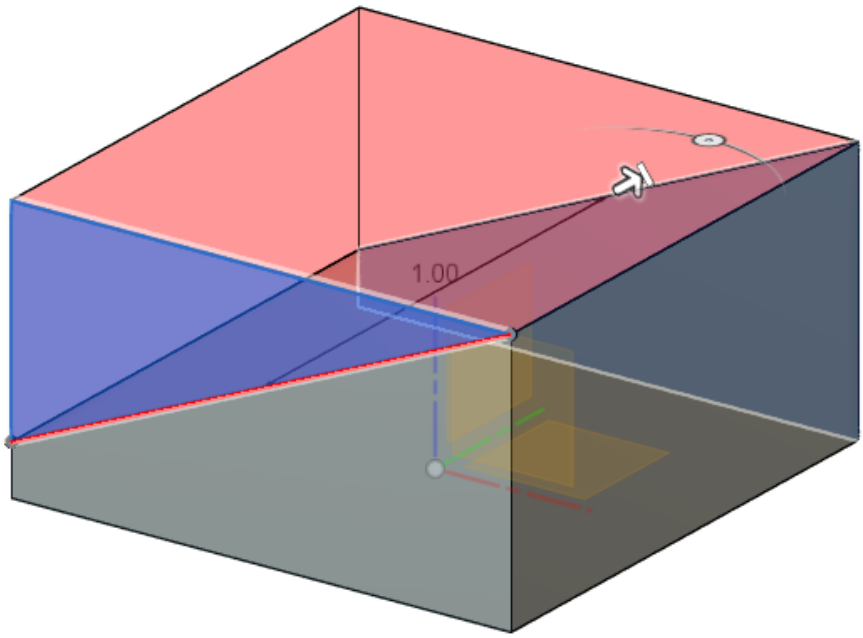


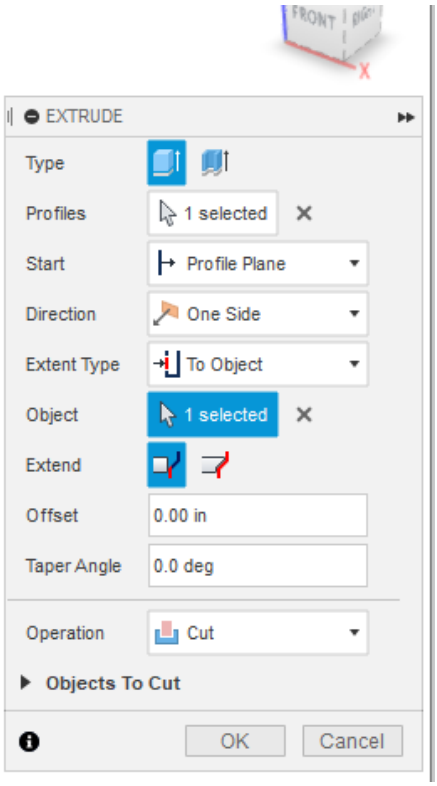


Notice the Extrude options on the right of the screen. Also notice the blue arrow that we can drag to get the extrusion working.
Change the dropdown menu option “Extent Type” to “To Object”

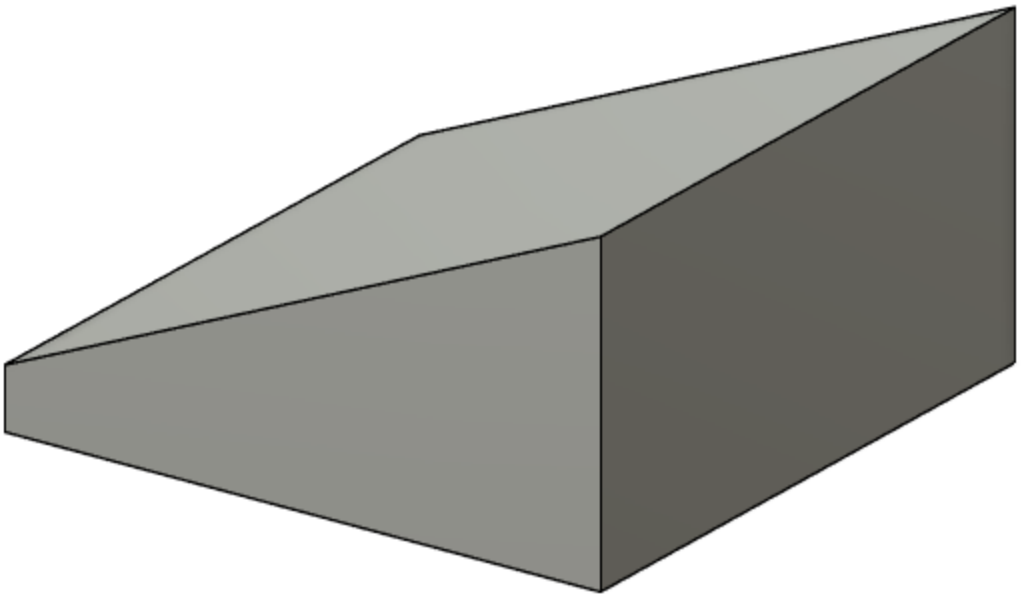


And select the back wall of the cube. We are telling the extrusion to travel all the way to the back face of the cube

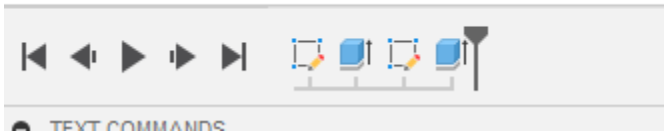




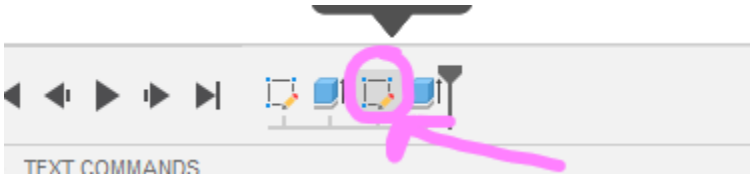
Notice that the extrusion is red because we are “cutting” away part of the geometry.
Press OK to complete the operation.
Our cube is now cut away



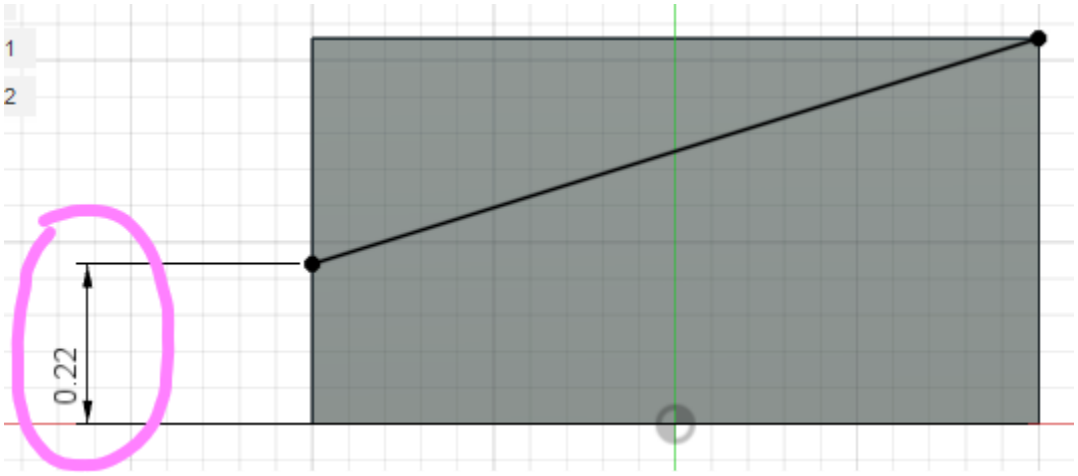
Also notice that our timeline below is collecting all of our actions so far



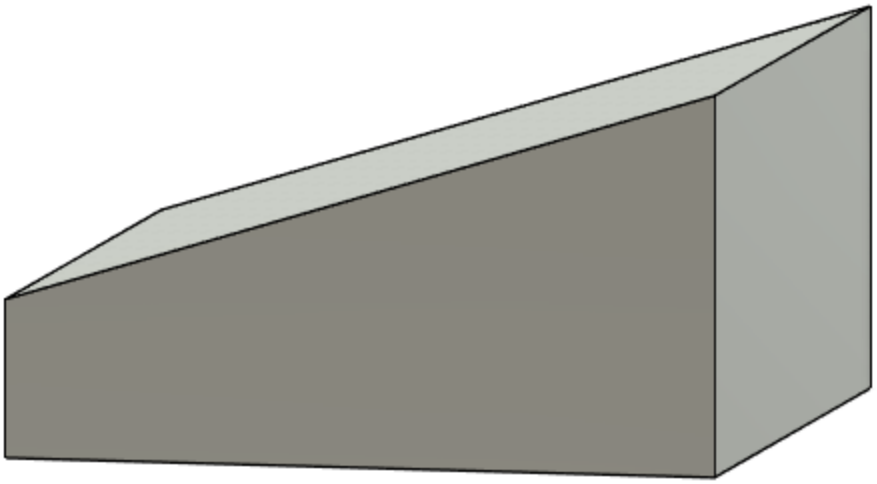
OOPS! The edge of the diagonal is too low. I meant to set it at 0.22” instead of 0.1”. Let's fix that!
Double click the second sketch icon in the timeline



