

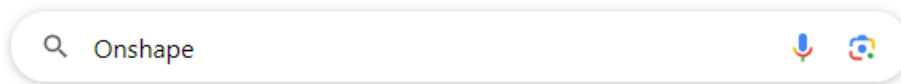
# Onshape Introduction

## Circles and Rectangles and Extrude

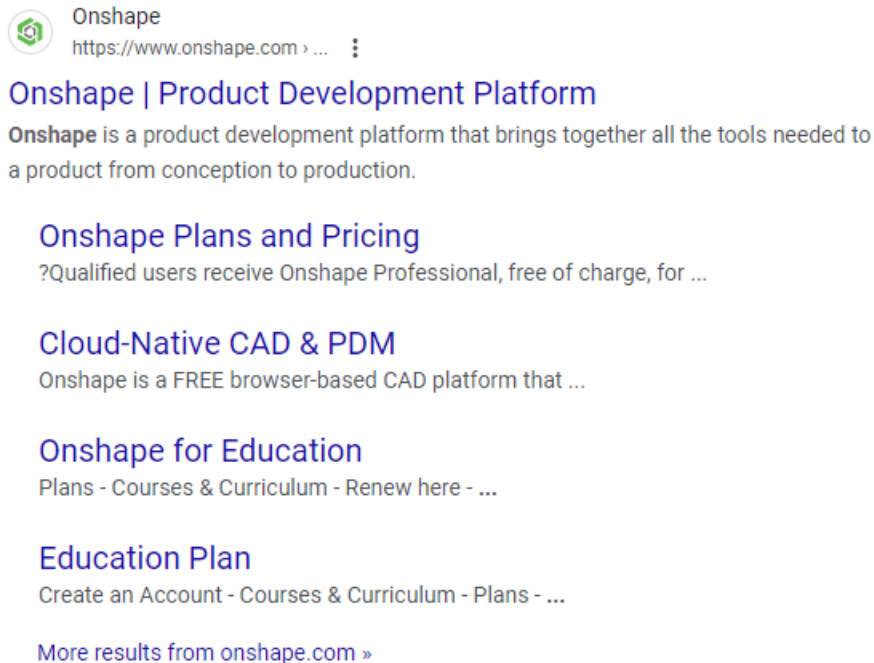
Learning a CAD program does take a lot of practice to master but anyone can learn the basics in a very short time. When starting out you will make mistakes but that is how you learn. Just try not to repeat the mistakes. Build on your successes and stay positive.

To begin, you will need to open the Onshape website. Onshape is a cloud-based program that will work on most computers. It is very helpful to have a mouse to draw the parts.

Type Onshape in the google search bar and click enter.



Left click on the Onshape for Education to open the program and create an account.

A screenshot of Google search results for "Onshape". The top result is from "Onshape" with the URL "https://www.onshape.com". The main heading is "Onshape | Product Development Platform". Below it is a description: "Onshape is a product development platform that brings together all the tools needed to a product from conception to production." There are four sub-links: "Onshape Plans and Pricing" (with a snippet: "?Qualified users receive Onshape Professional, free of charge, for ..."), "Cloud-Native CAD & PDM" (with a snippet: "Onshape is a FREE browser-based CAD platform that ..."), "Onshape for Education" (with a snippet: "Plans - Courses & Curriculum - Renew here - ..."), and "Education Plan" (with a snippet: "Create an Account - Courses & Curriculum - Plans - ..."). At the bottom is a link for "More results from onshape.com »".

Onshape  
https://www.onshape.com > ...

### Onshape | Product Development Platform

Onshape is a product development platform that brings together all the tools needed to a product from conception to production.

#### Onshape Plans and Pricing

?Qualified users receive Onshape Professional, free of charge, for ...

#### Cloud-Native CAD & PDM

Onshape is a FREE browser-based CAD platform that ...

#### Onshape for Education

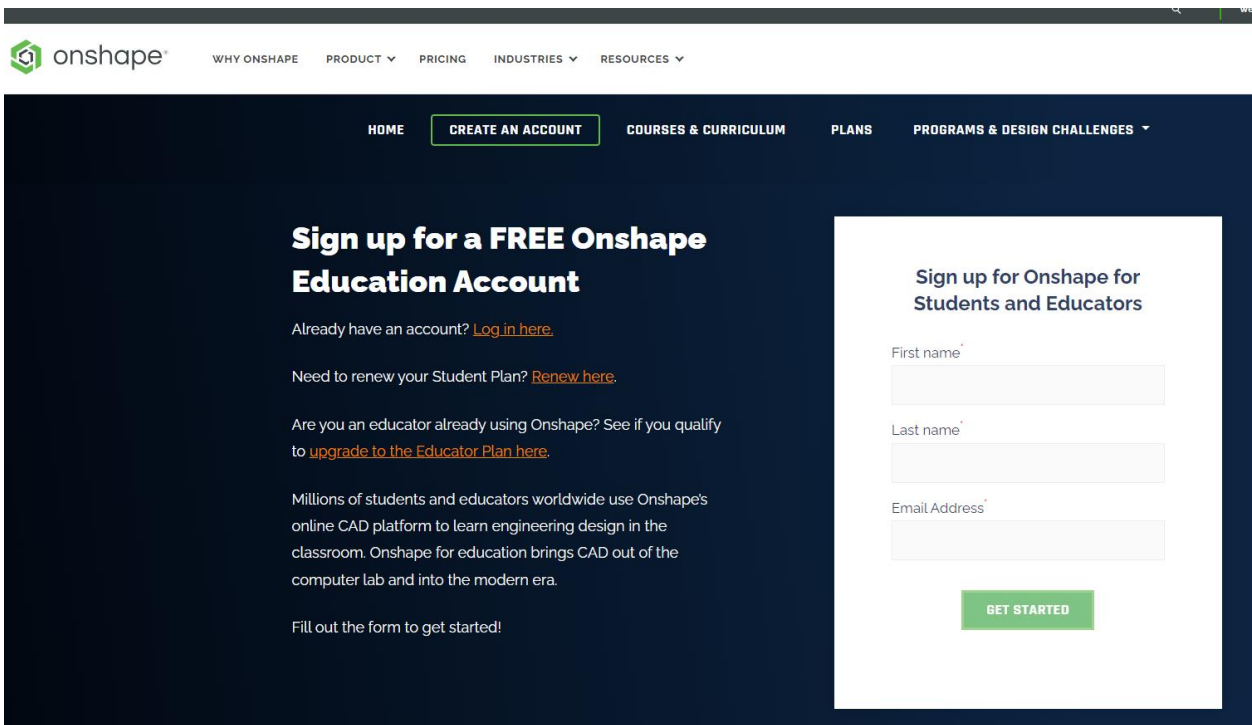
Plans - Courses & Curriculum - Renew here - ...

#### Education Plan

Create an Account - Courses & Curriculum - Plans - ...

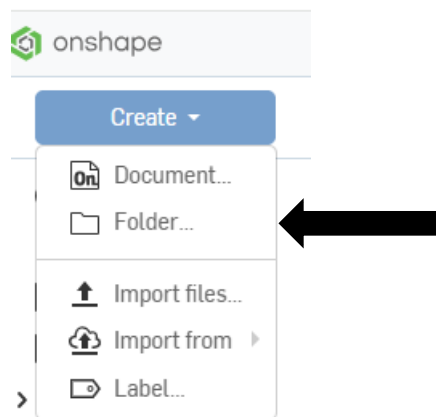
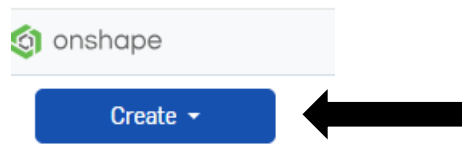
[More results from onshape.com »](#)

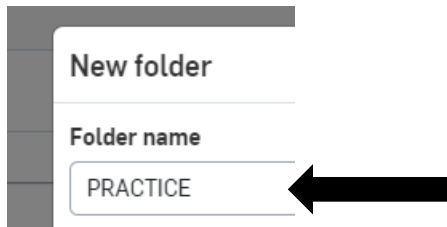
Fill in the boxes and be sure to use your school email account.



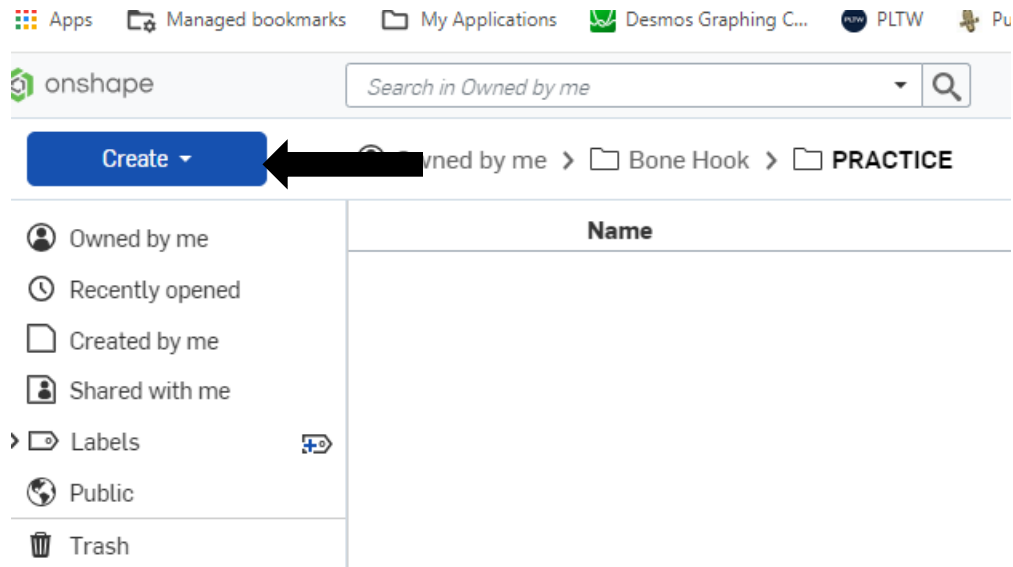
Follow the prompts to complete your account. After you complete your setup you will need to create a folder before drawing any sketches. The folders you create will help keep you organized. The first folder should be PRACTICE so you can learn the basics and keep your files organized.

Left click on the green Onshape logo on the left side of the ribbon to create a new folder and then click on folder. Name the folder PRACTICE and left click Create.

