



The Center For Advanced
Surface Engineering



Introduction to 3D Printing

Activity 1:
Design a keychain using
computer-aided design software

In this activity we'll design a keychain name tag and learn the fundamentals of computer-aided design, the first step in creating your own 3D printed structures.



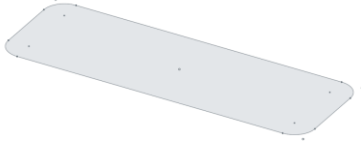


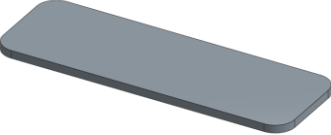

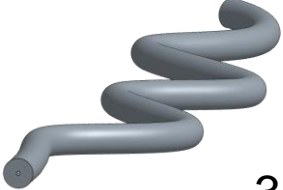
Your 3D CAD design



3D printed prototype

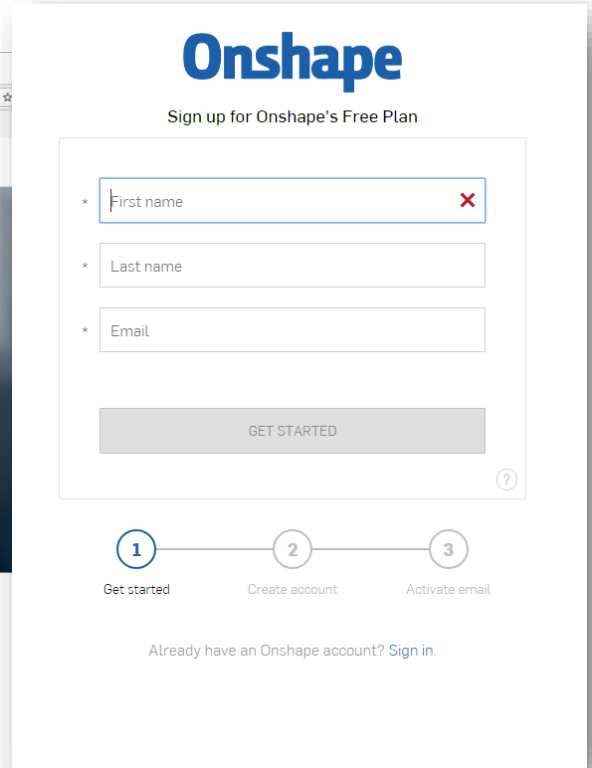
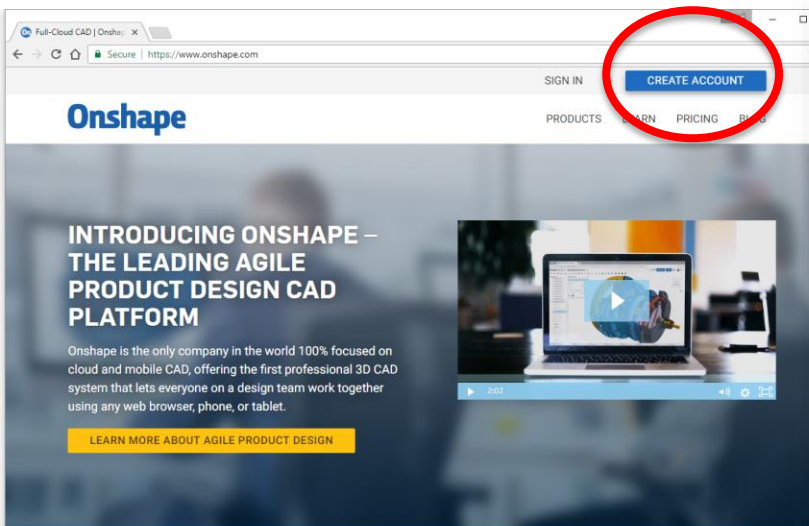
3D computer aided design (CAD) fundamentals

3D CAD designs begin with a closed 2D shape that defines a surface, and then another action to define a 3D volume. There are many ways to describe a volume, here are the three most common:

	Extrude	Revolve	Sweep
2D Surface			
3D Volume			


Let's start by creating our own 3D CAD keychain tag.

Create a free account at www.onshape.com.

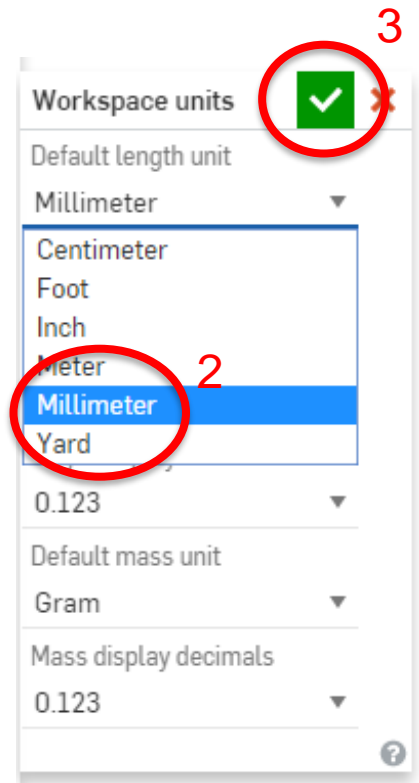
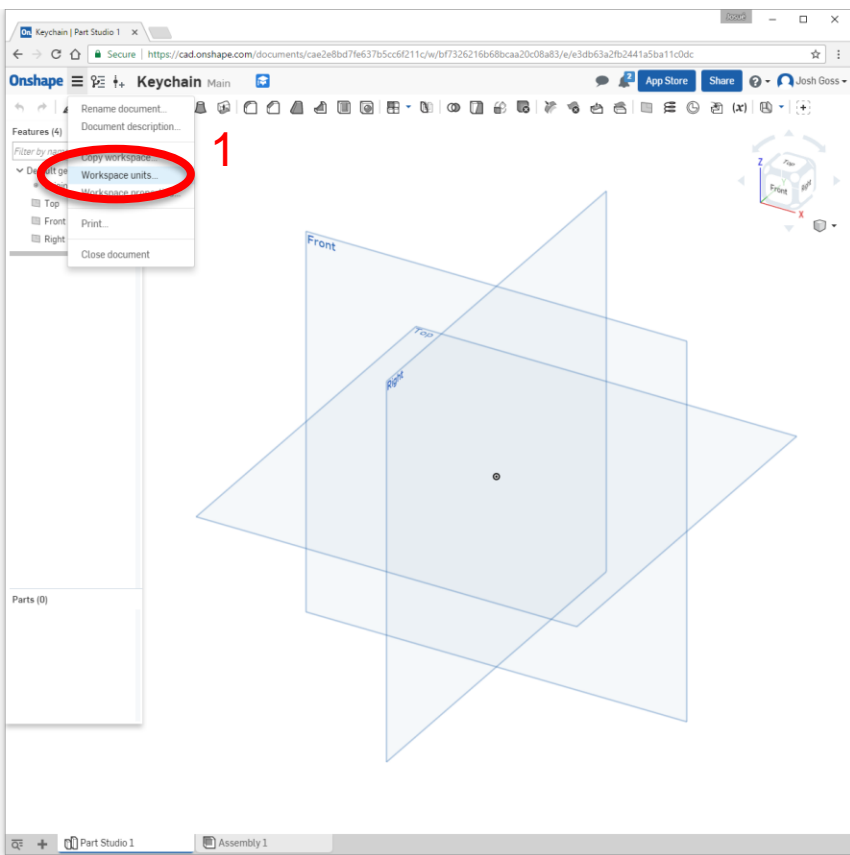