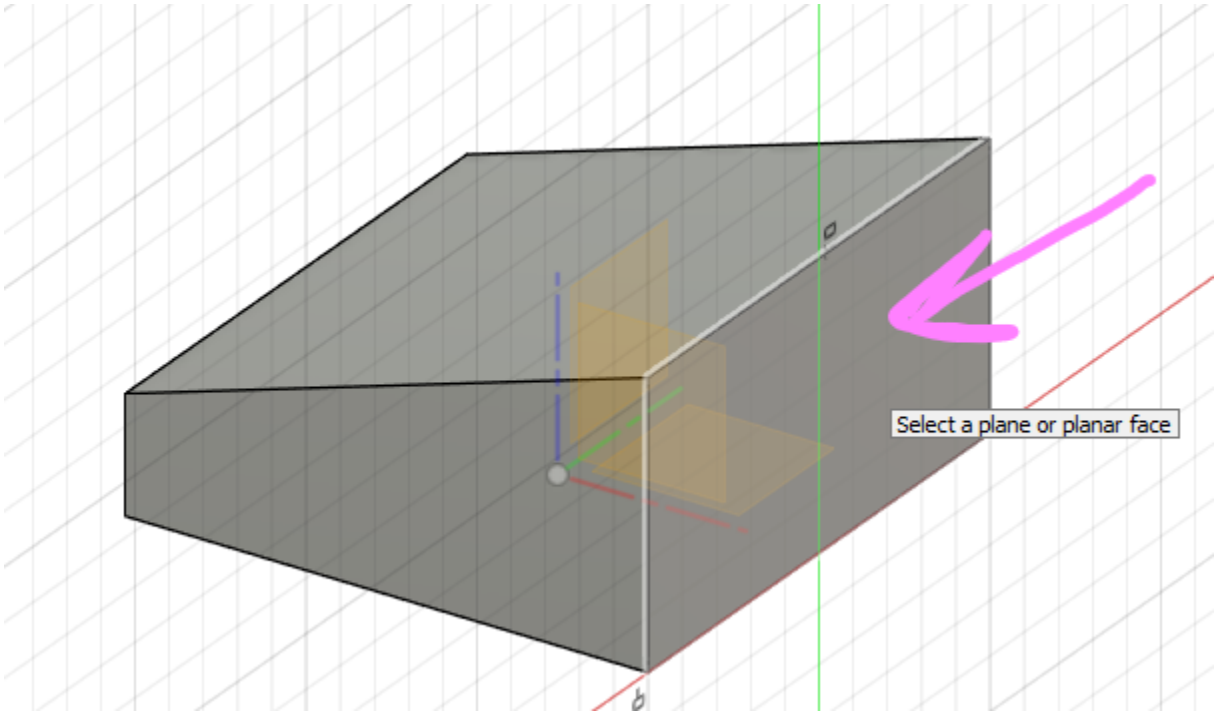
Notice that the view returns to the sketch mode and we can make changes there.
Change the dimension of the slant line from 0.1” to 0.22”



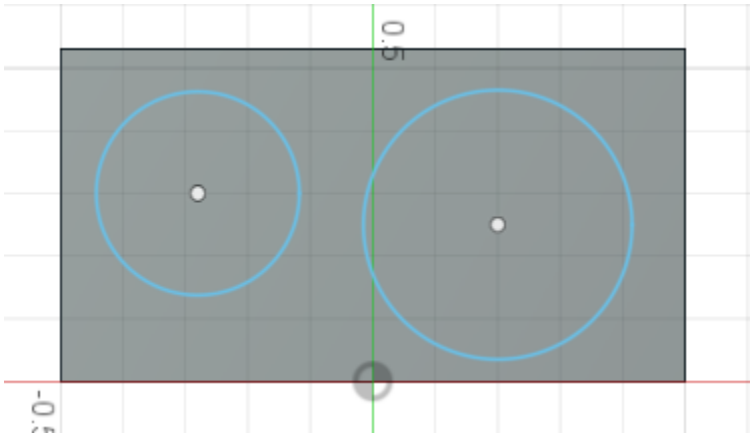
Then click the “Finish Sketch” button in the top right.
Notice that our changes propagate forward into the extrusion we later performed. This is the primary feature of Fusion360. We can change things from the past and their effects propagate into the current state of the model.’Our sharpener is now updated with the correct dimension.



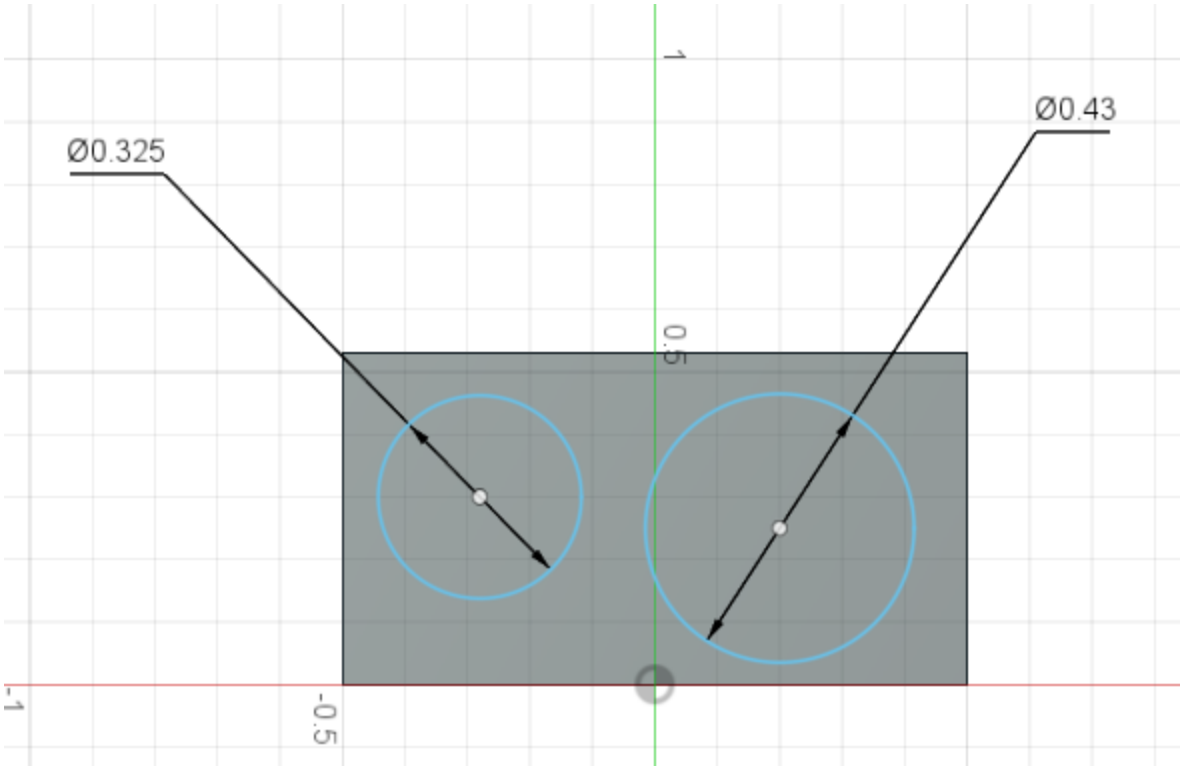
Let's add the holes for the sharpener.
Let's add a new sketch on the tall side of the sharpener
Click the “Create Sketch” button and select the tall side of the sharpener



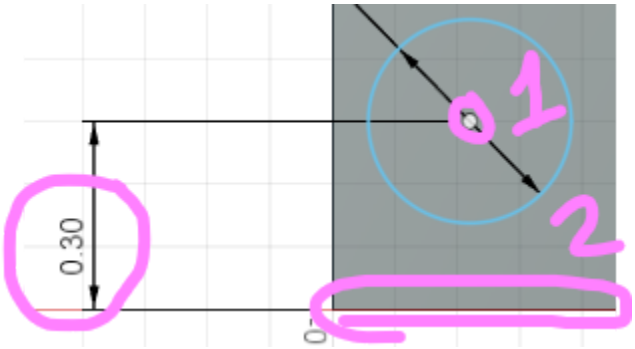
Click the circle button and draw two circles on the face of the cube.