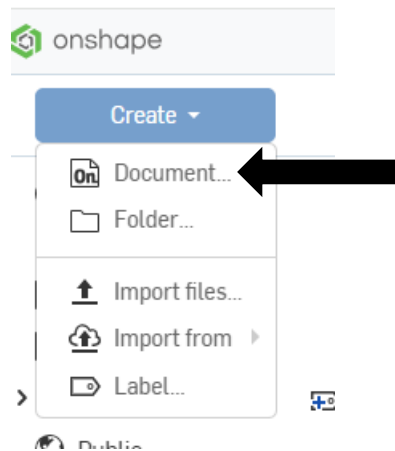




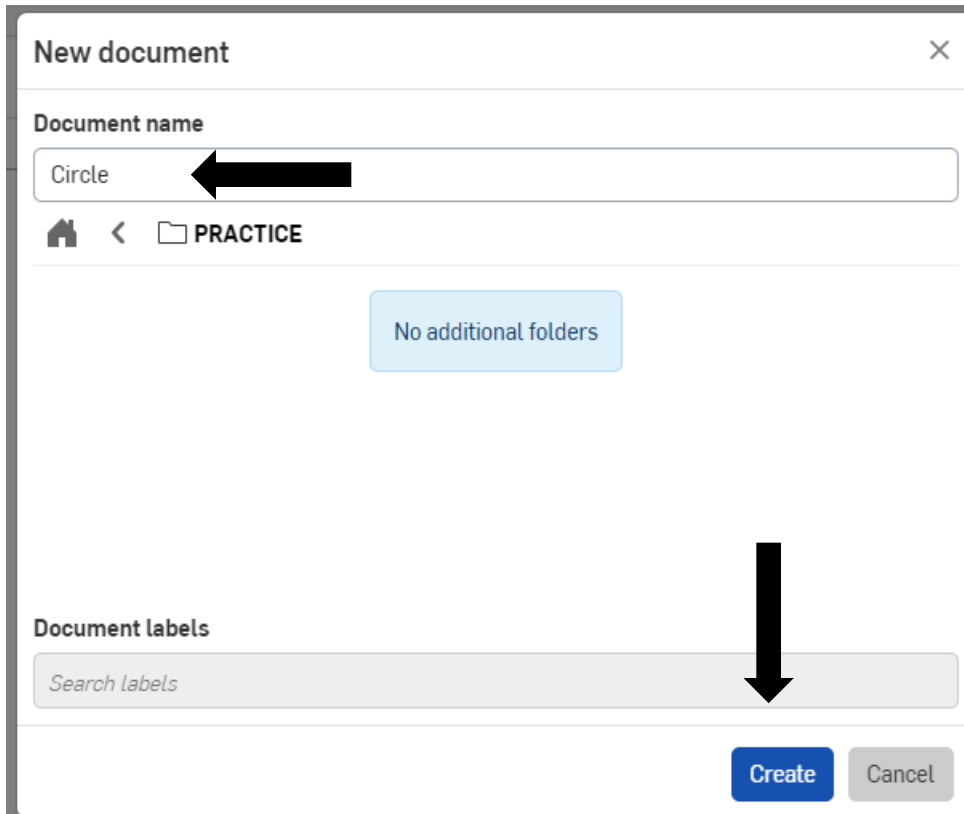
Double left click on the PRACTICE folder to make it the active folder where all the practice drawings will be stored. The PRACTICE folder will be darker so you know it is active.



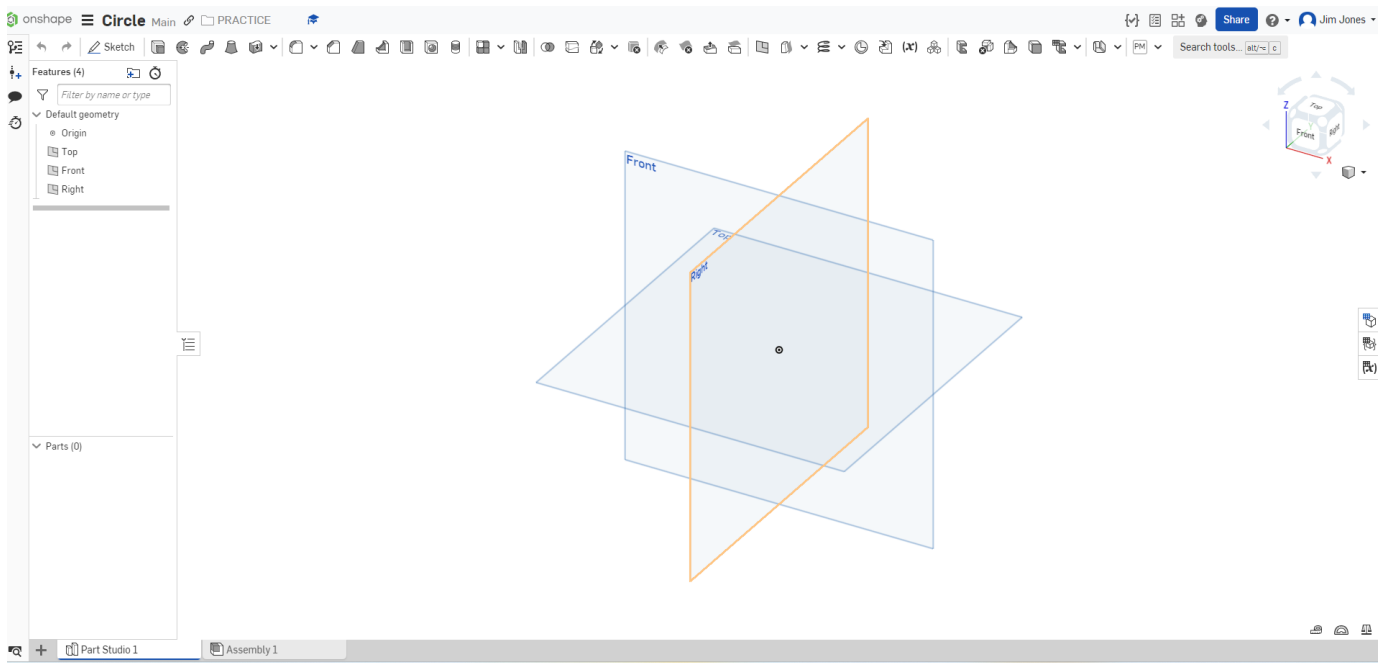
Next you will left click on the blue Create drop down to create a document which is actually the new drawing part.



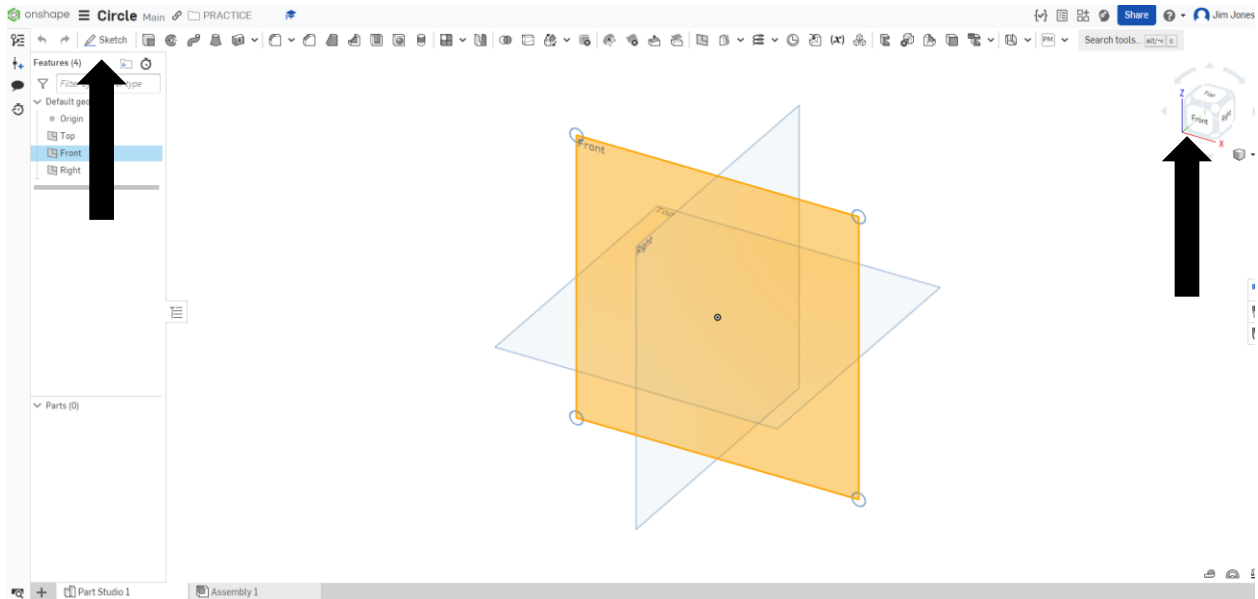
Name the Document Circle and then left click on Create. You will now create some circles.



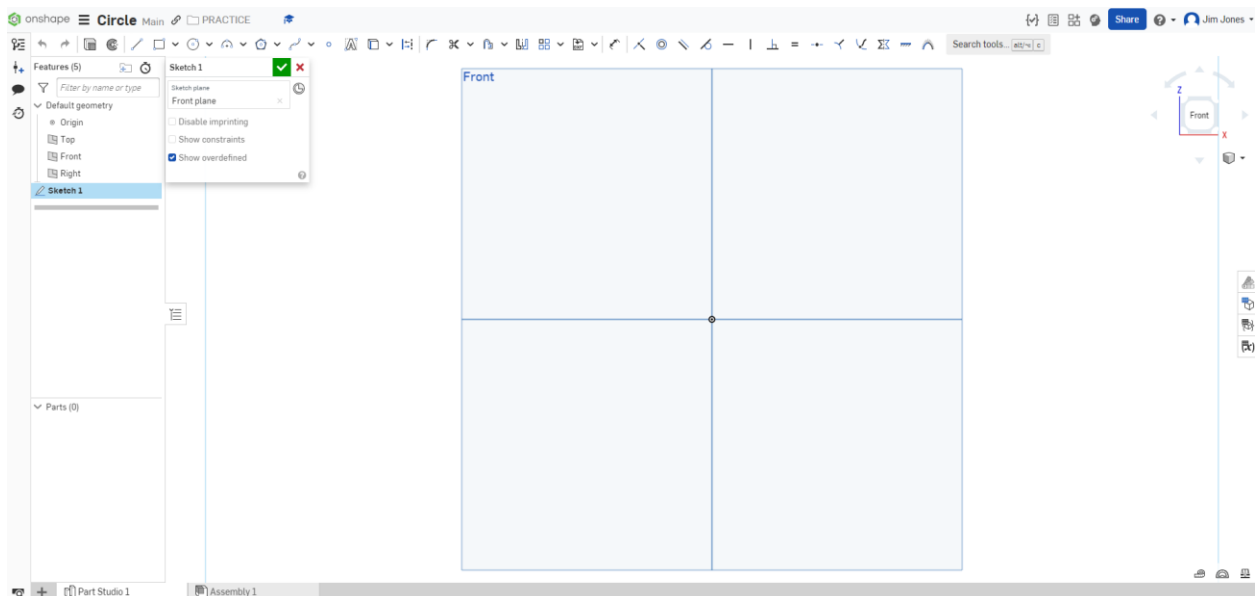
The whole screen changes and you can now Sketch some circles. You will need to know the parts of the screen so take a minute to get familiar with the different areas. The top line shows the name of the document Circles. There is a line of objects below the document name along with the browser on the left side and the large drawing area to create the circles. The three drawing planes are labeled Front, Top and Right. In the top right corner is the view cube with the XYZ coordinates.



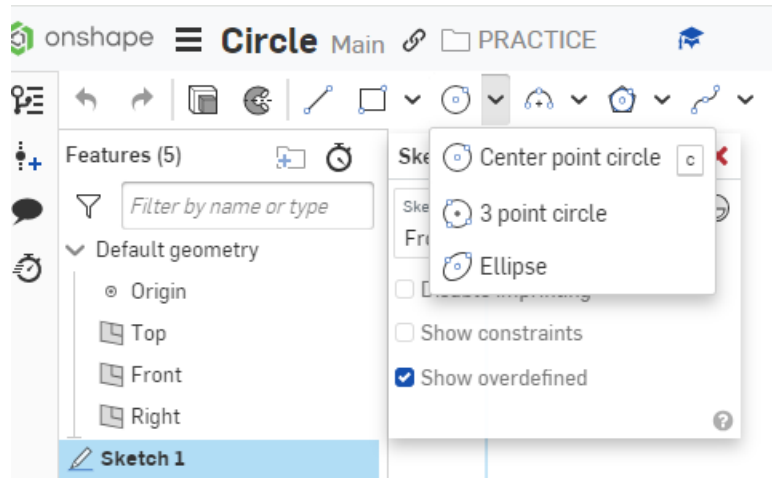
When you hover over the lines of the work planes they will turn yellow and you want the Front plane to start the sketch. Left click on the Front plane and then left click on Sketch in the ribbon.