Press the “Create” button to start a new document.
Change the document name to Keychain.

The image shows a screenshot of the Onshape web interface. In the top-left corner, a red circle highlights the "Create" button, with a red number "1" next to it. The main content area shows a workspace titled "CASE Mech STEM" with a message "No documents for CASE Mech S...". A "New document" dialog box is open in the foreground, with a red circle highlighting the "Document name" input field containing the text "Keychain", and a red number "2" next to it. The dialog also shows a "Public" button and "OK" and "Cancel" buttons at the bottom right.

Click on the menu  button, then select “workspace units”.

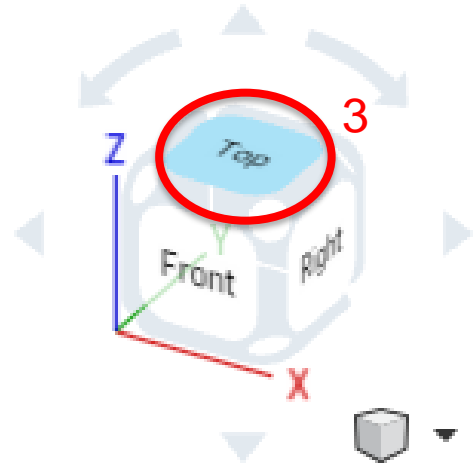
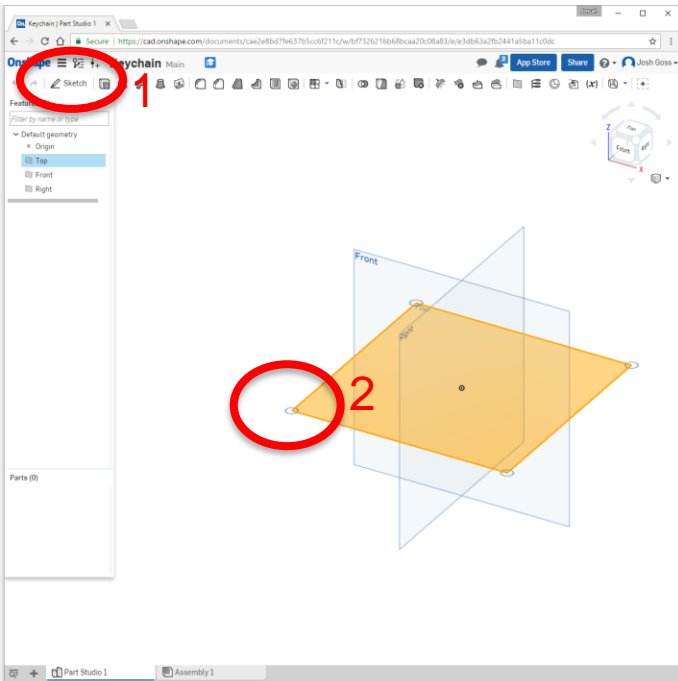
Set the default length units to millimeters and press  to confirm.



Click on  Sketch to create a new 2D sketch.

When asked to pick a plane to sketch on, click on the Top plane.

In the upper right corner, click on the word “top” on the 3D cube to see a top down view of your workspace.