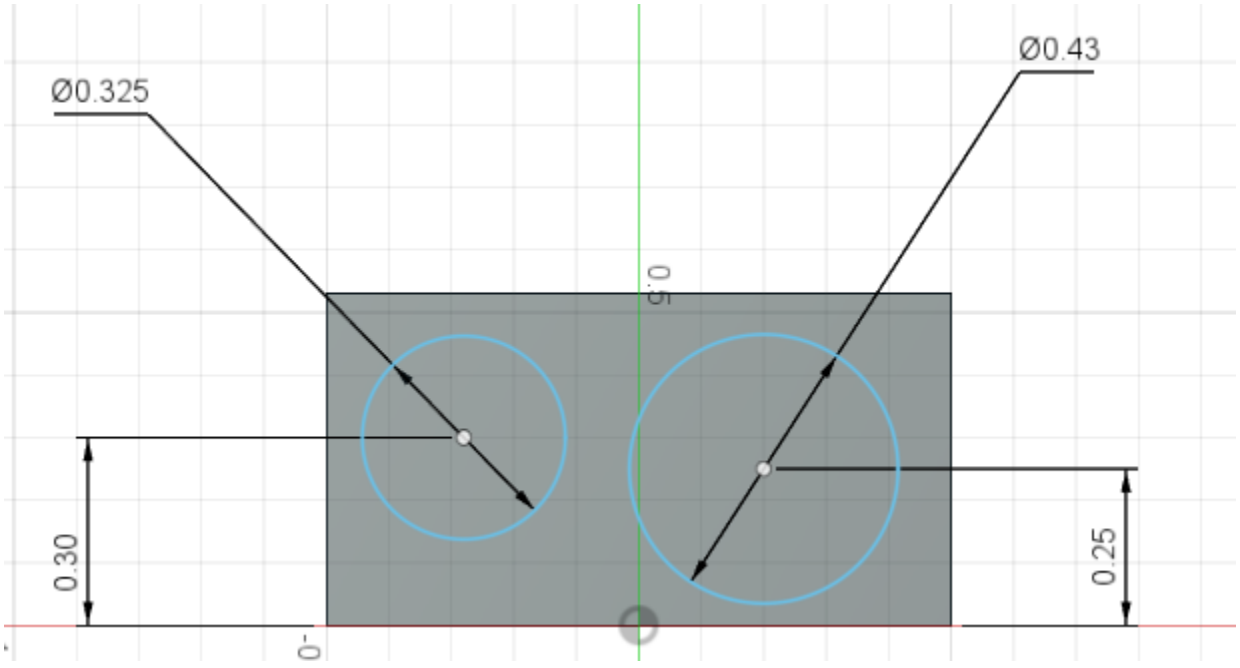
Select the dimension tool (“d” key) and give the left circle a dimension of 0.325”.
Give the right circle a dimension of 0.43”



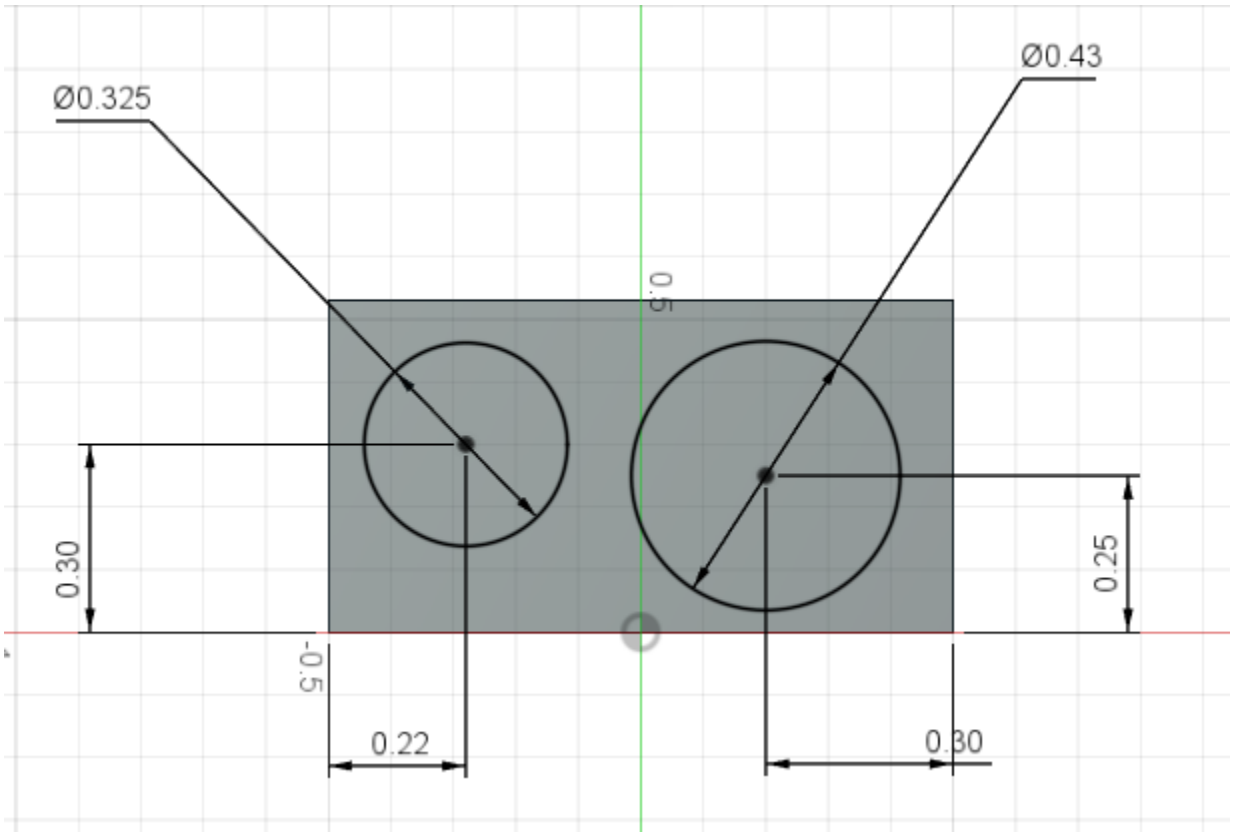
Lets define the dimension for the distance fo the circle tothe edges of the sharpener.
For the left circle, add a dimension from the center of the circle to the bottom edge of the sharpener with a value of 0.3”



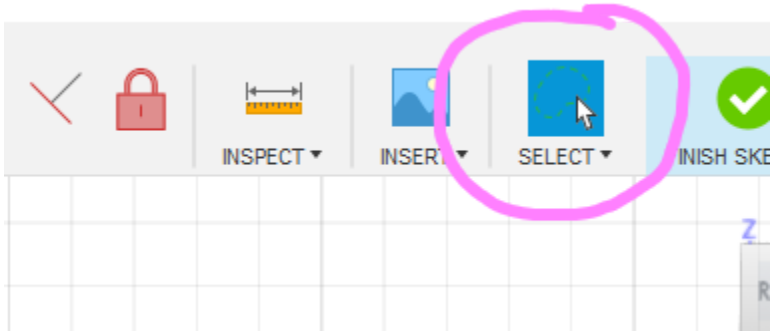
For the right circle, add a dimension from the center of the circle to the bottom edge of the sharpener with a value of 0.25”

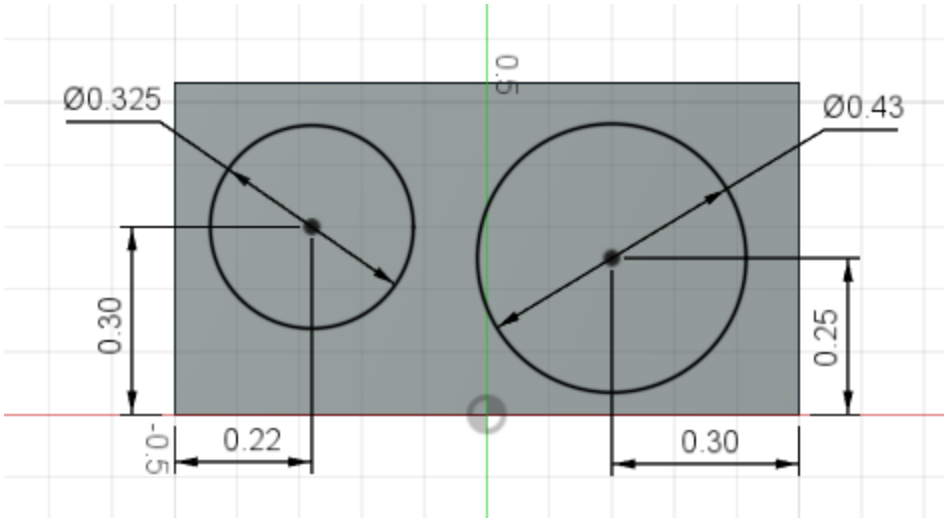


Notice that our circles are still blue, meaning that they are not fully constrained yet.
Lets add more dimensions
For the left circle, add a dimension from the center of the circle to the left edge of the sharpener with a value of 0.22”
For the right circle, add a dimension from the center of the circle to the rightedge of the sharpener with a value of 0.30”