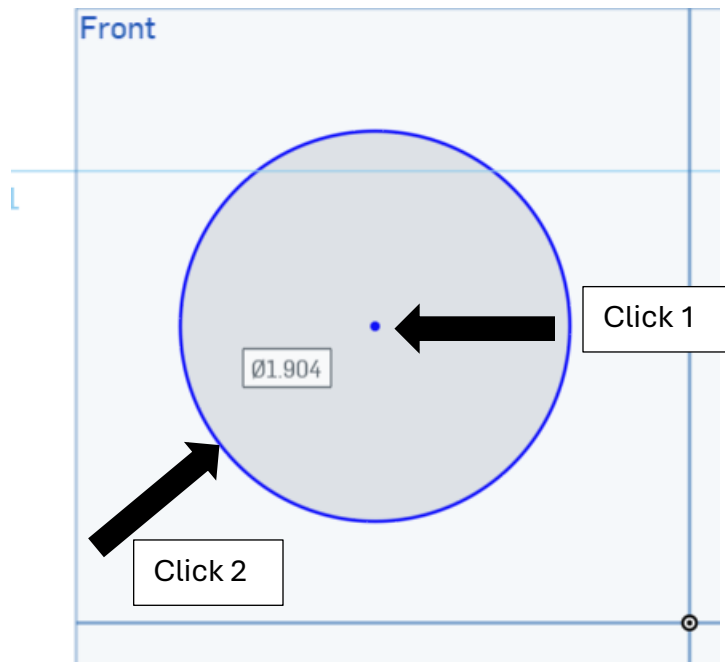
After you left click on Sketch, left click on Front on the view cube in the top right corner. The Front plane will rotate and fill the drawing space.



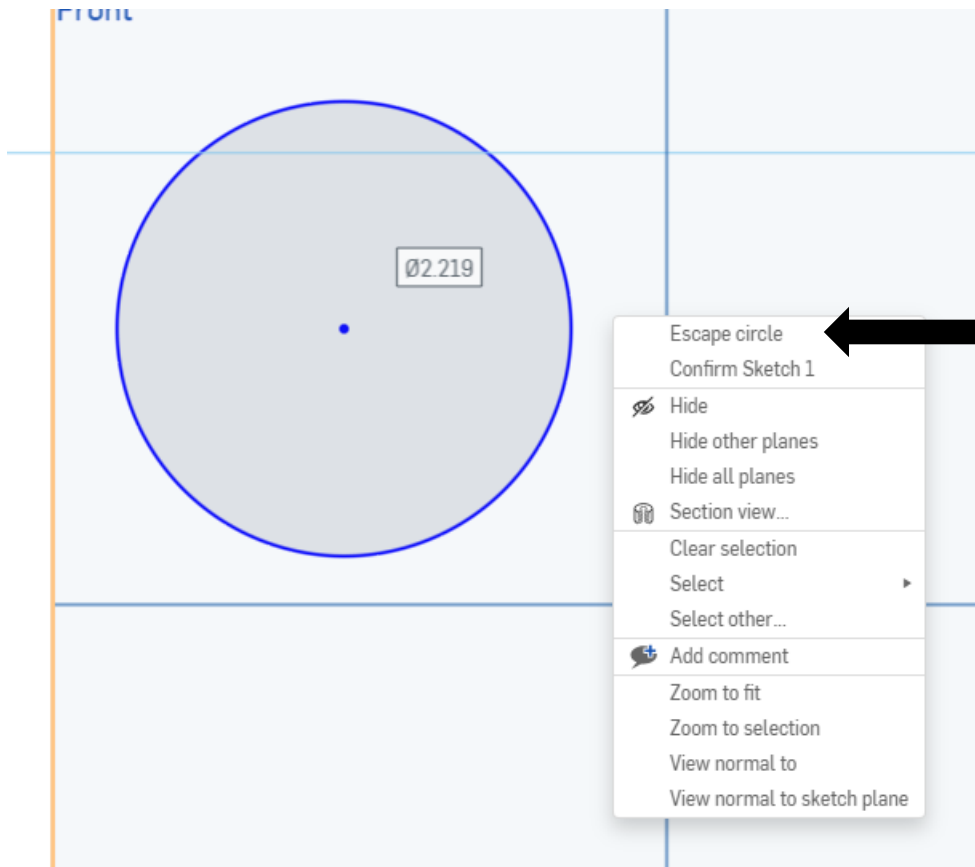
Notice the top ribbon has changed to show the sketch objects available to make a drawing. There is also a Sketch 1 box next to the Browser in the top right side. Hover over the drawing icons and click on some of the drop down arrows to show how many options are available for each one. There are Lines, Circles, Arcs, Polygons, Splines, Points, Text and many more. You will draw at least two of each type of Circle on the Front plane.



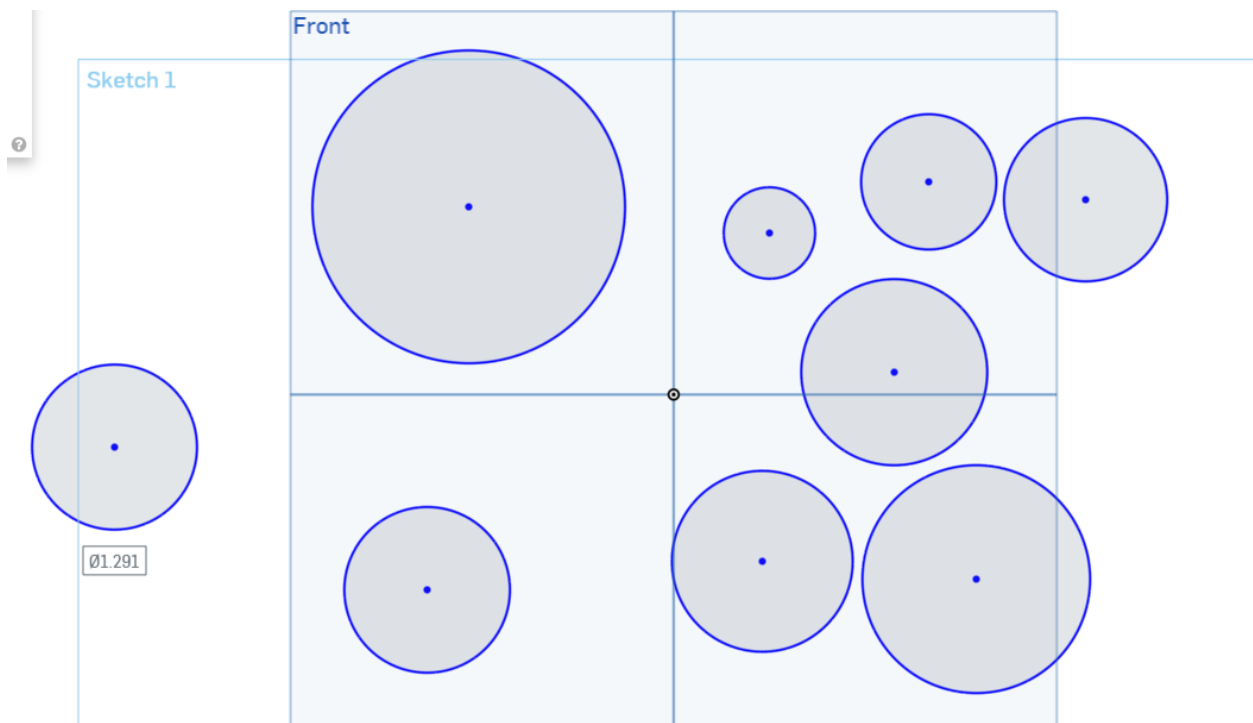
Left click on the Center point circle. The circle will take two clicks of the mouse. First click is the center point circle and the second click sets the outside of the circle. The diameter of the circle is shown as you move the cursor.



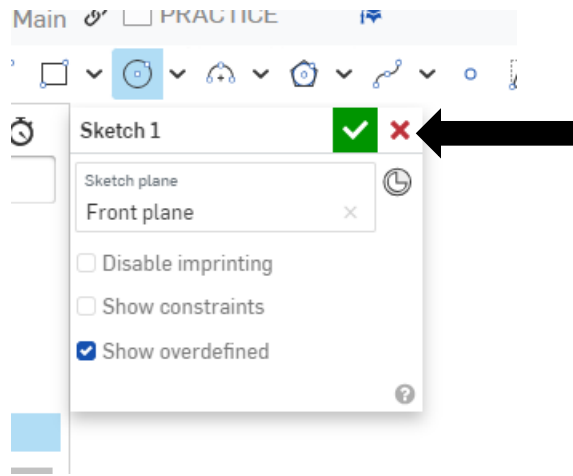
Right click the mouse and then click on Escape circle to finish the circle. Another way to set the circle is to press the ESC button on the keyboard.



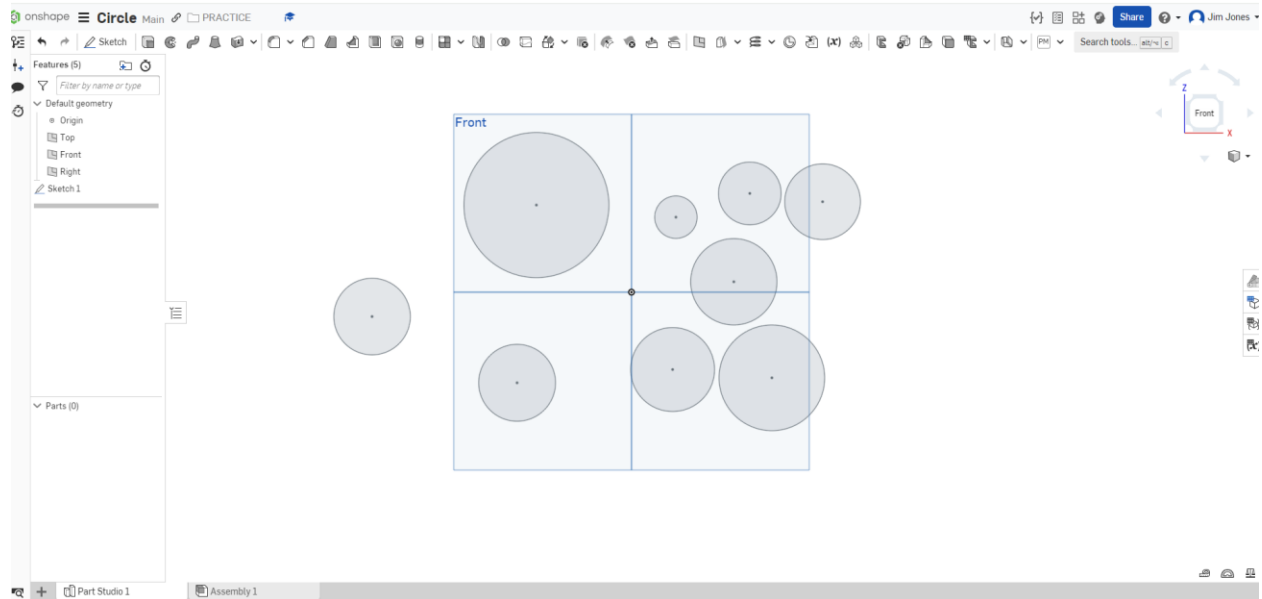
If you don't escape circle or press the ESC button on the keyboard you will continue to make circles. Try it.



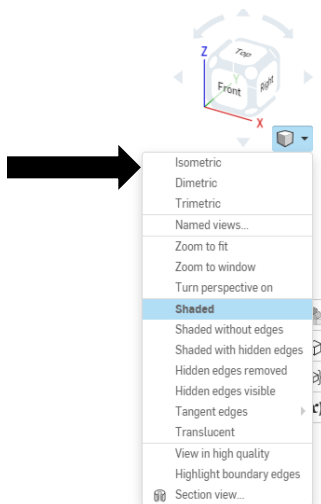
To finish this sketch, left click on the green check mark on the sketch 1 menu to save all the circles.