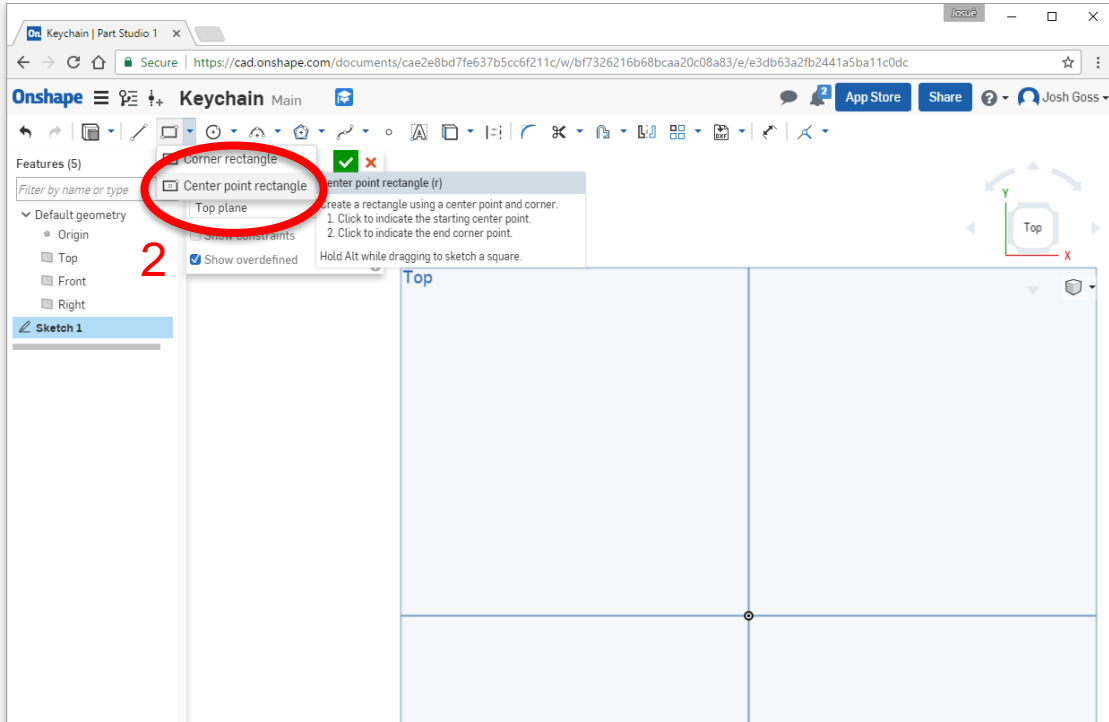
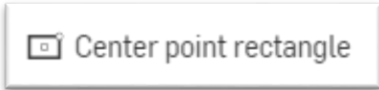


Begin sketching a center point rectangle.

Click on the small down arrow to the right of the rectangle icon



and select

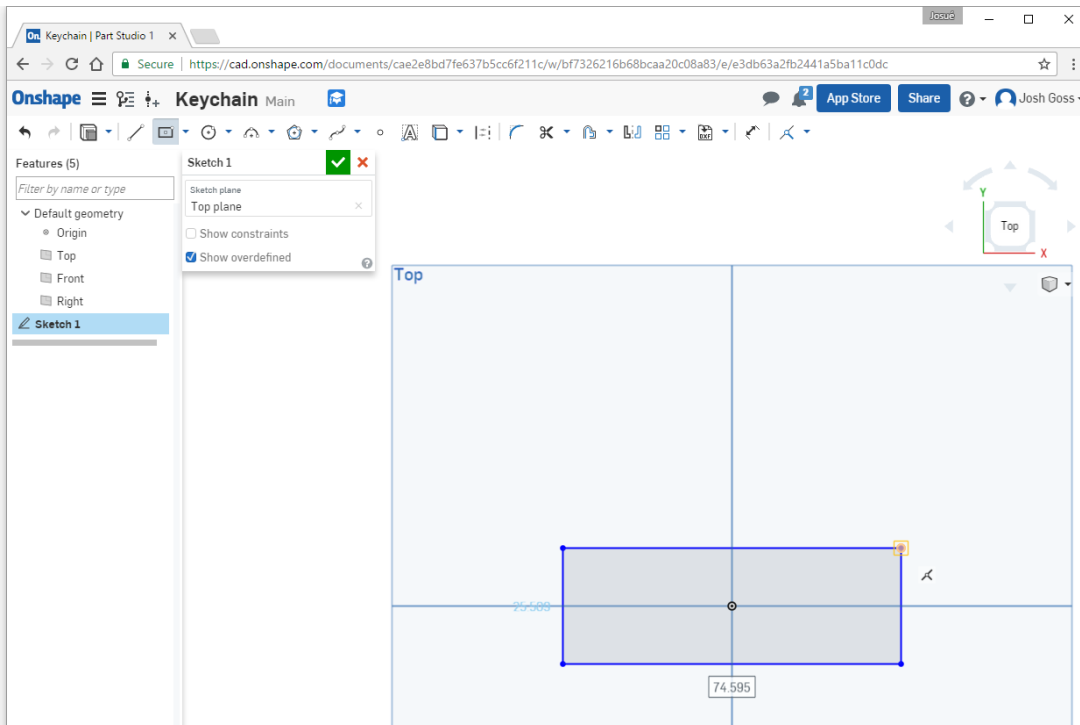


Center point rectangles require two clicks to place a rectangle.

Make your first click in the center dot



and your second click anywhere in the 4 quadrants to make a rectangle that is about 75 mm long and 25 mm wide.