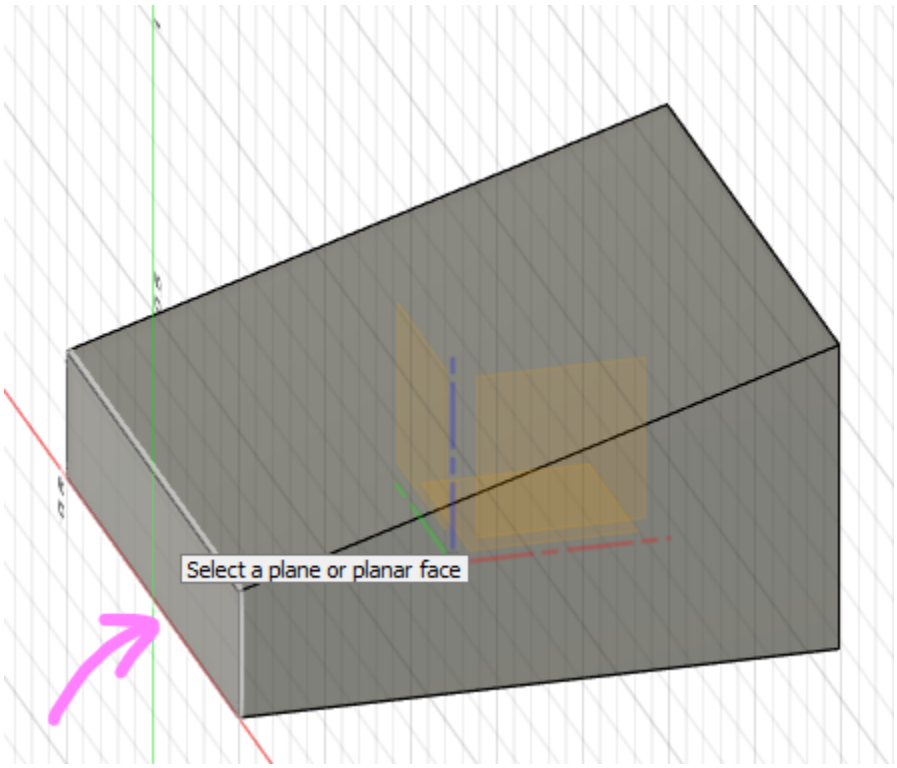


Now our circles are fully constrained (they turned black)
We can adjust the dimension lines to position them at better locations by clicking and dragging them with the “select” option enabled:

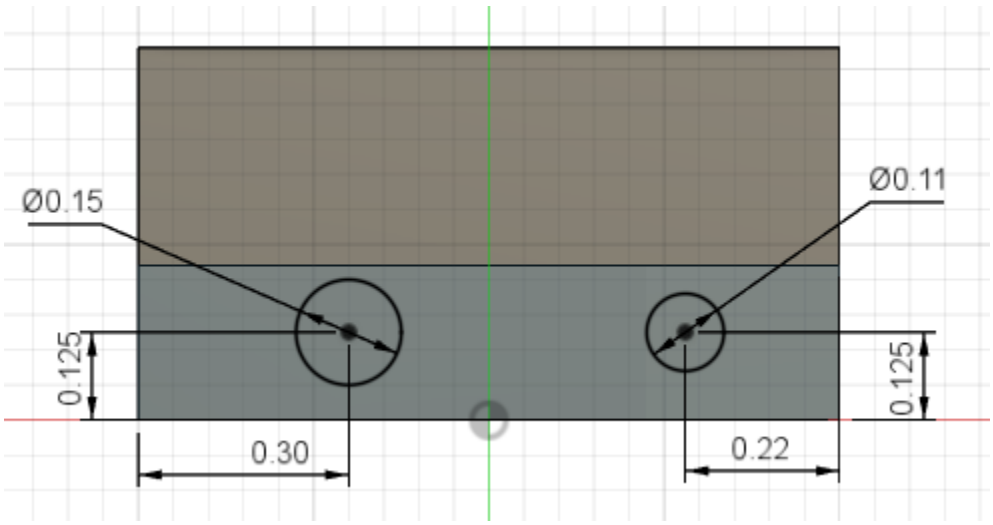




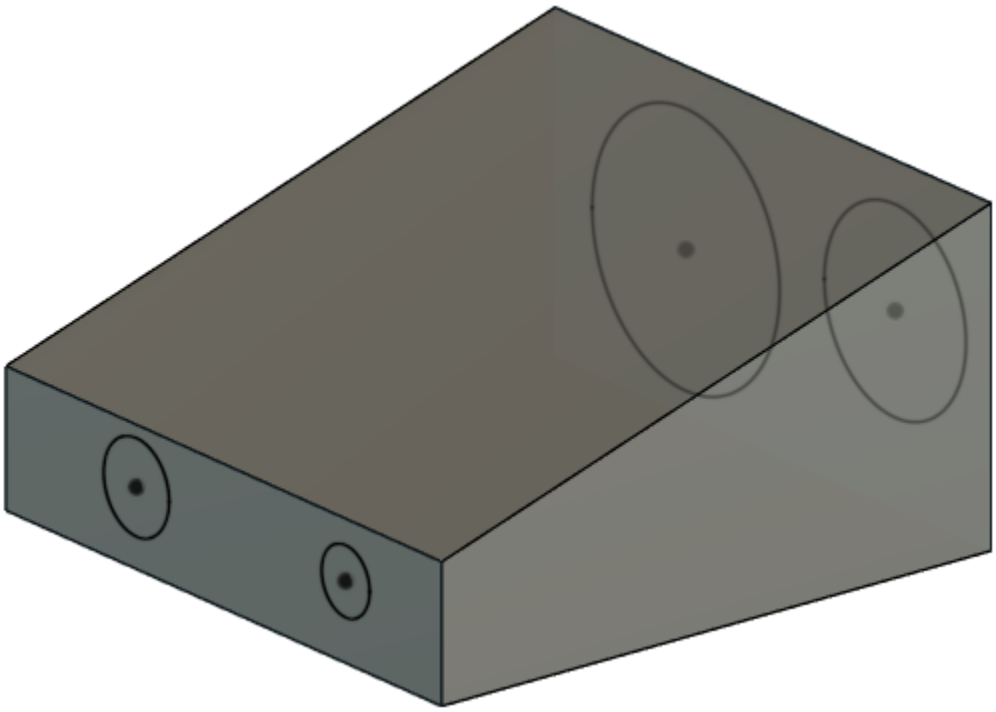
Click the “Finish Sketch” button.
We will now add another sketch to the front of the sharpener with more circles.
Click “Create Sketch” and select the front smaller face of the sharpener to apply the sketch there.



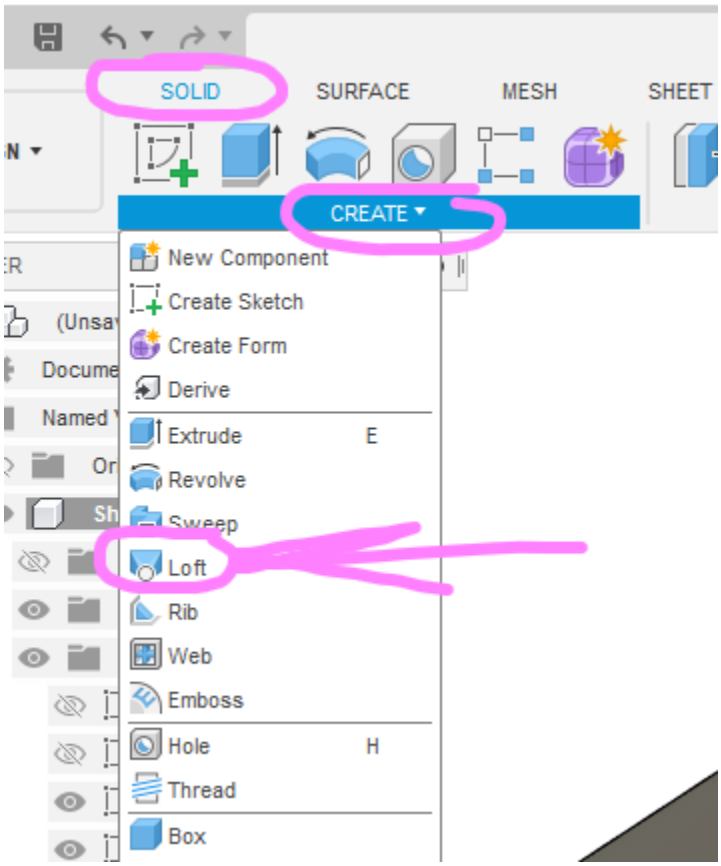
Repeat the process above, this time using smaller circles with the following dimension:



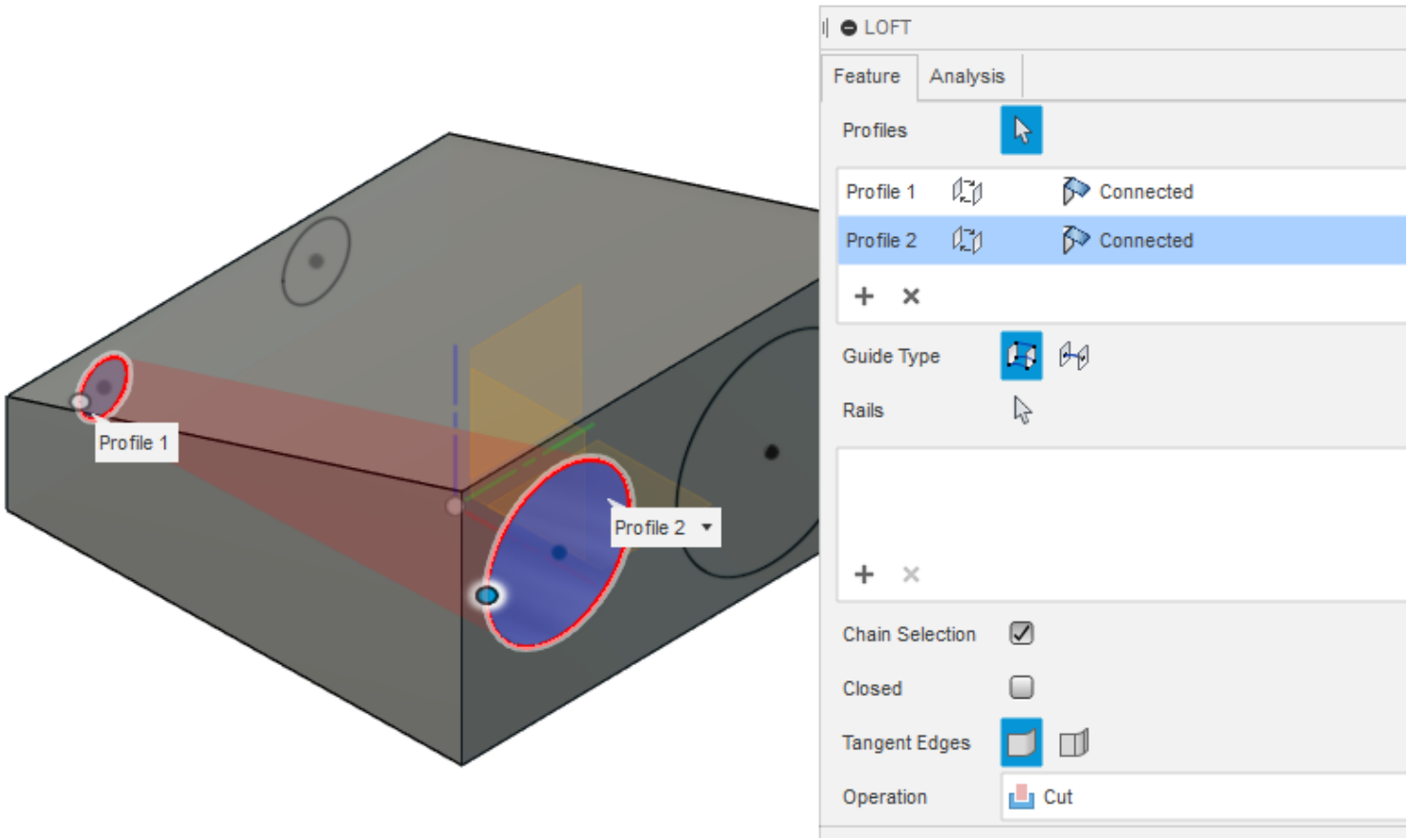
Remember to draw the circles first and then press the “d” key to give them sizes and to lock them into constrained positions until they match the image above and the circles are black (fully constrained)
Once complete, click “Finish Sketch” to return to the Design view which looks like this:



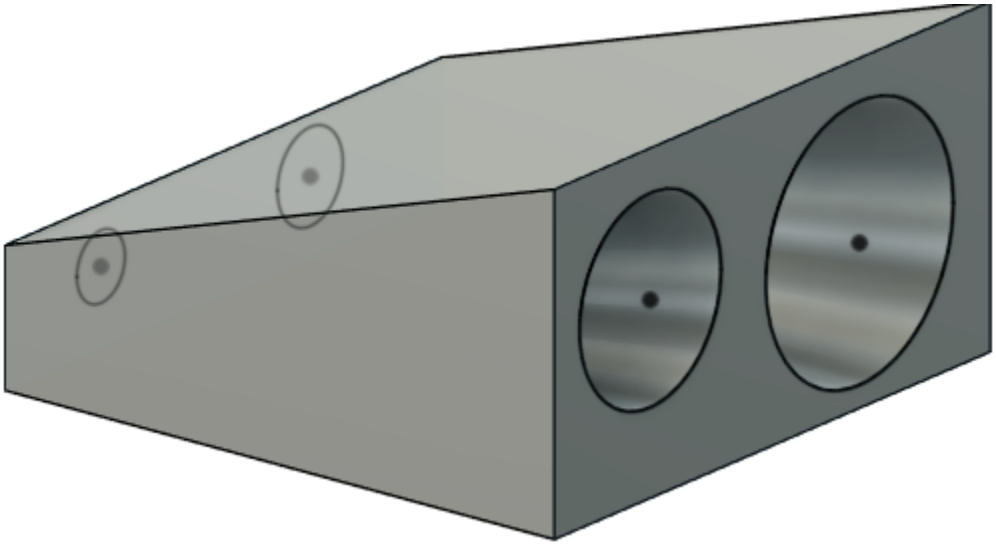
Now we will Loft the circles to cut out the sharpener holes.
Click the Loft command from the Create dropdown menu.



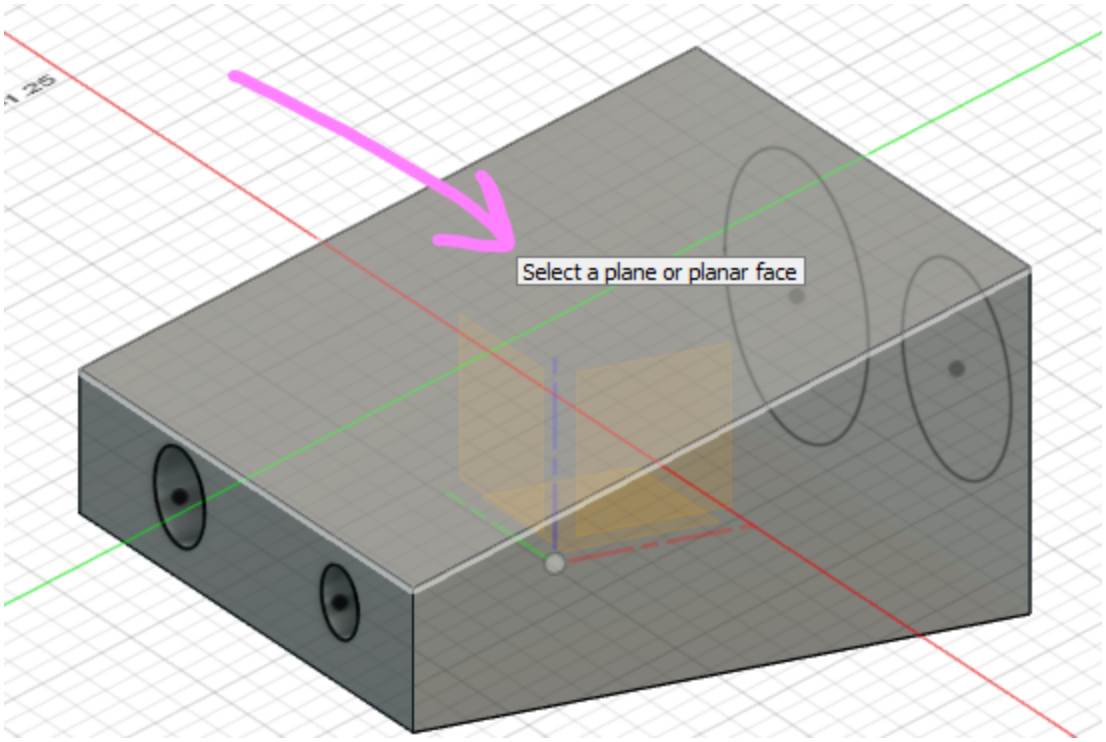
Within the Loft command, the Loft menu options appears on the right of the screen. This is normal. Select the first Small circle, and then select the first large circle.



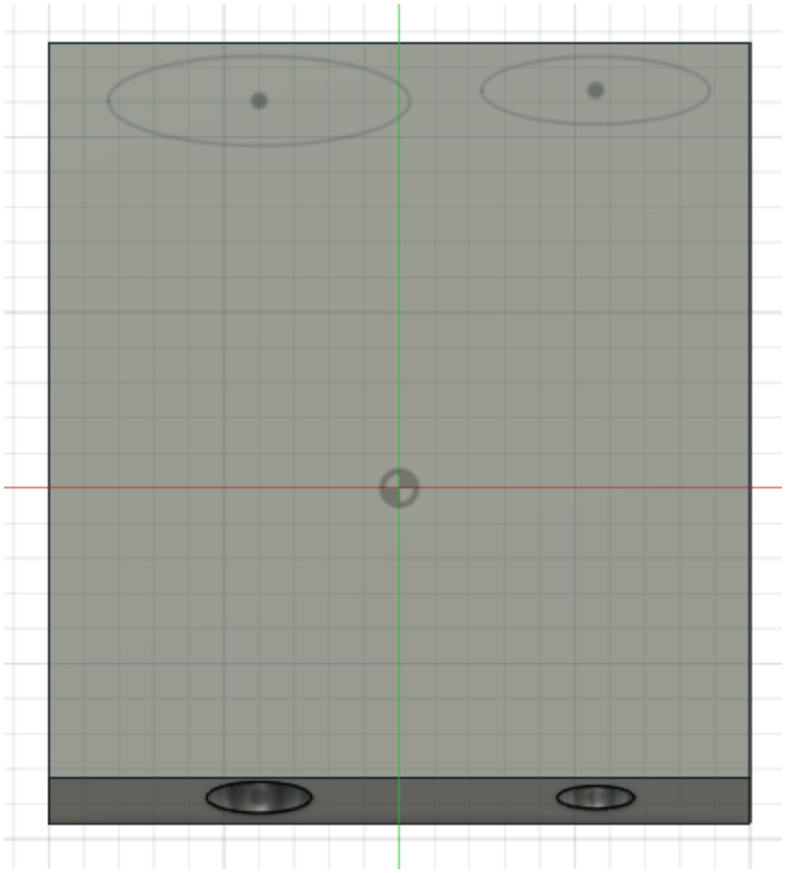
The Loft Menu options will reflect our selection, and the preview will show the red area that will be cut away.
Click OK to perform the loft.
Next repeat the process for the other two circles.