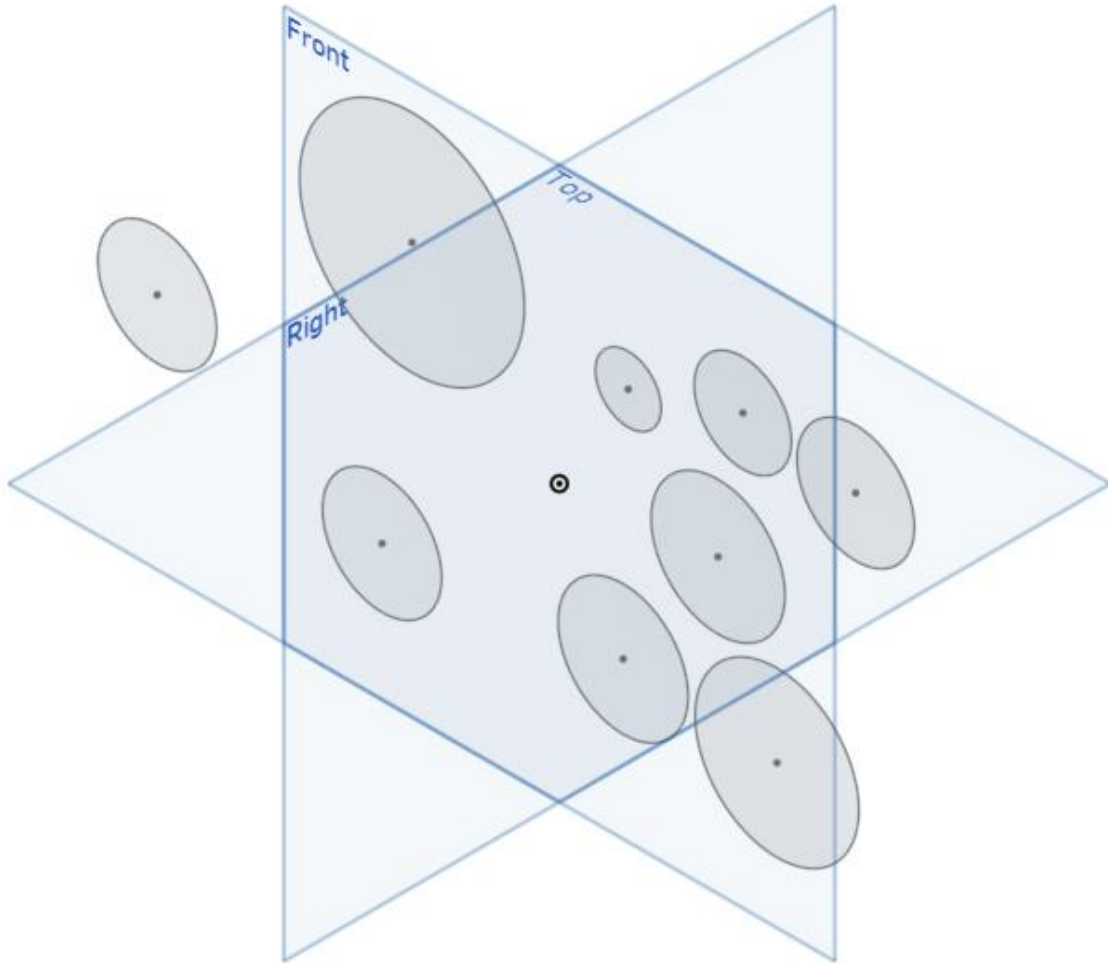
After clicking the green check mark, the circles are set and the 3D model ribbon returns.



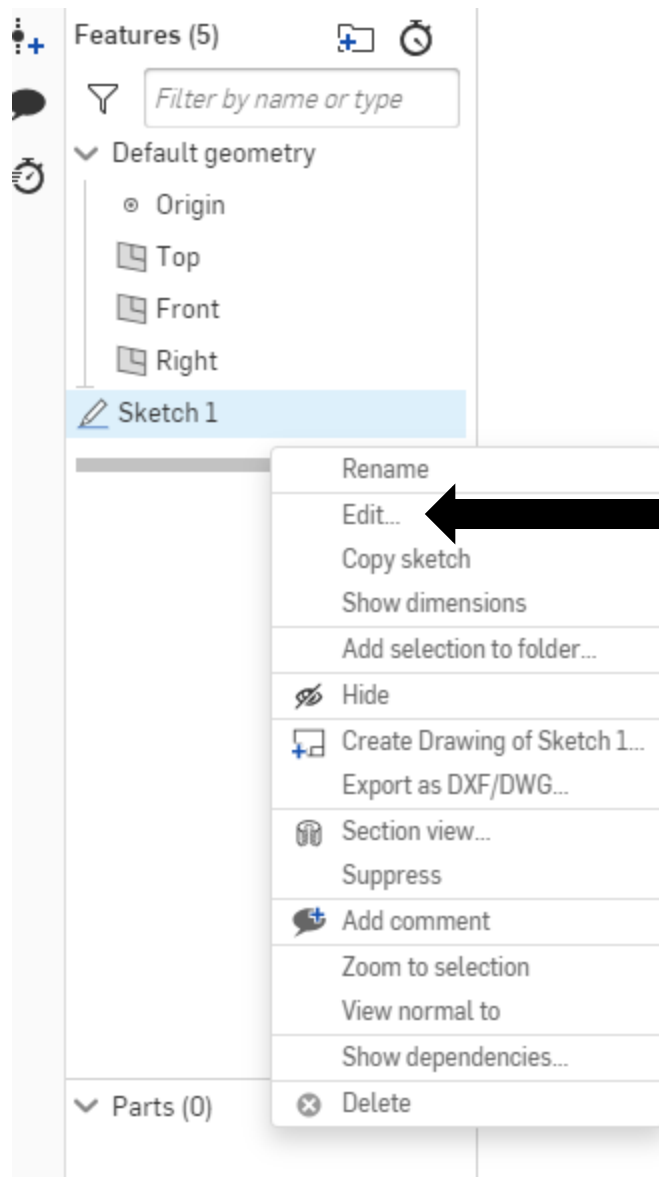
The view cube will rotate the view of the screen clicking on the Front, Top or Right. You can also click on Isometric to view the drawing in an isometric angle.



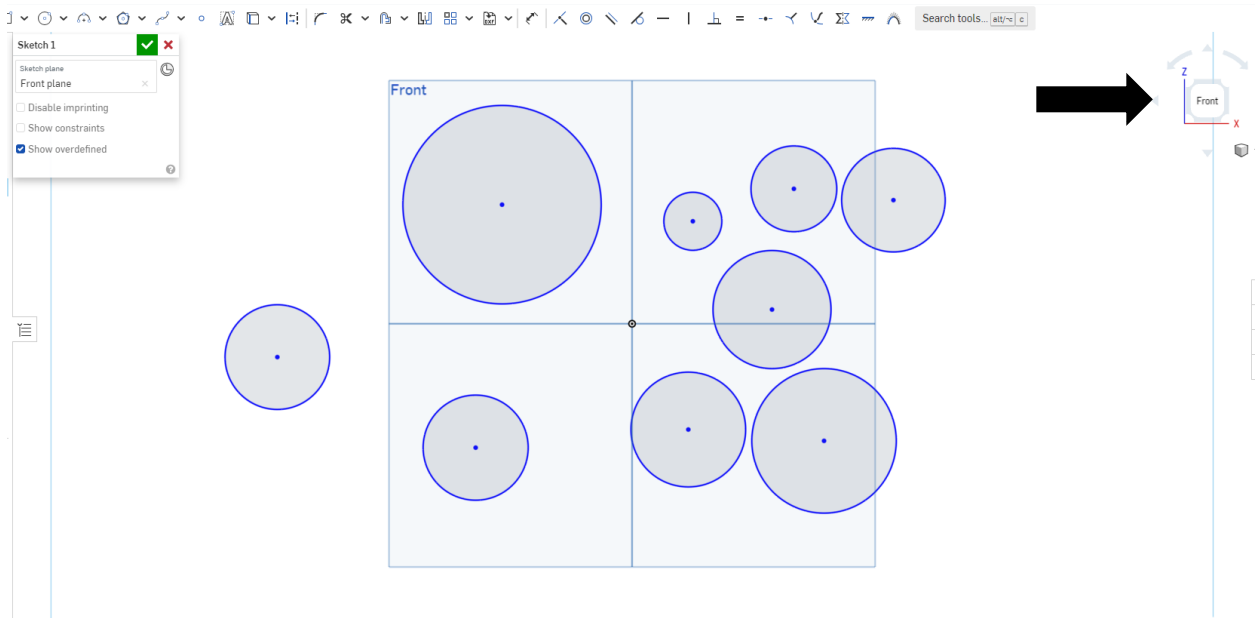




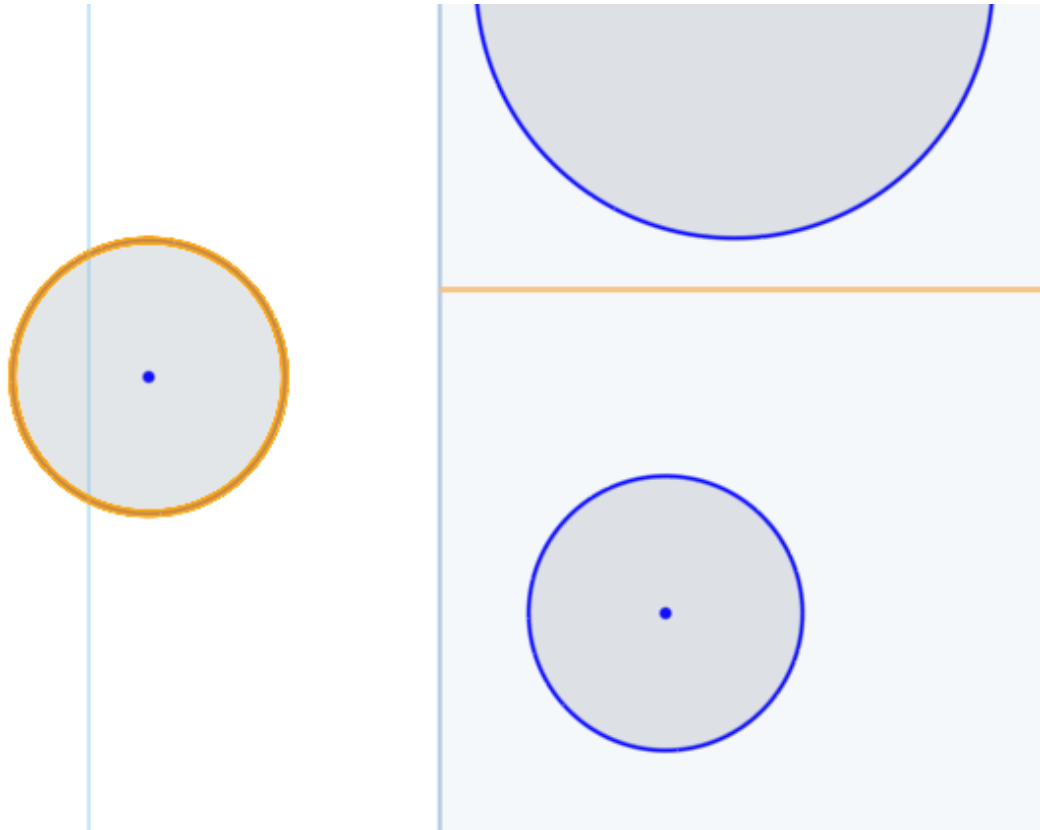
This is a great angle to use the Extrude tool to add depth to the parts. Before Extruding the parts we need to continue to draw circles using the other two methods. You will need to right click on Sketch 1 and Edit the sketch to delete most of the extra circles.