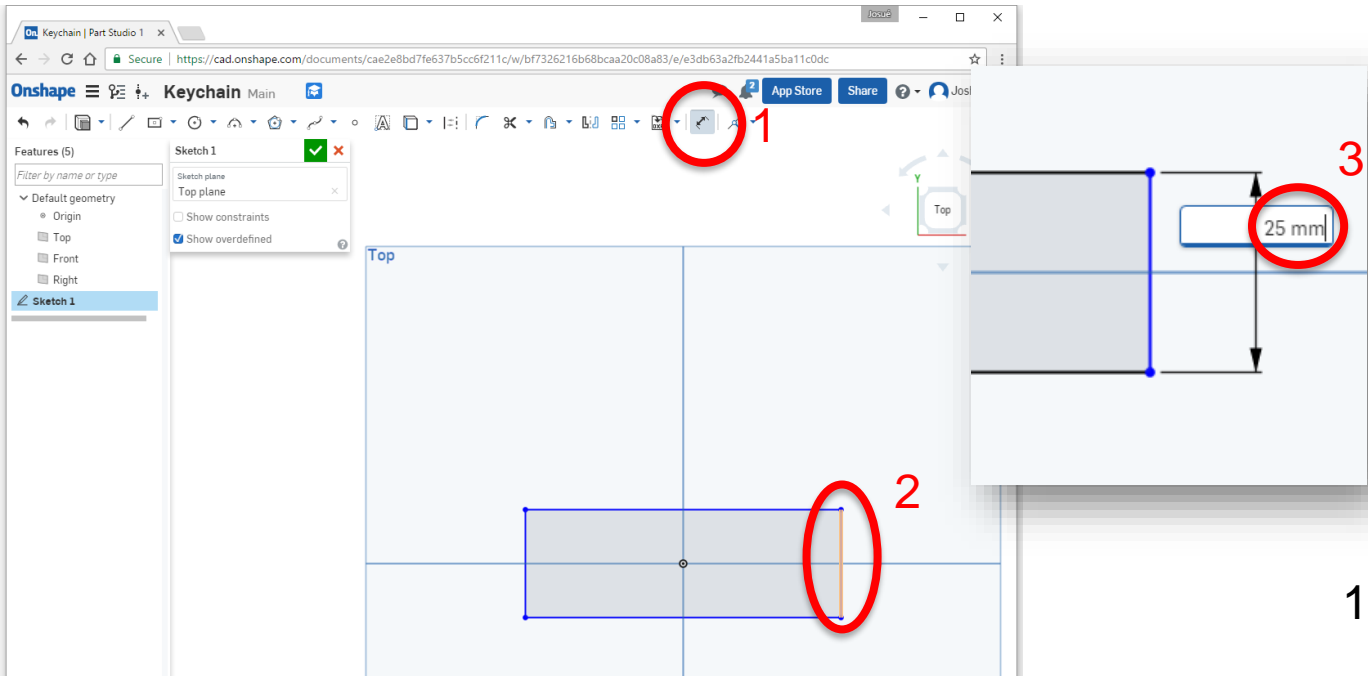
Click on the dimension button



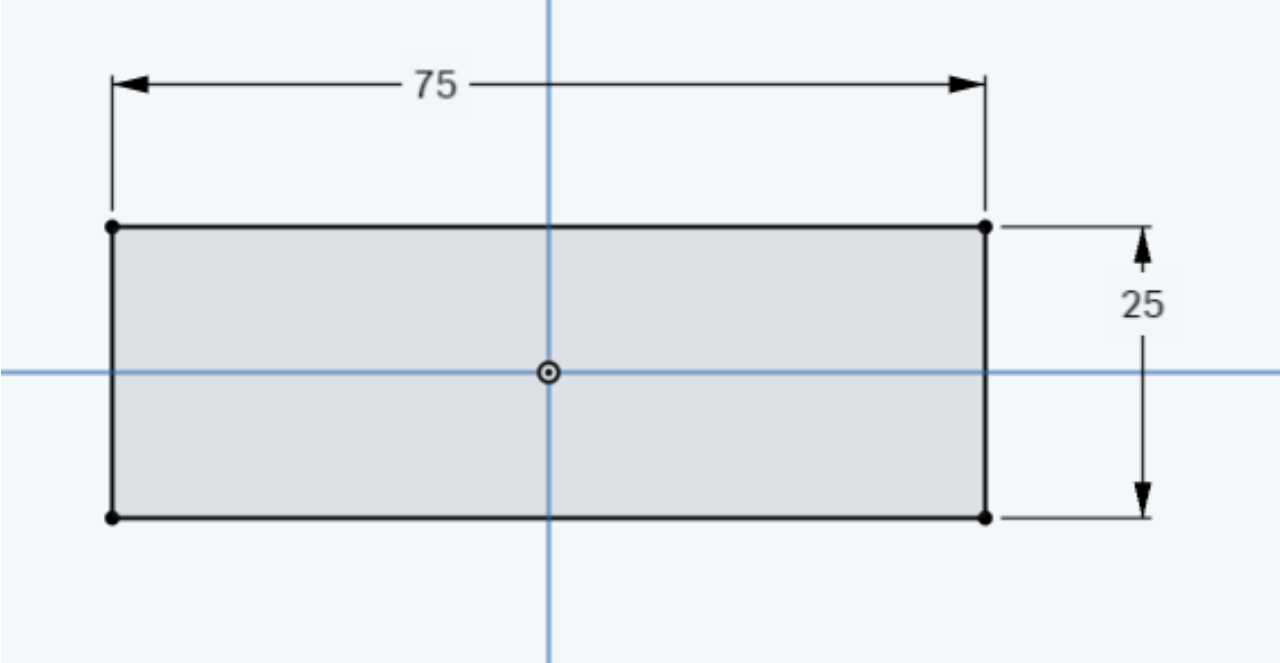
then click on the right blue line of your rectangle.

Click again to the right of the line to place the dimension.

Type in 25 to change the width to 25 mm. Press enter when done.



Repeat for length, make it 75 mm long. The rectangle changes from blue to black to show that all features are fully defined.