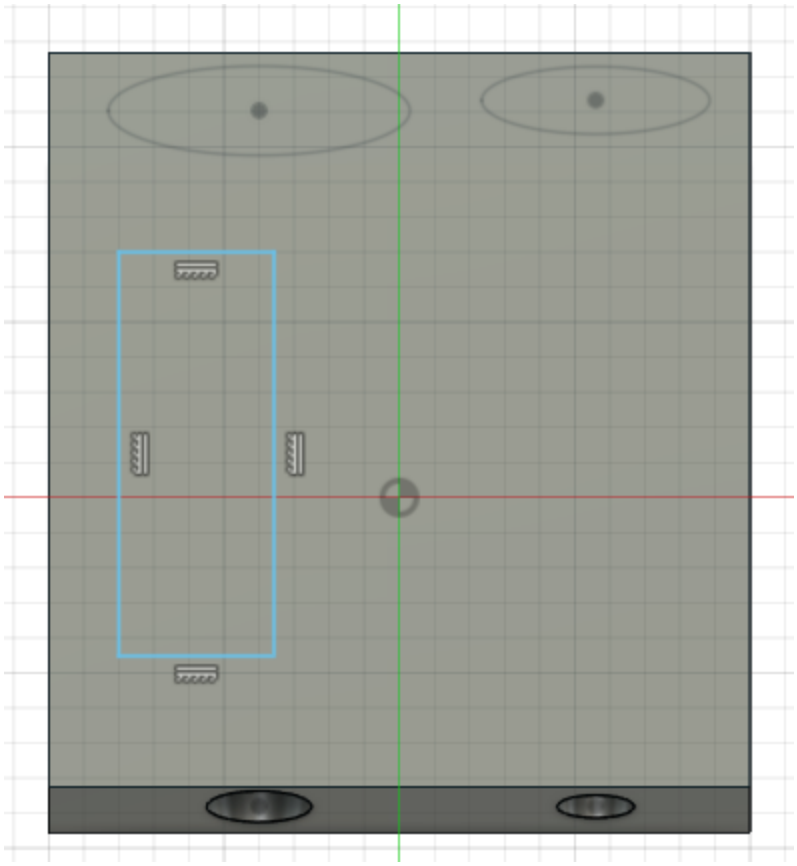
The holes are now lofted and cut away!
Next lets create the channel for the sharpener blade
Create a new sketch and apply it to the top slanted surface of the sharpener



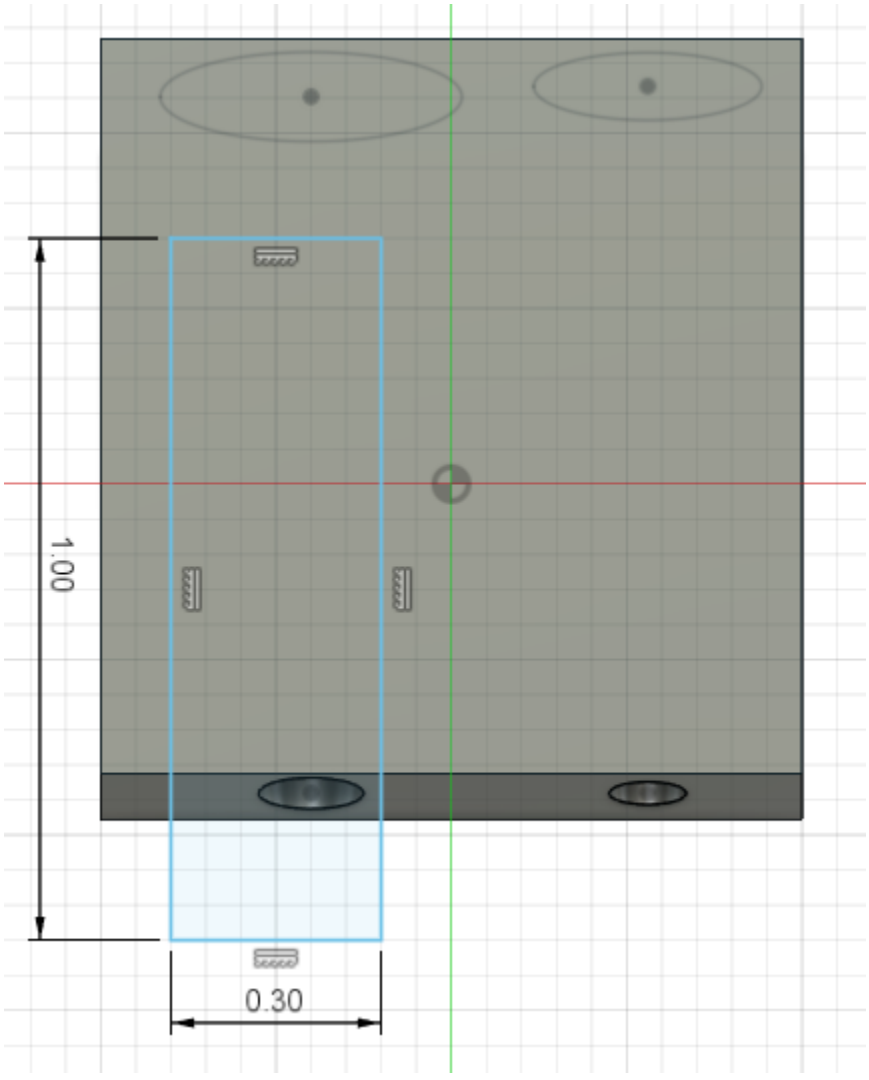
Our view changes to align with the slanted edge while in sketch mode



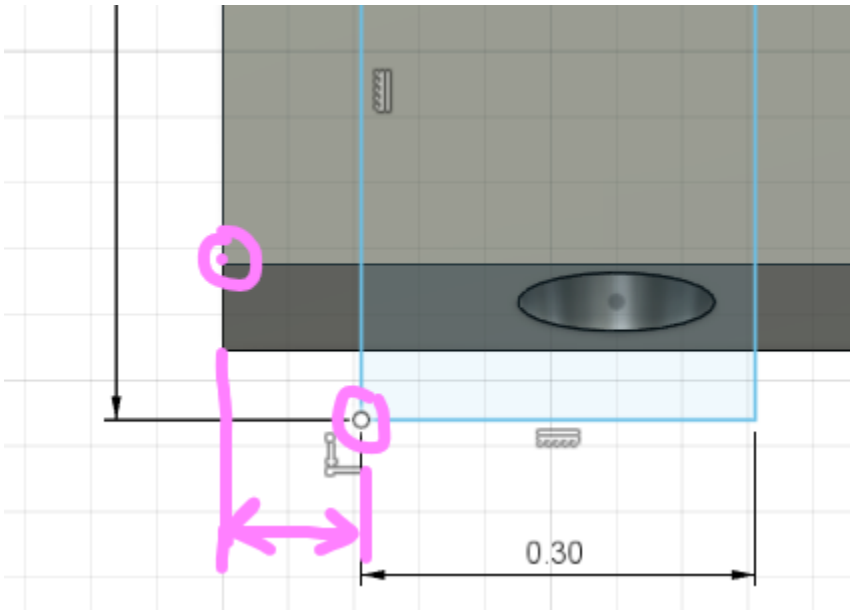
Draw a rectangle like this



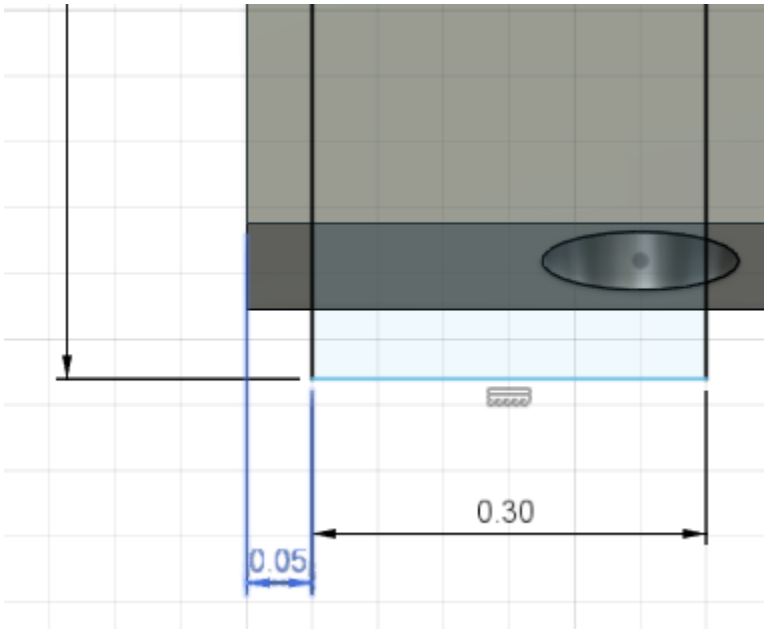
Give the rectangle a dimension of 0.3” width and a height of 1.0”