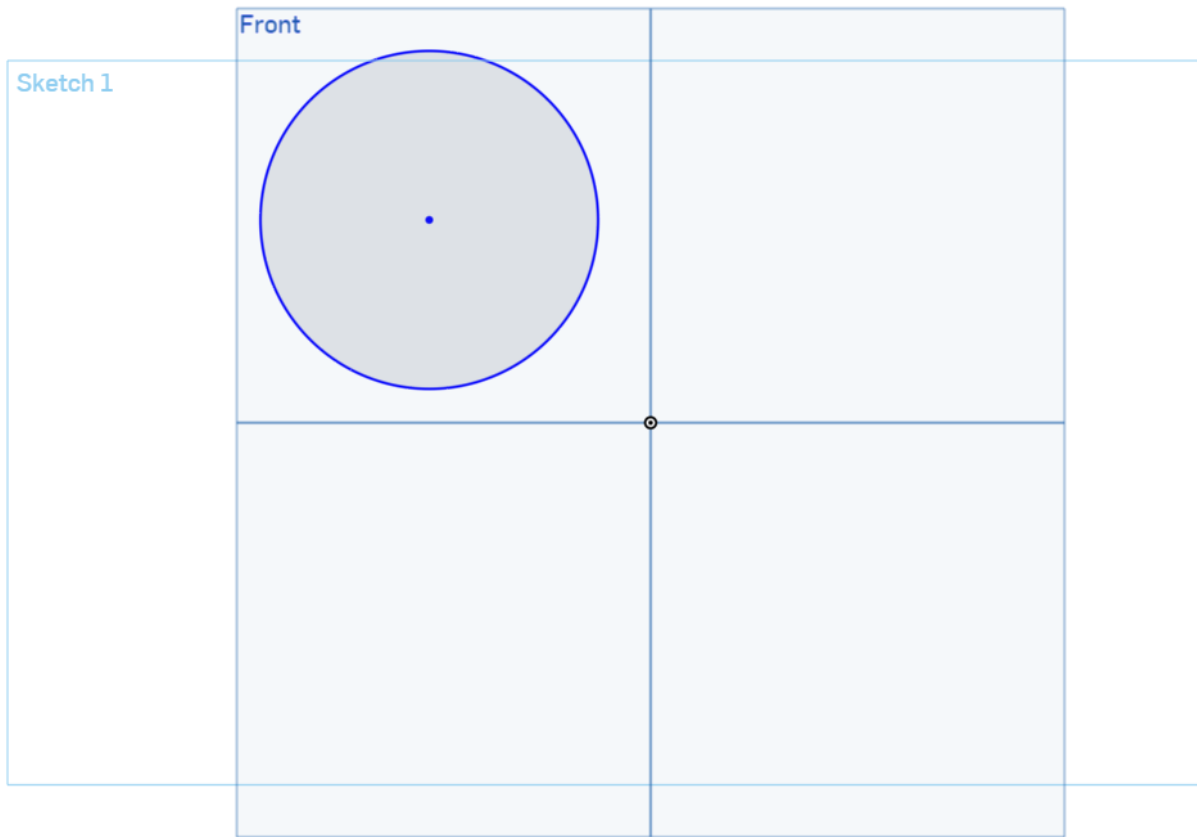


Now the circles are highlighted. Left click on Front in the view cube to bring back the Front view and then delete all the circles except the first circle.

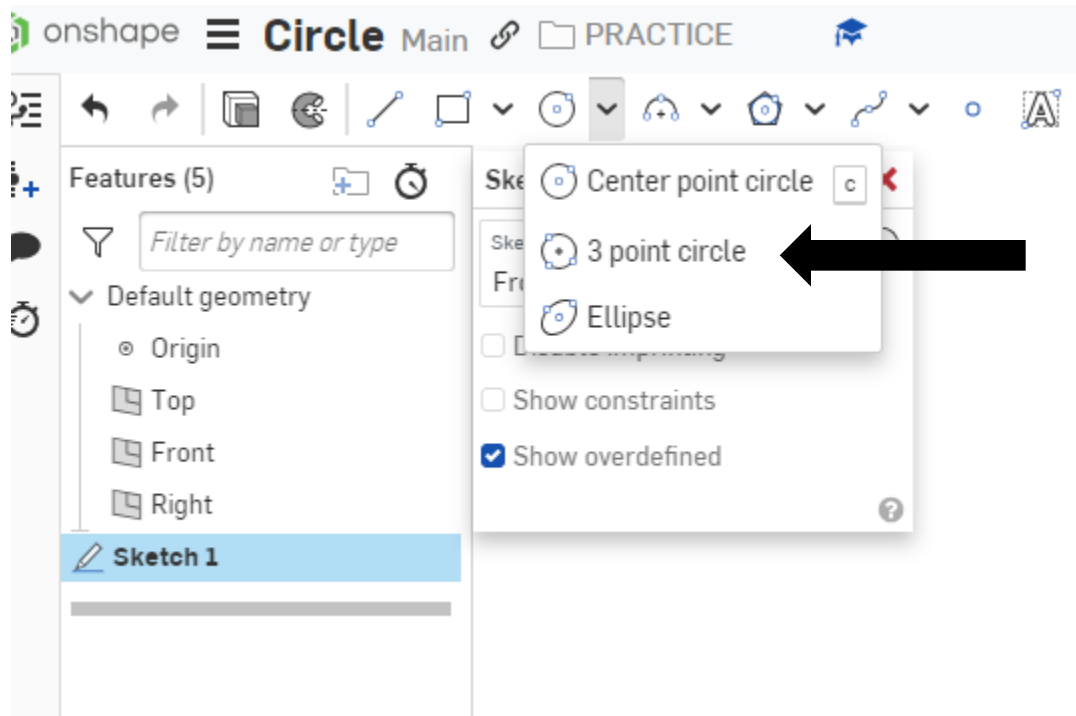


Left click on a circle and it will change to yellow and then press the delete button.  
Continue to delete all but the first circle.

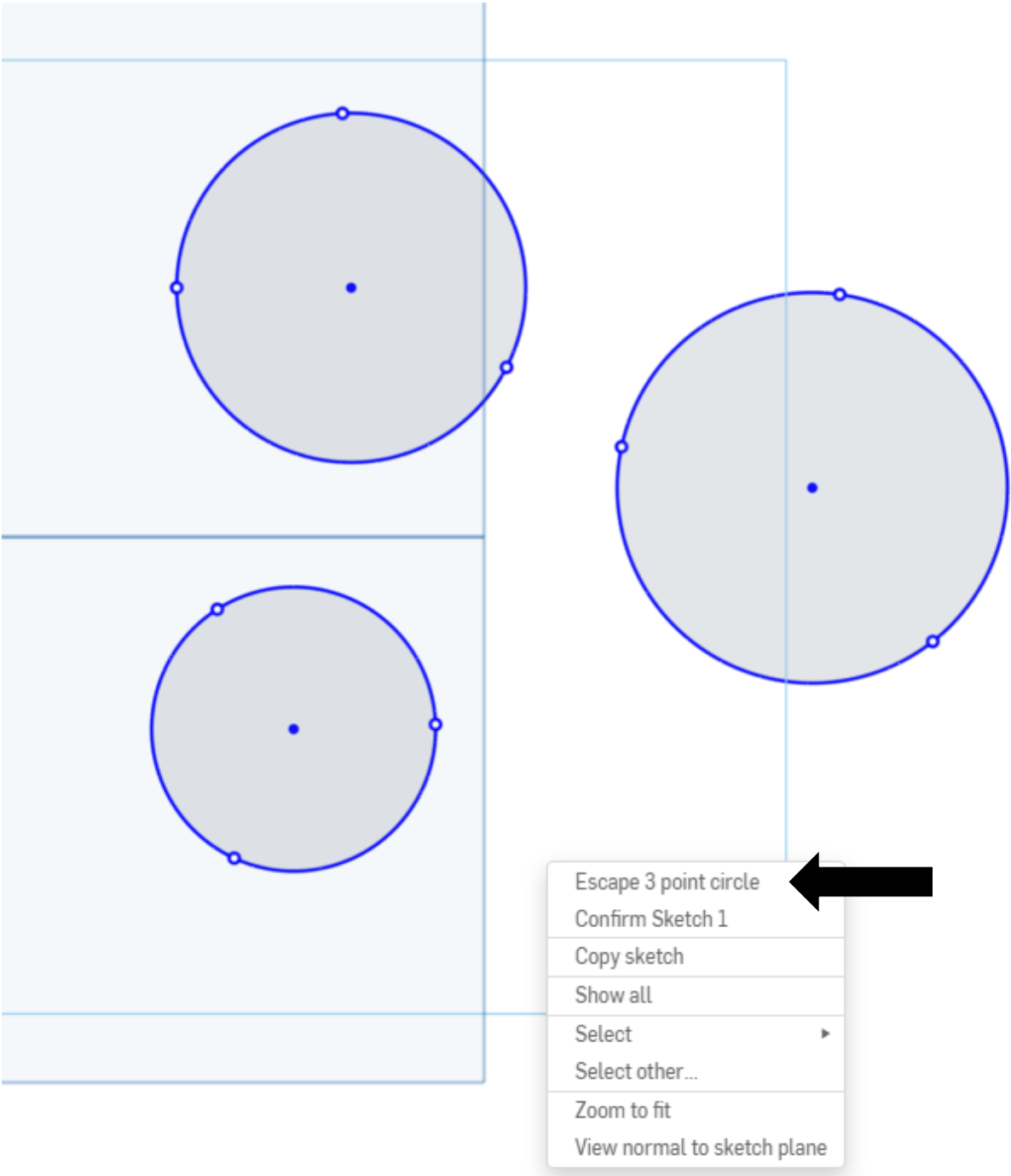




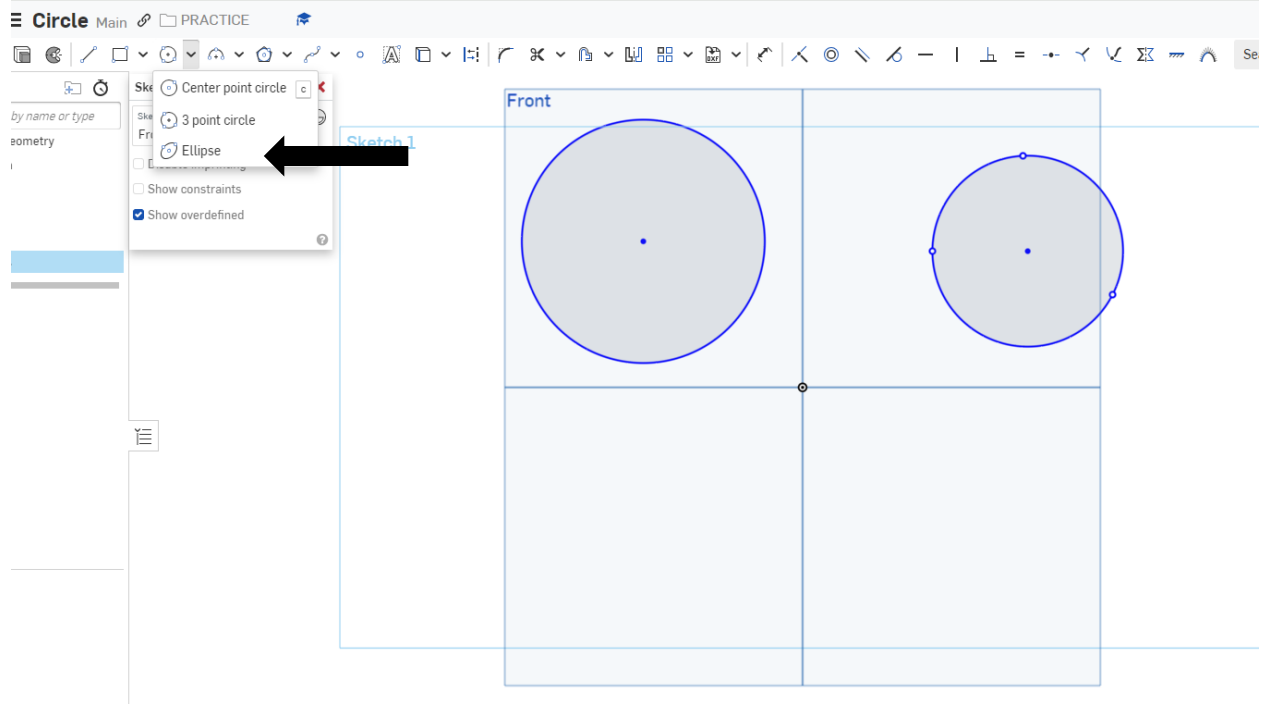
Left click on the Circle drop down arrow and select 3 point circle. Left click in the top right corner of the quadrant and then create a circle with 3 left clicks.