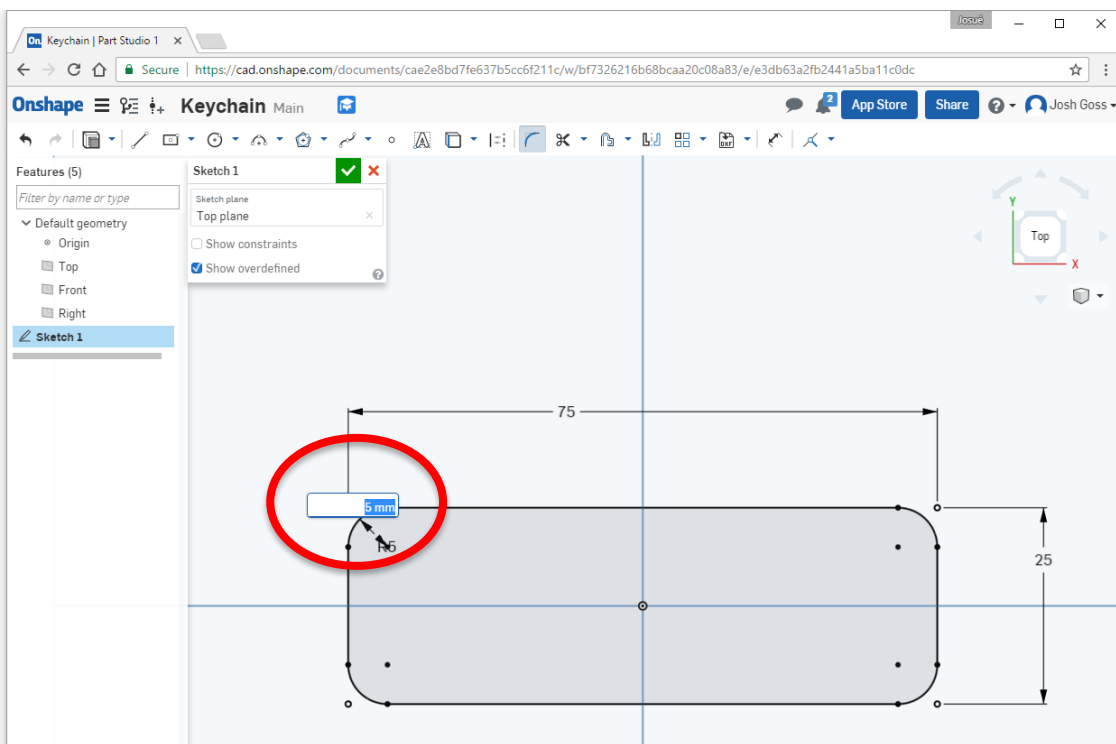


Click on the sketch fillet tool



(pronounced “fill-it”)

and then click on all 4 corners. The default radius of 5 mm is ok or you can customize the radius of your keychain corners anywhere between 2 and 10 mm. Press enter when done.



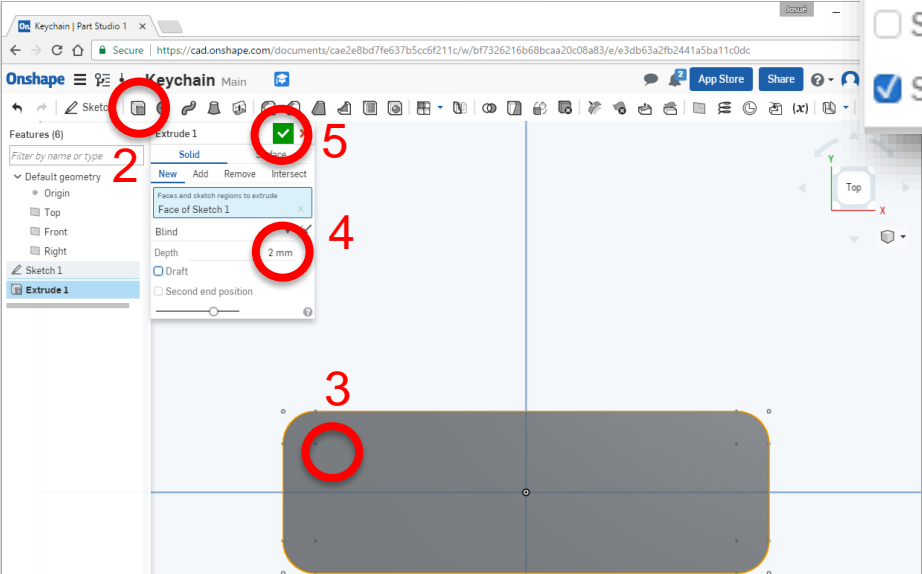
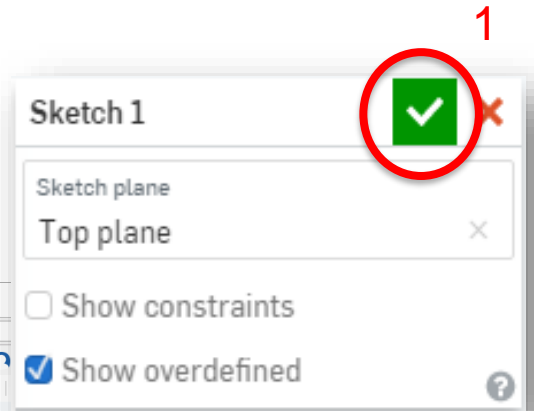
Now it's time to go 3D! Click on the Sketch 1  to close the sketch.

Click on the Extrude  button.

Click anywhere in the rectangle's top face.

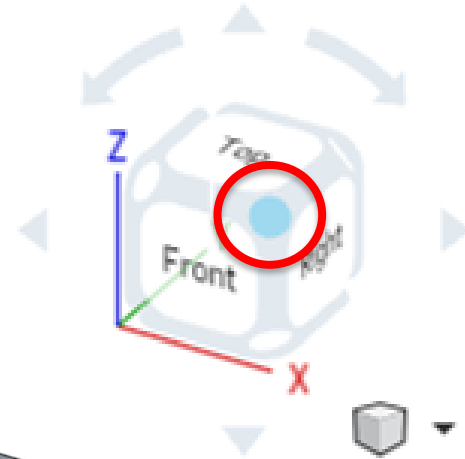
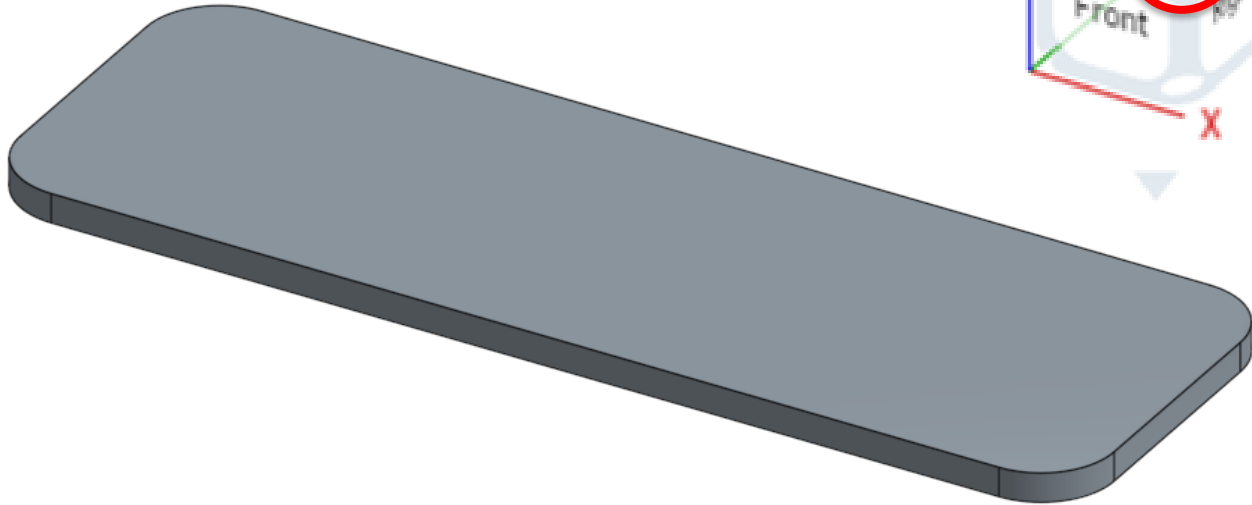
Type in a depth of 2 mm to extrude.