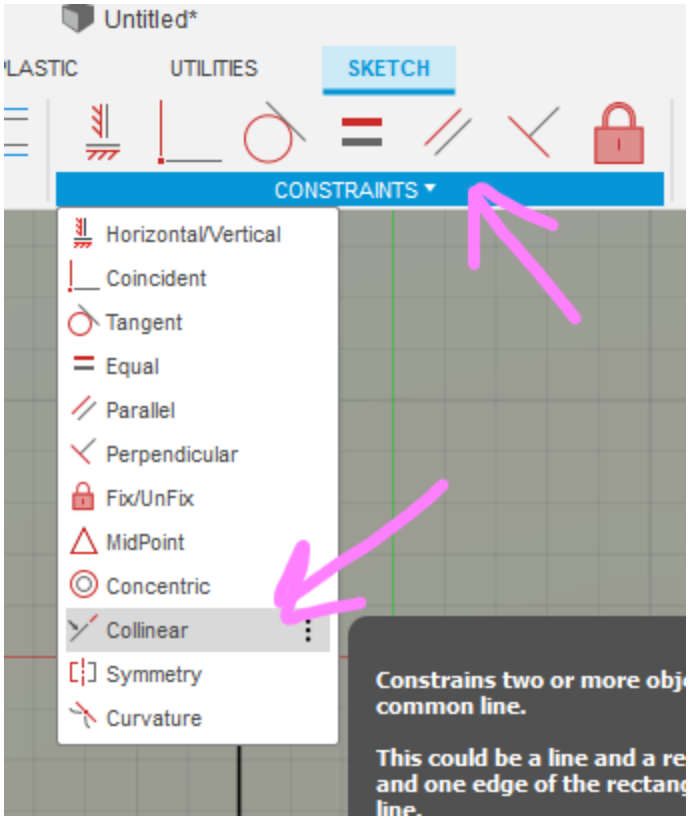
Next lets define the distance from the bottom left point of the rectangle to the bottom left point ofthe sharpener surface like this



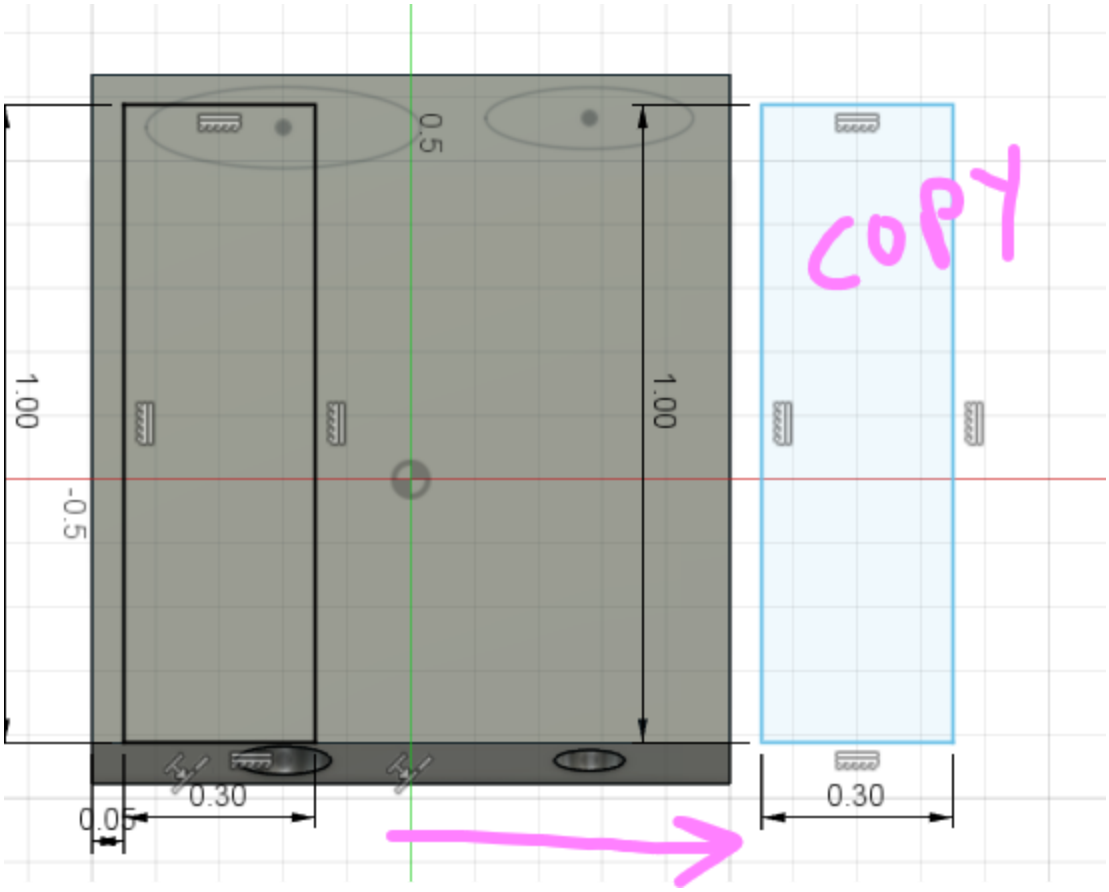
Give it a value of 0.05”



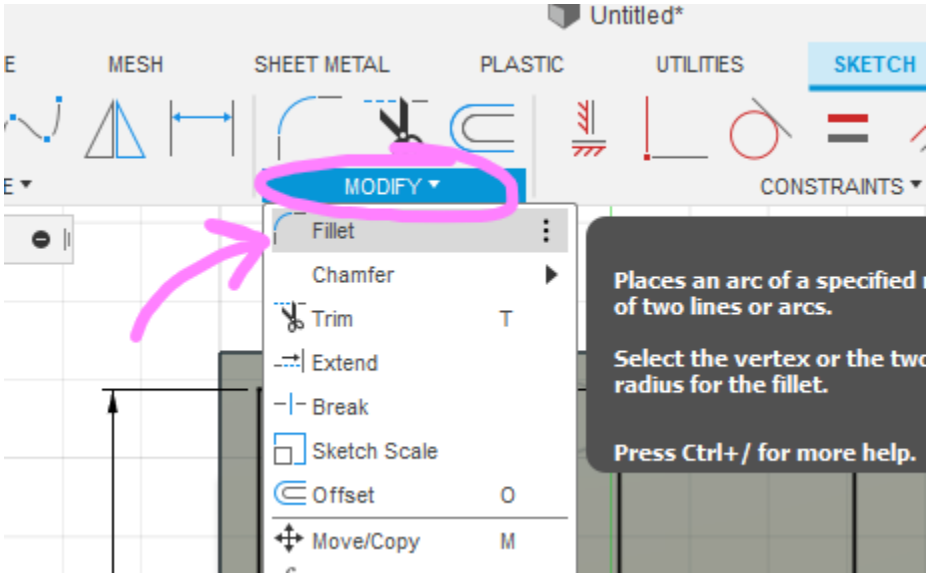
And lets give the lower edges of the rectangle a collinear constraint with the lower edges of the sharpener



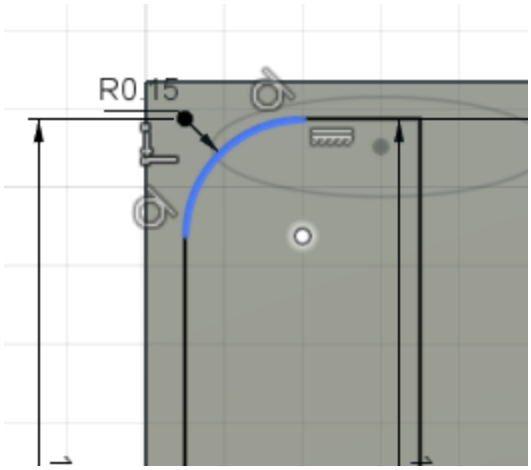
The edges of the rectangle snaps to the edge of the sharpener and it is now fully constrained (all black)
Double click the edges of the newly constrained rectangle and copy and paste it using the shortcut keys ctrl+c and ctrl+v.
Click the arrows to drag the new copy over to the right side