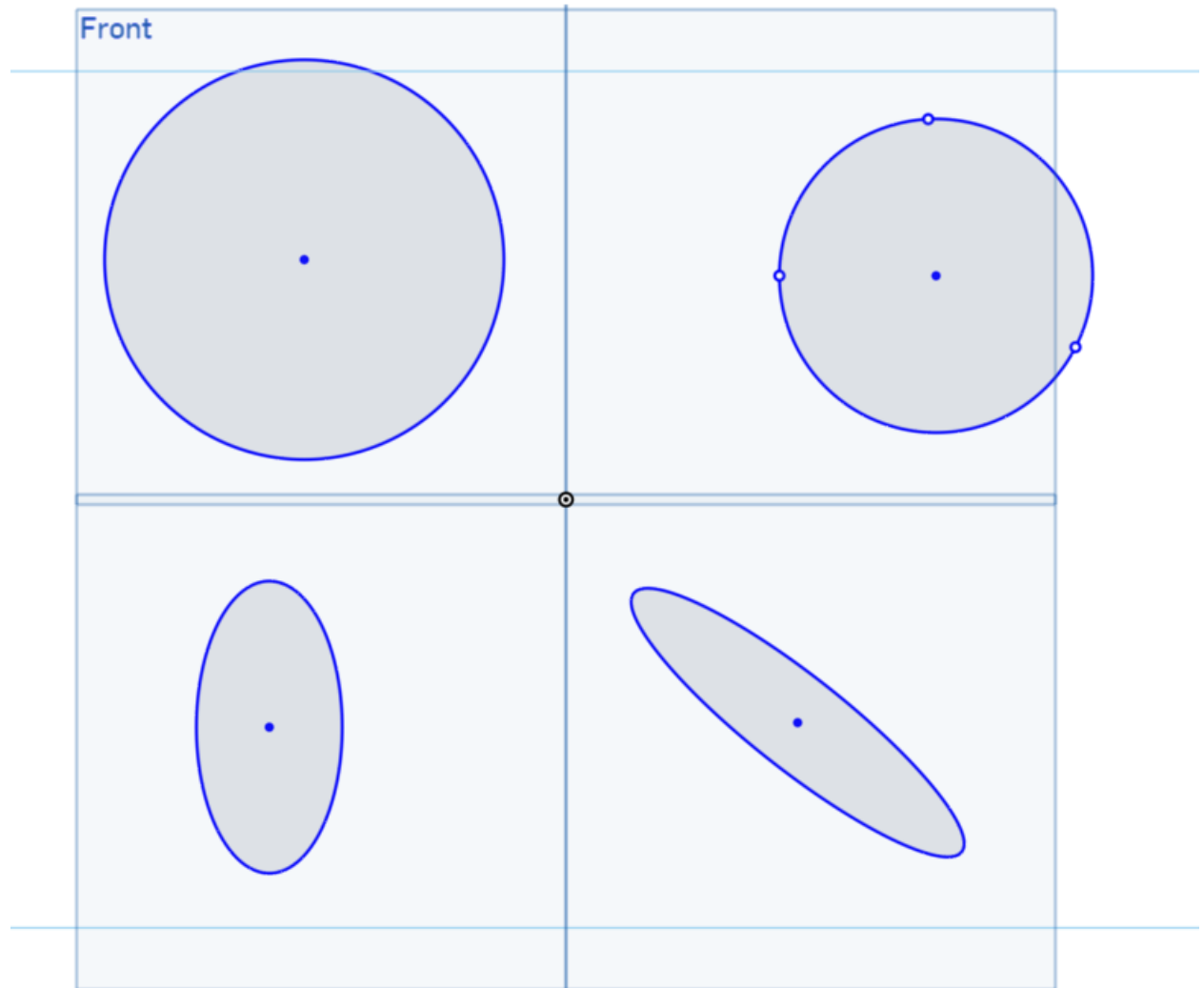
The three points you clicked will show on the screen and you can make a few more 3 point circles. Right click the mouse and left click on Escape 3 point circle or just press the Escape key to stop making circles.



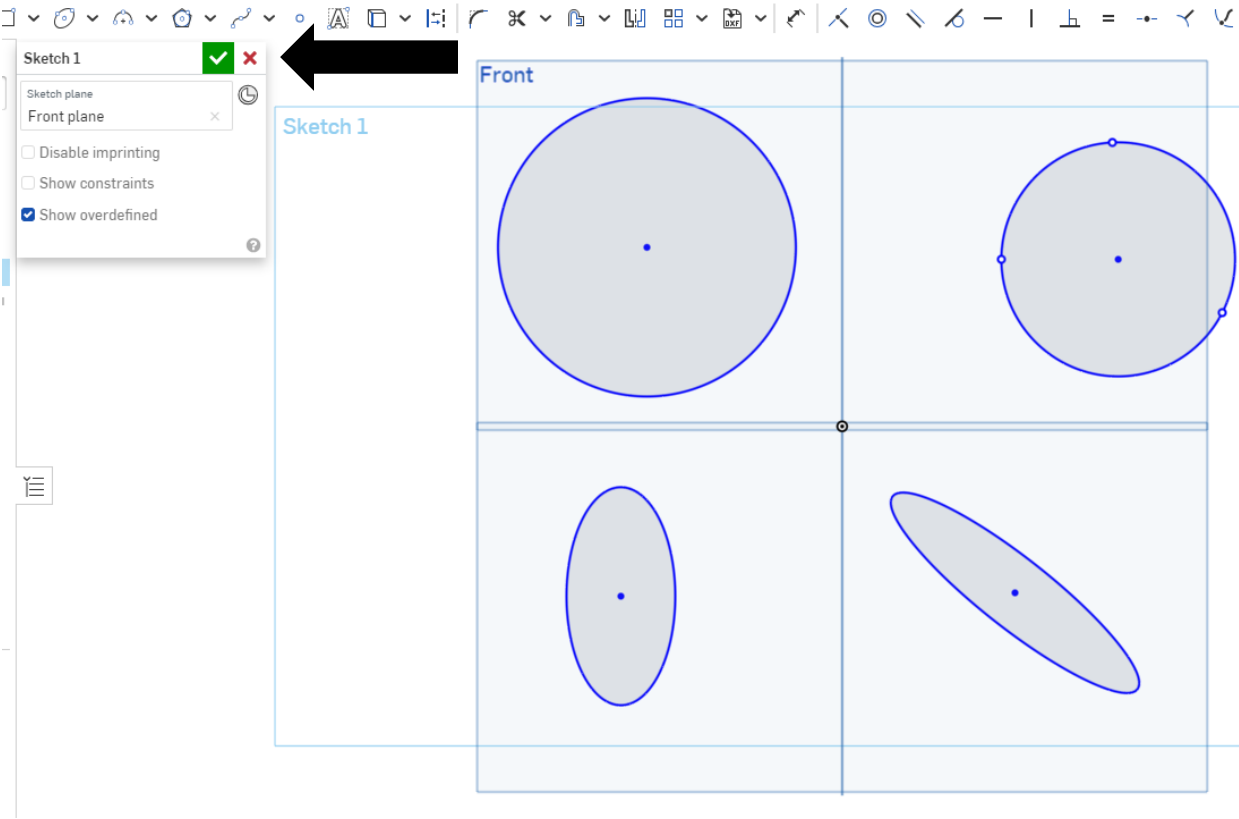
Left click on all but the first 3 point circle to delete the extra circles so only the first 3 point circle is still on the screen. Then left click on the circle drop down to select the third choice, ellipse.



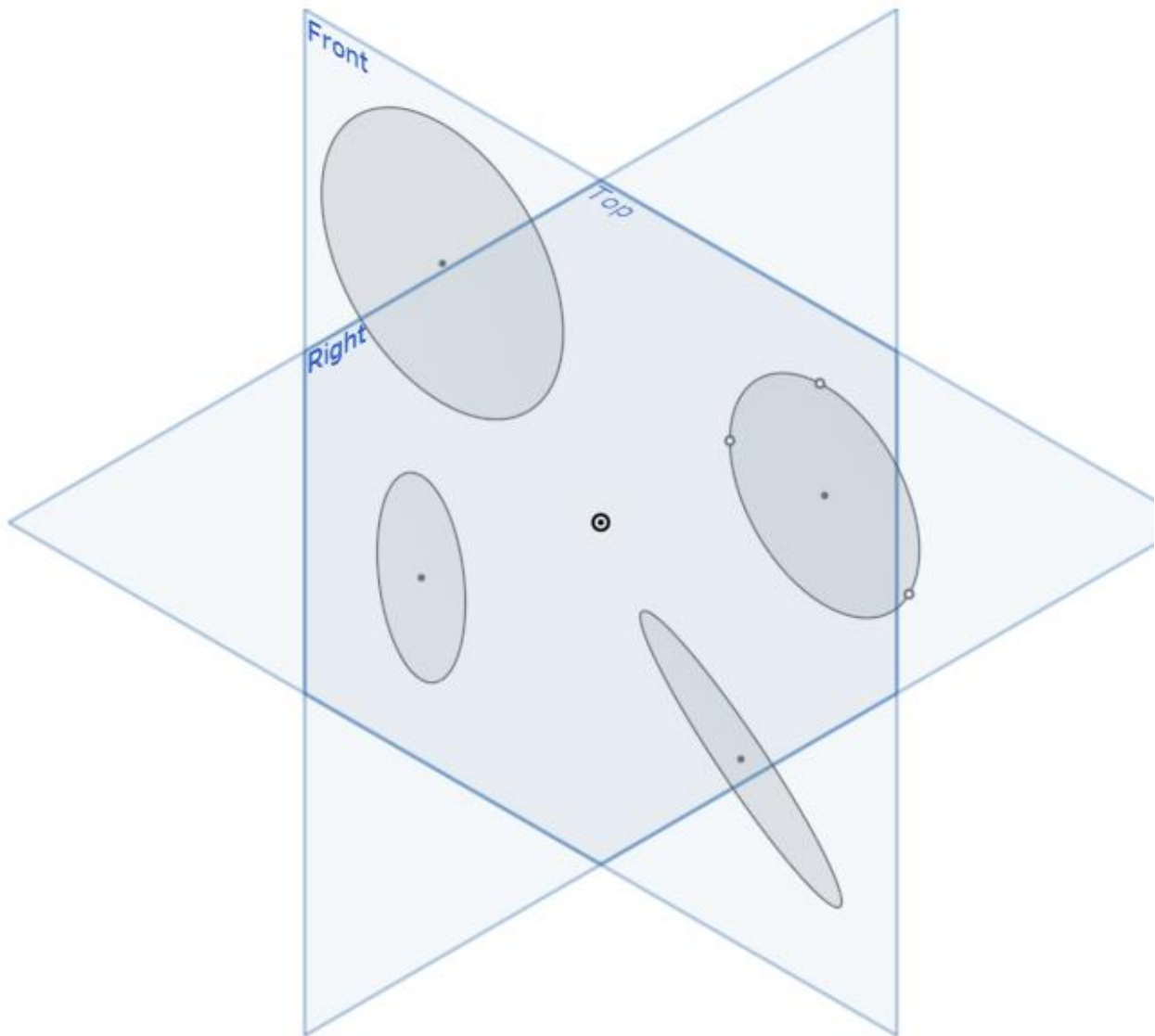
The ellipse will also take three left clicks. Create a couple of ellipses on the screen.