Click on the Extrude 1  to confirm.

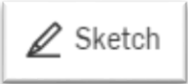



You now have a 3D part. To look around, use the cube in the top right corner to navigate around. Clicking on the corners of the cube gives you an **isometric** 3D view.


You can go back to the top view when ready.

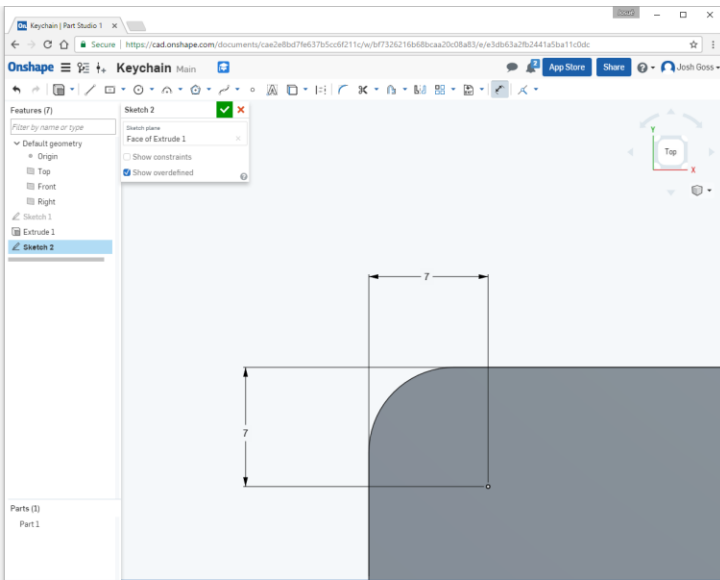


Let's make a new sketch for the hole. Holes have a special 3D button that puts a hole anywhere you have a sketch point.

Click on  and then select the top face of your part.

Click on the point button  and place it in the top left of the rectangle, then dimension it 7mm from top and left slides.

Click on  to close Sketch 2.



For these dimensions, you will have to click three times:

1. The point
2. The left line (not the radiuses)
3. Anywhere above the rectangle to place the dimension

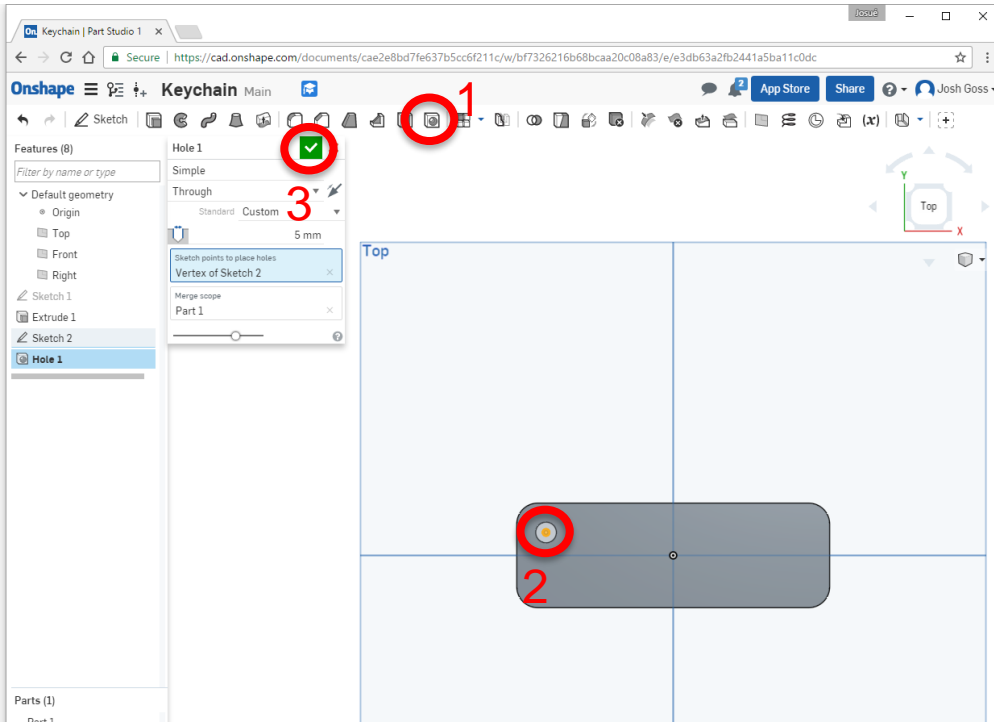
Key in 7 mm and press enter to move the dot precisely where you want.

Repeat for the top dimension.


Click on the Hole button  then select the point.

The default values should be to create a through hole (as in it goes all the way through the part) that is 5 mm in diameter.