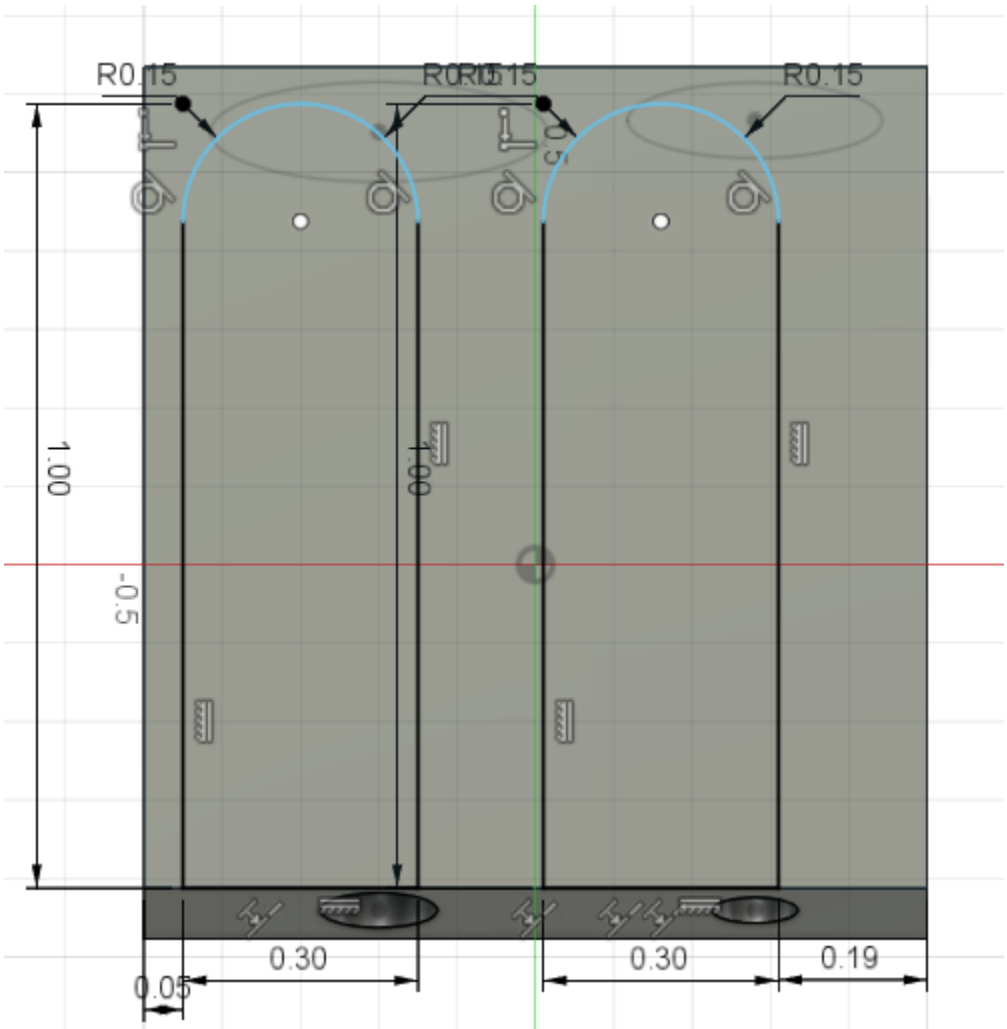
We can now constrain this new copy
The bottom right corner of the new rect should be 0.19” away from the bottom right corner of the sharpener body
The bottom edge of the new rectangle should have a collinear constraint with the bottom edge of the sharpener body
Next, Lets round the edges of the tops of the two rectangles using the fillet command



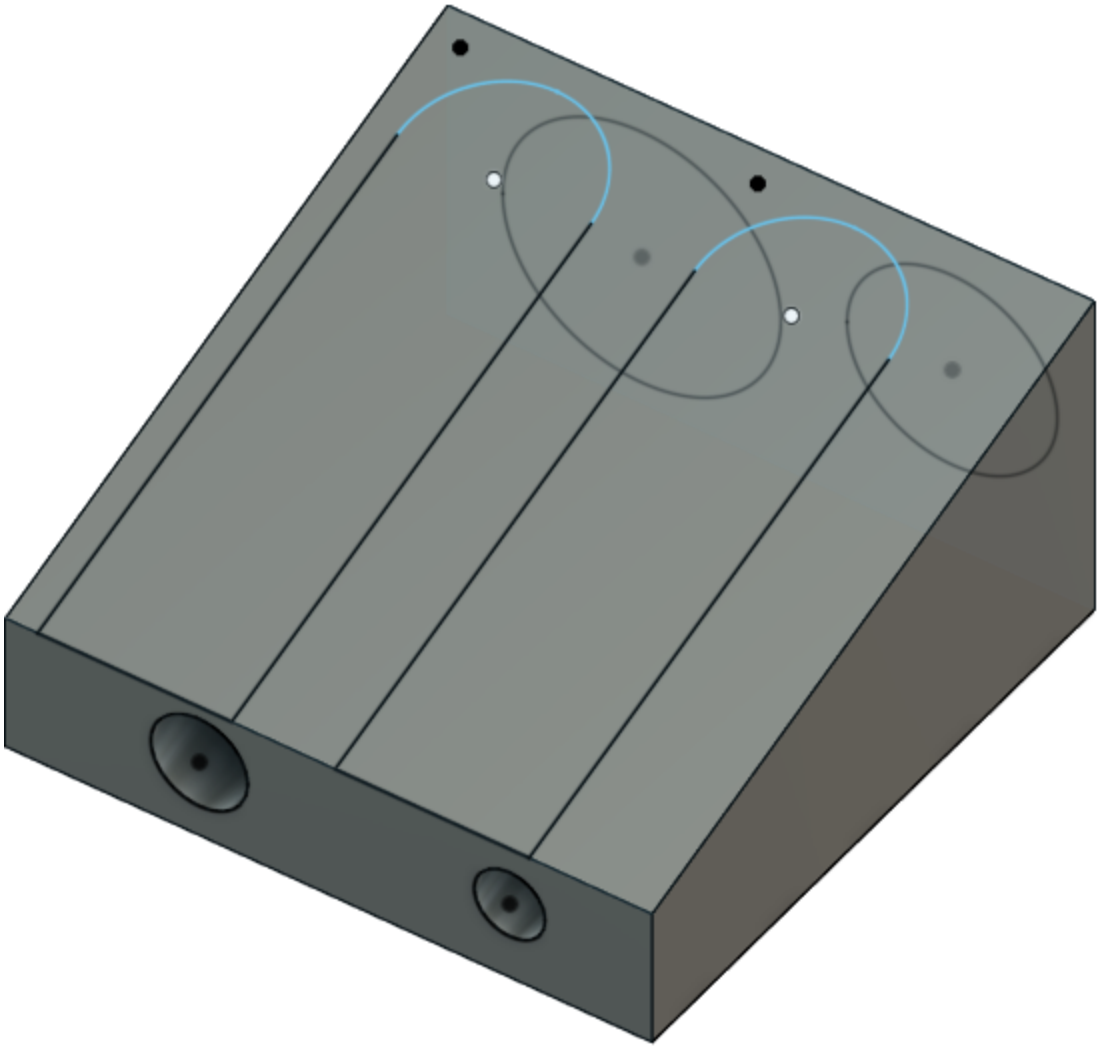
Click the fillet command and select the top and left edge of the rectangle
Set the fillet amount oto 0.15”



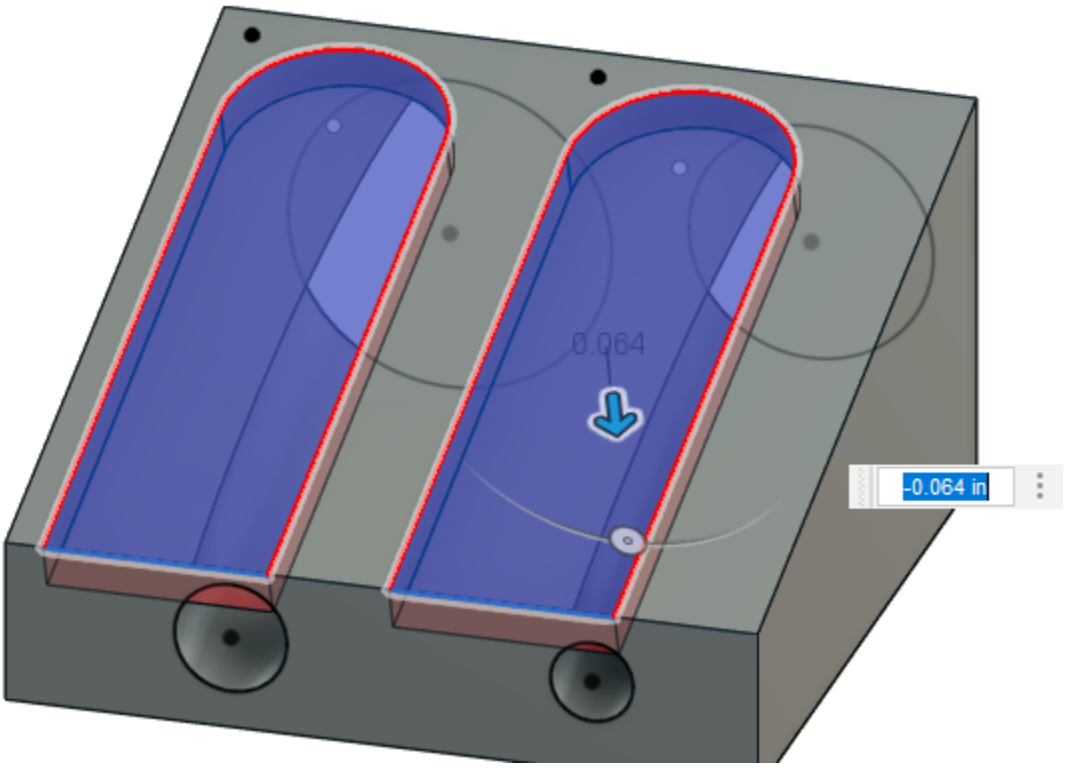
You may see a warning about losing a constraint in the process. That is okay to ignore at the moment.
Repeat the process on the other top edges of the rectangles so that it look like this:



Click “Finish the sketch” (notice the rounded edges are not black so they are not fully constrained. That is okay anw we can continue with them that way without too much of an issue)



We will now extrude these two shapes to cut the holes for the blades of our sharpener.
Press the “E” key and extrude the two shapes by the amount of -0.064”



Press OK and complete the extrusion Cutting away the recesses for the sharpener blades.