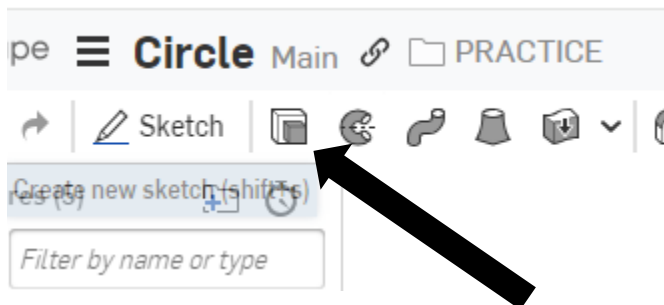
Finish Sketch 1 by clicking on the green check mark.



Left click on the Isometric button to rotate the screen.

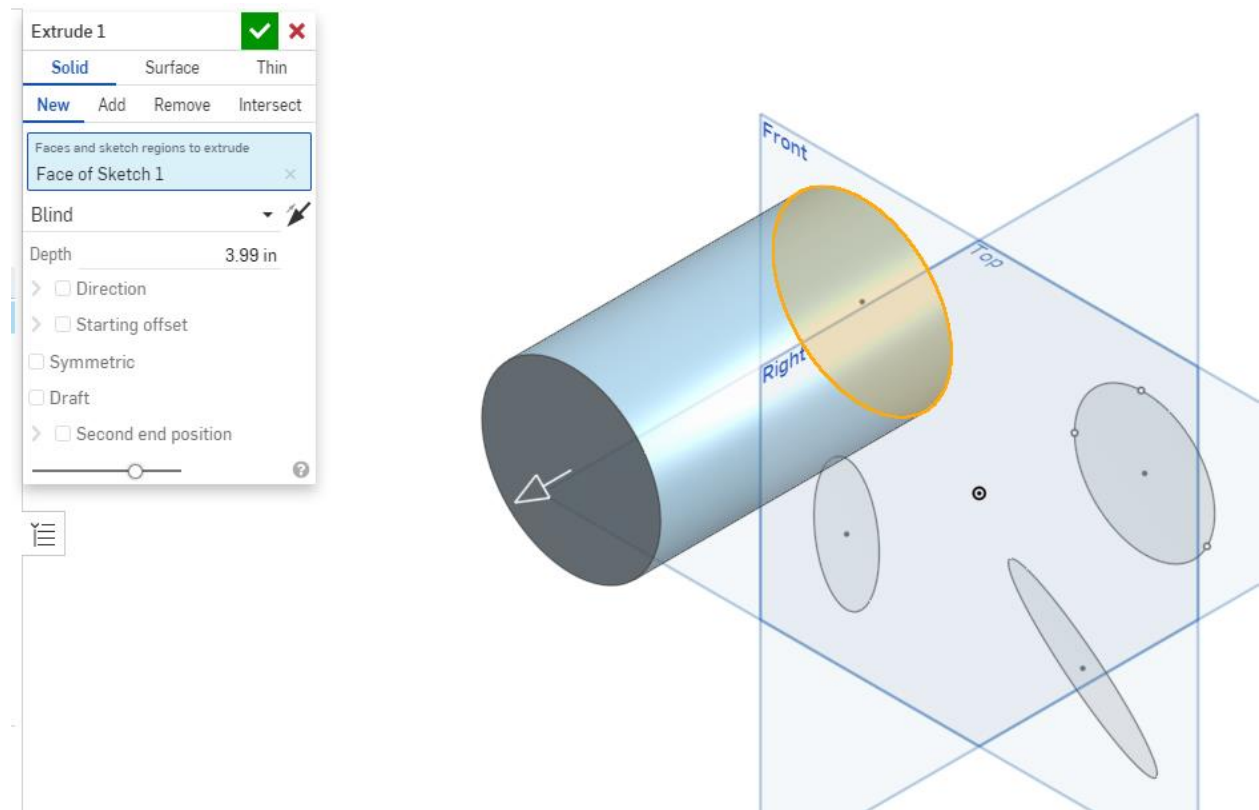
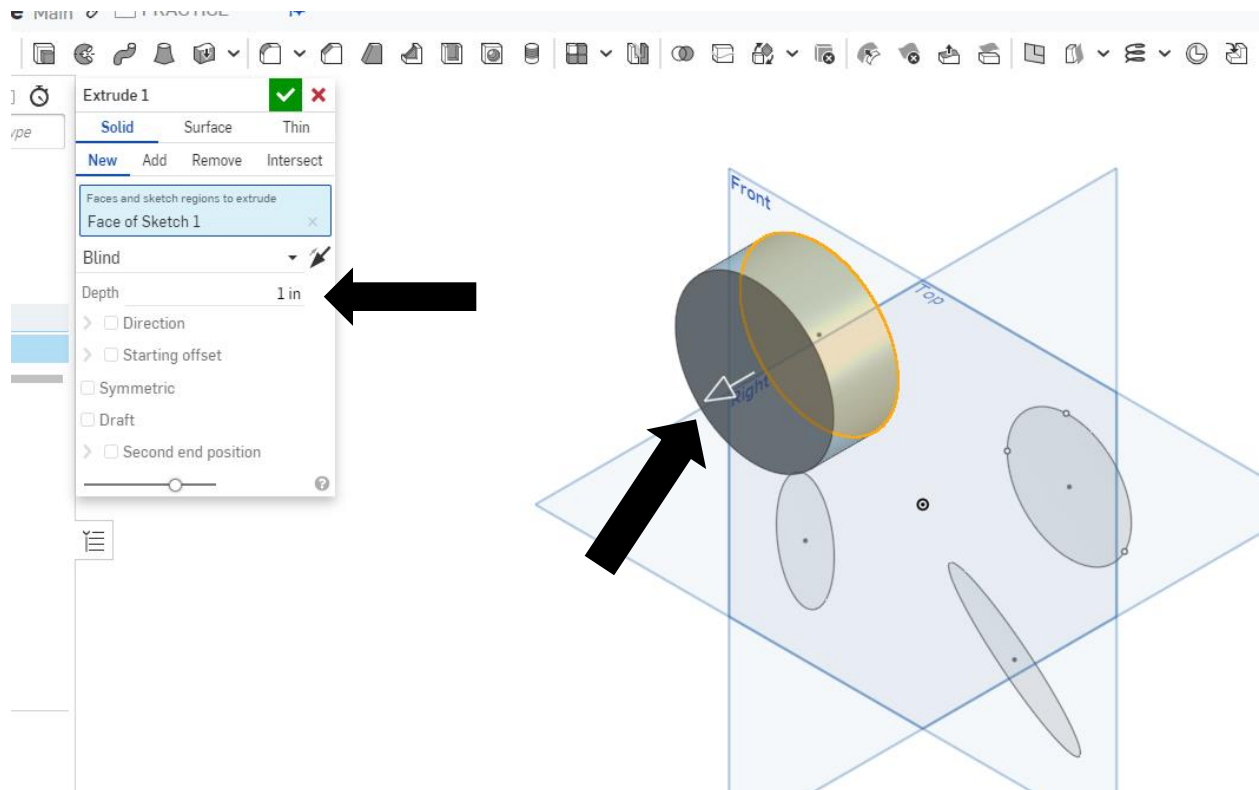


It is now time to Extrude the circles using the Extrude icon on the top ribbon. Left click on the Extrude icon and then left click on the first circle.

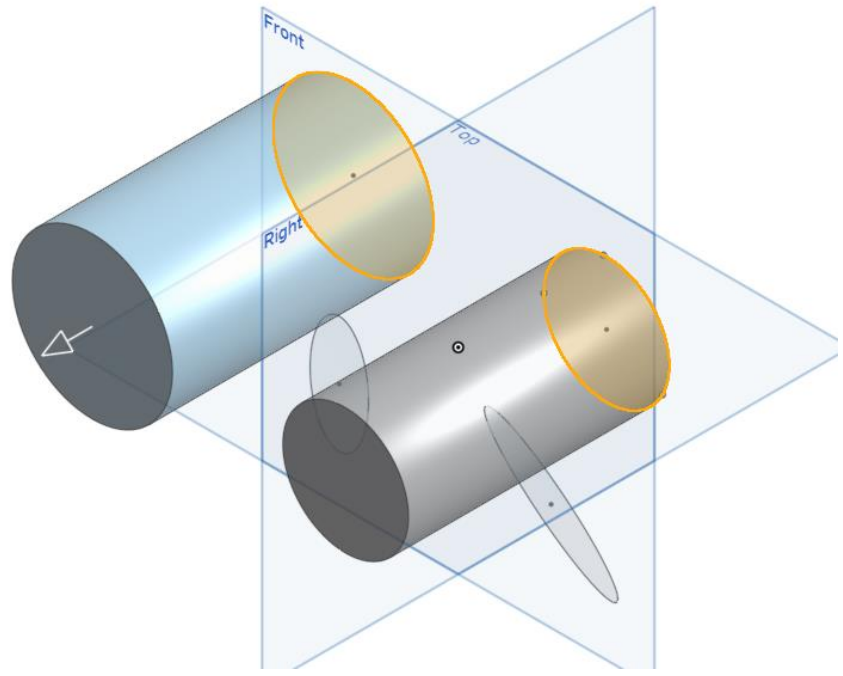
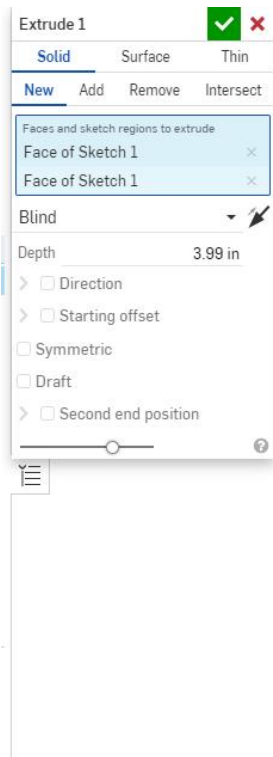


Left click on the first circle and it will extrude 1 inch by default. You can type in the distance or drag the arrow to make the extrude bigger.