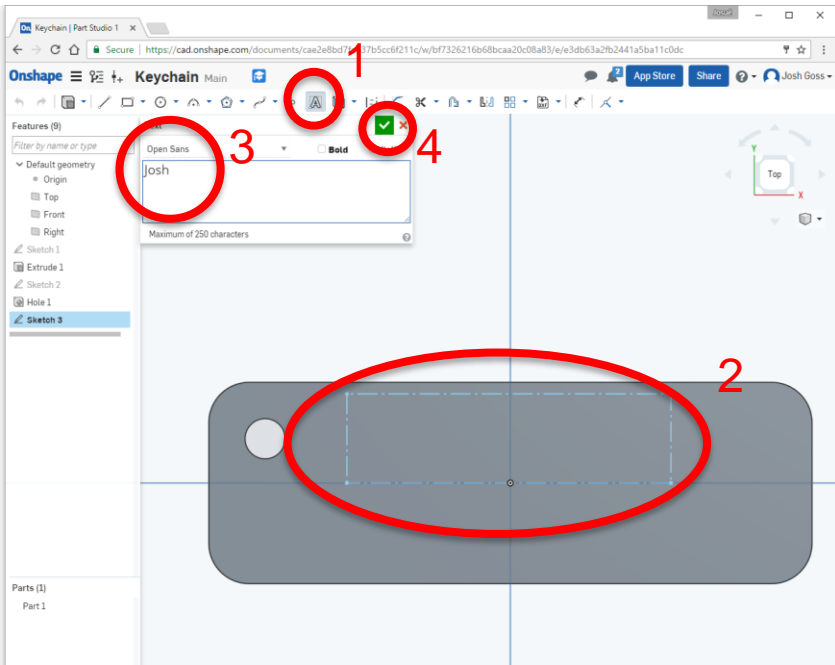
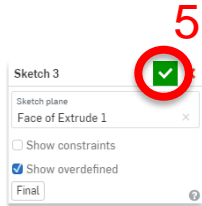
Press  to confirm hole.
















Let's make a sketch to extrude cut your first name. Make a new sketch on the top surface of your part.

Click on text button  and click-drag to make a box for your first name.


Press  in text box to confirm and  again to exit sketch.



Constraints

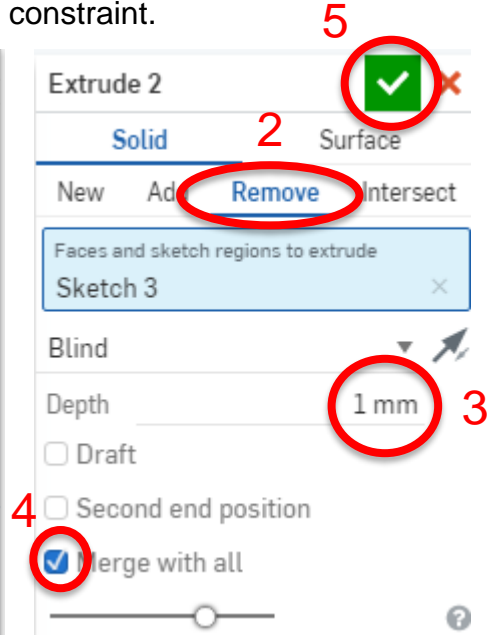
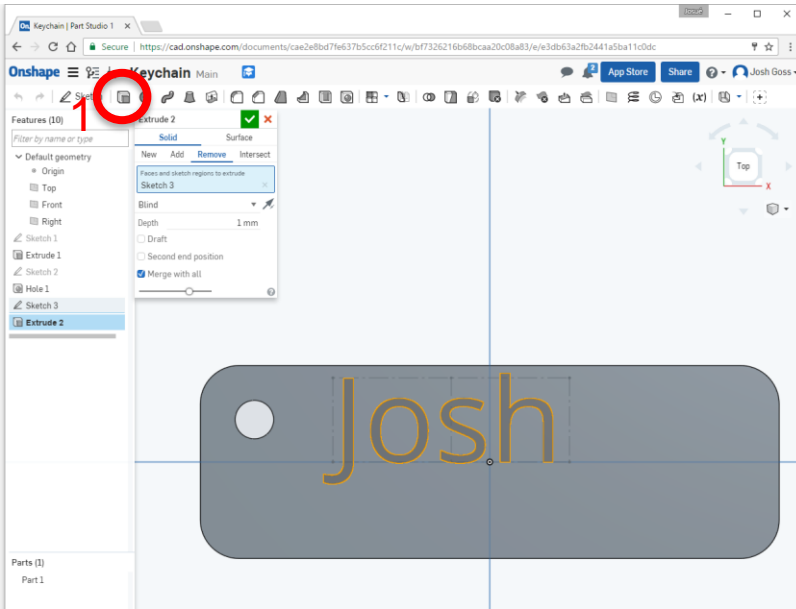
-  Coincident
-  Concentric
-  Parallel
-  Tangent
-  Horizontal
-  Vertical
-  Perpendicular
-  Equal
-  Midpoint
-  Normal
-  Pierce
-  Symmetric
-  Fix

Pro tip: You'll notice some features turn orange when you mouse close to them. This "snap to" feature allows you to quickly and accurately place sketch components that have a relationship with the pre-existing geometry. This feature is called a constraint.

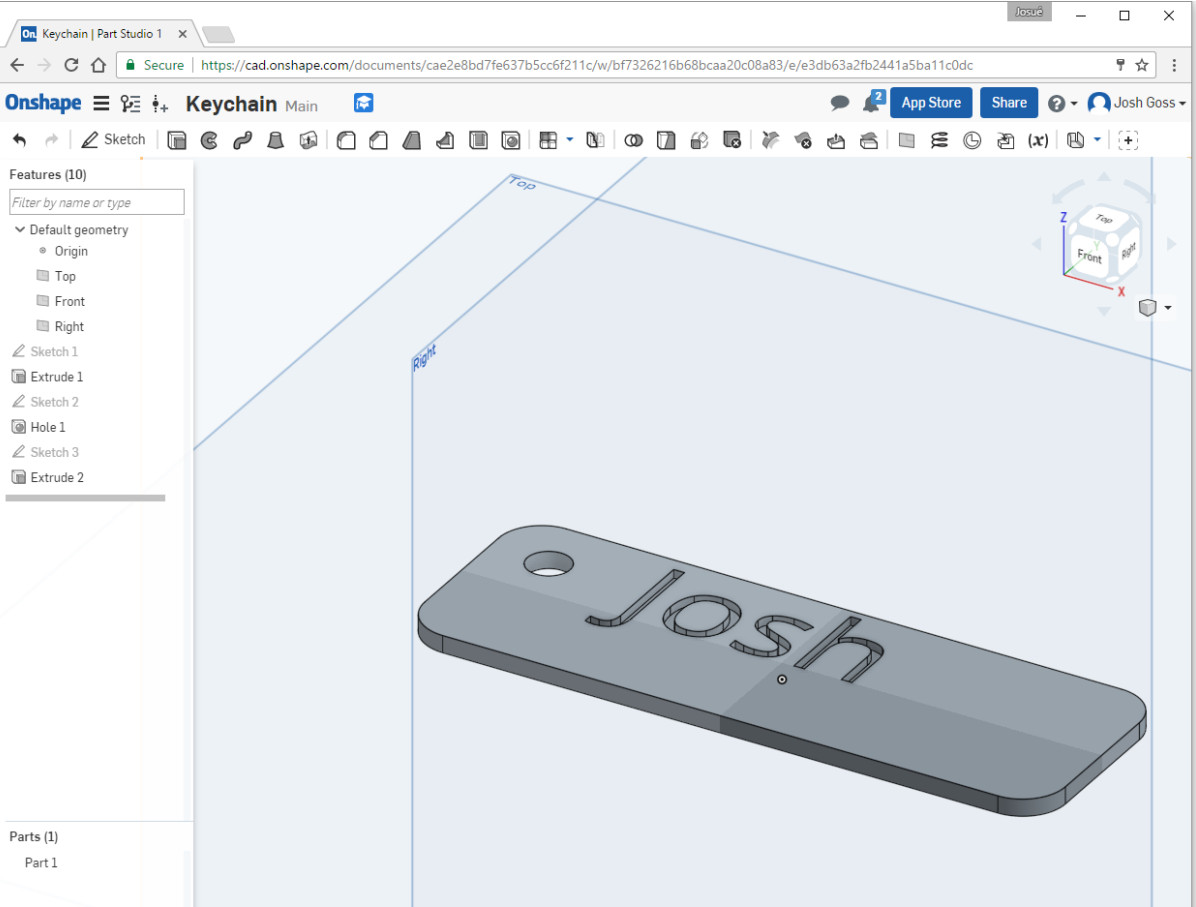
Click on Extrude  select the remove feature and a depth of 1 mm and tick “merge with all”.

Click on  to make the extrude cut.

Pro tip: Make sure your name fits inside the top half of the part without extending to the outer edge which would weaken the sidewall. If you want to move or change the size of your text box, right click (two finger click on Chromebooks) on the Sketch 3 in the left column and select edit to drag and size your text box. If you accidentally established a constraint when you placed the box (you’ll see the line as colored black instead of blue) you can click on the line, then click on the constraint icon and press delete to remove the constraint.

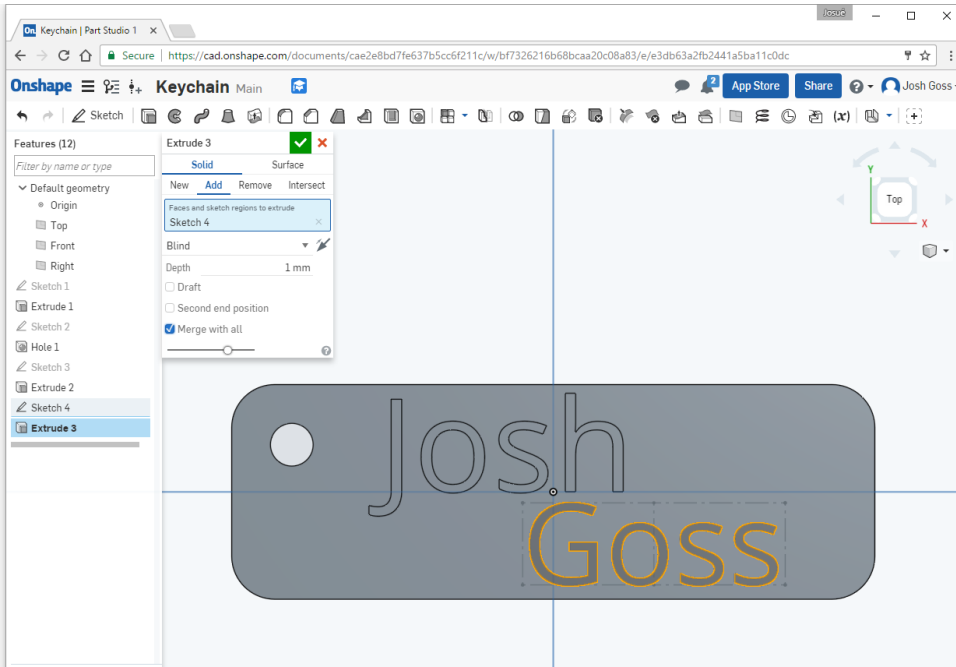
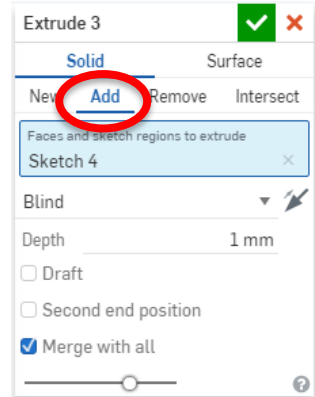


Your first name should now be extruded out (removed) from the solid part volume.



Add your last name exactly like you added your first name with the following changes:

1. Place the text box in the lower half of your keychain
2. Choose “Add” to create a positive extrusion



Your last name should be an extruded solid from the part's top face.

Well done! You just designed your very own 3D CAD model!



To save your part as an STL file for 3D printing, right click on the part 1 at the bottom left of the screen and select “export”.

The default values of binary STL, millimeter units, and fine resolution are great. Rename file to “Keychain” followed by your name and press OK to download the STL.