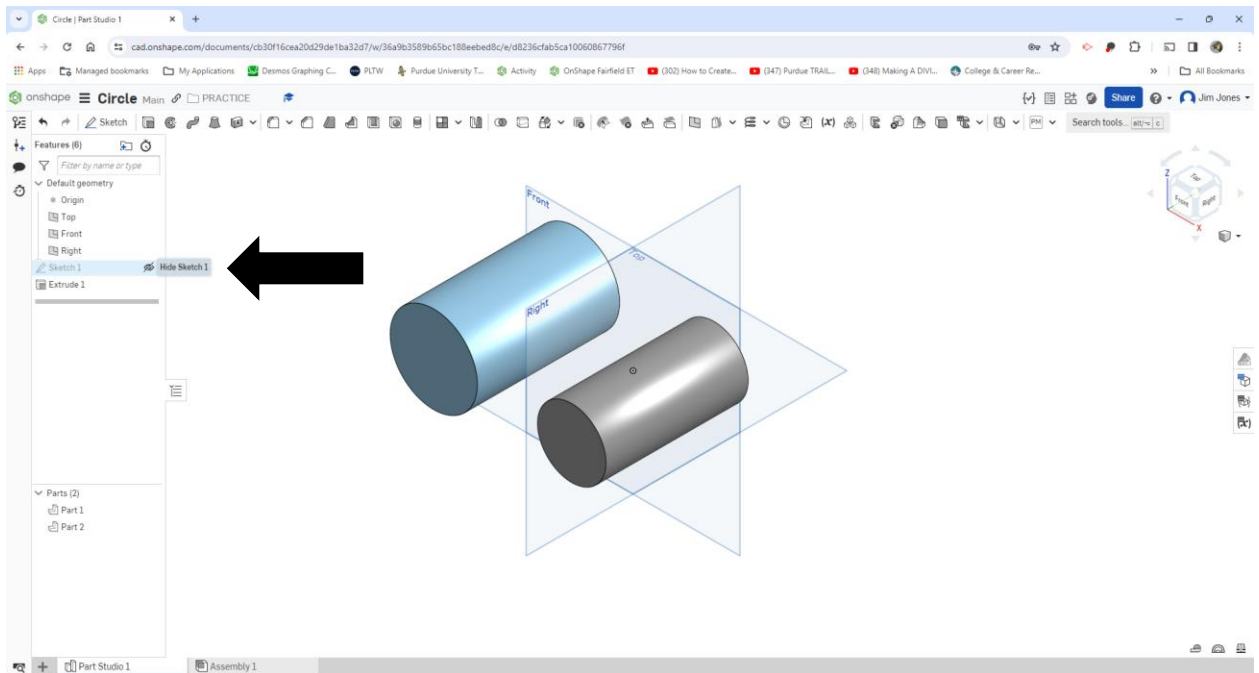


The number distance changes if you use the arrow. You can extrude another circle and it will match the first extrude.

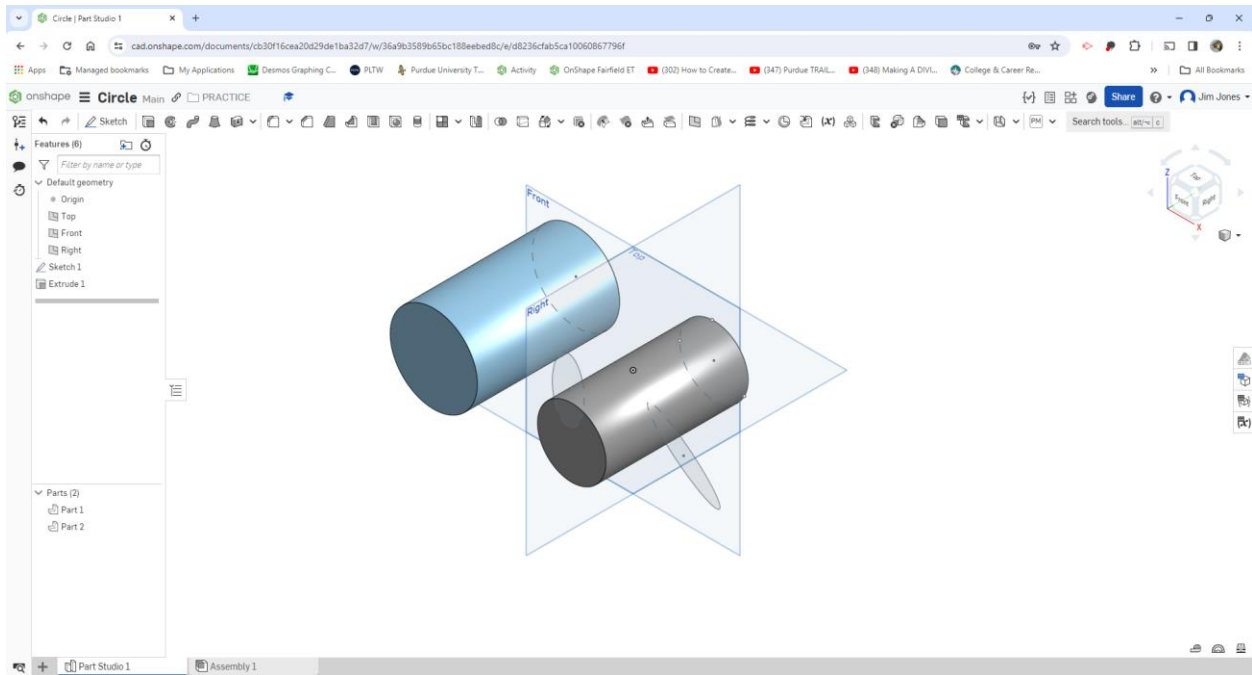


Click the green check mark to stop this extrude and then you can extrude the other shapes with an additional extrude command.

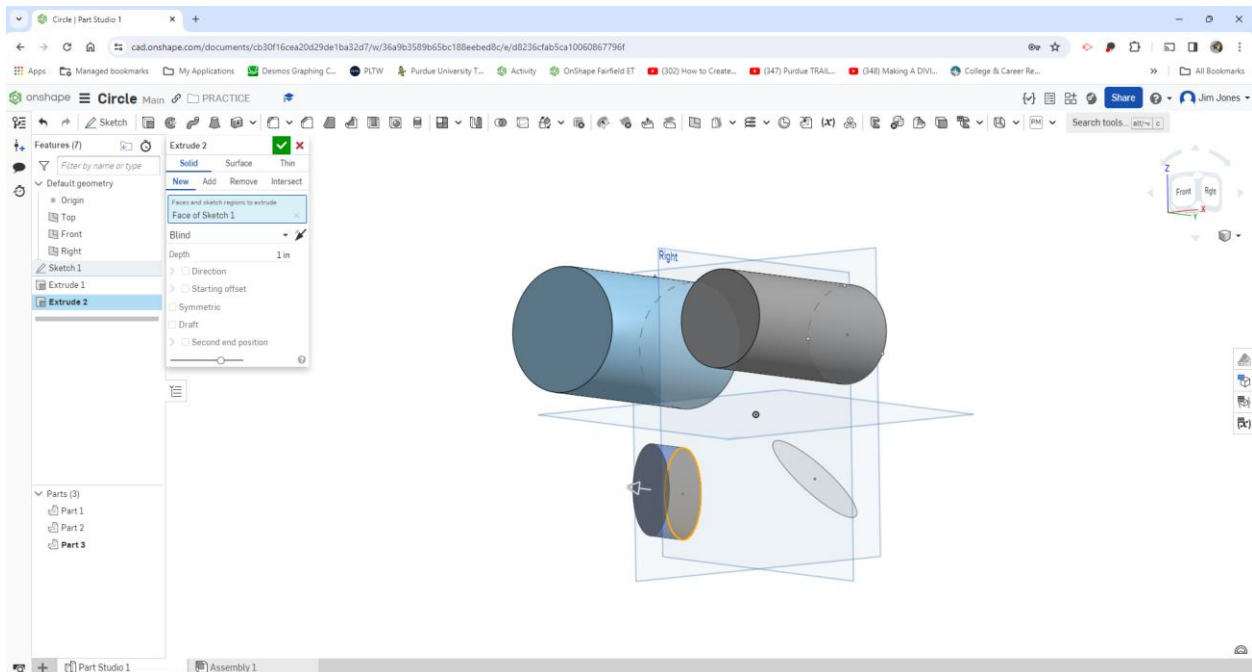
Left click on Sketch one and left click on the eye to turn on the additional sketches of the circles and ellipses.



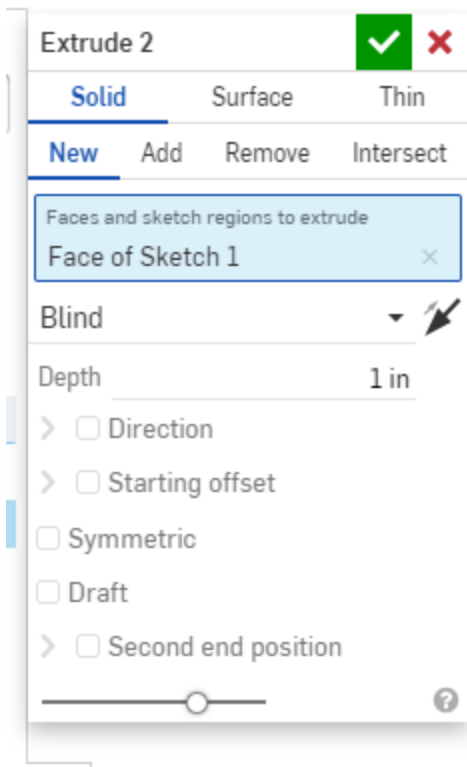
Clicking on the eye will turn on the other sketches.



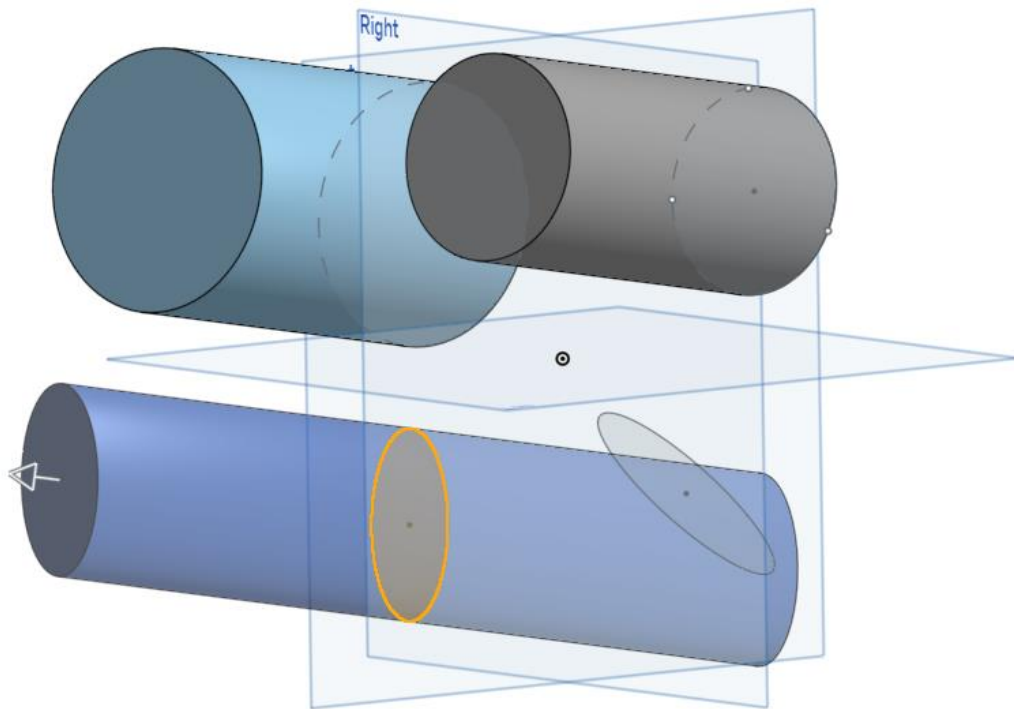
Hold down the right mouse button and you can rotate the drawing screen to make it easier to click on the other circles or ellipses. Click extrude and select another circle or ellipse.

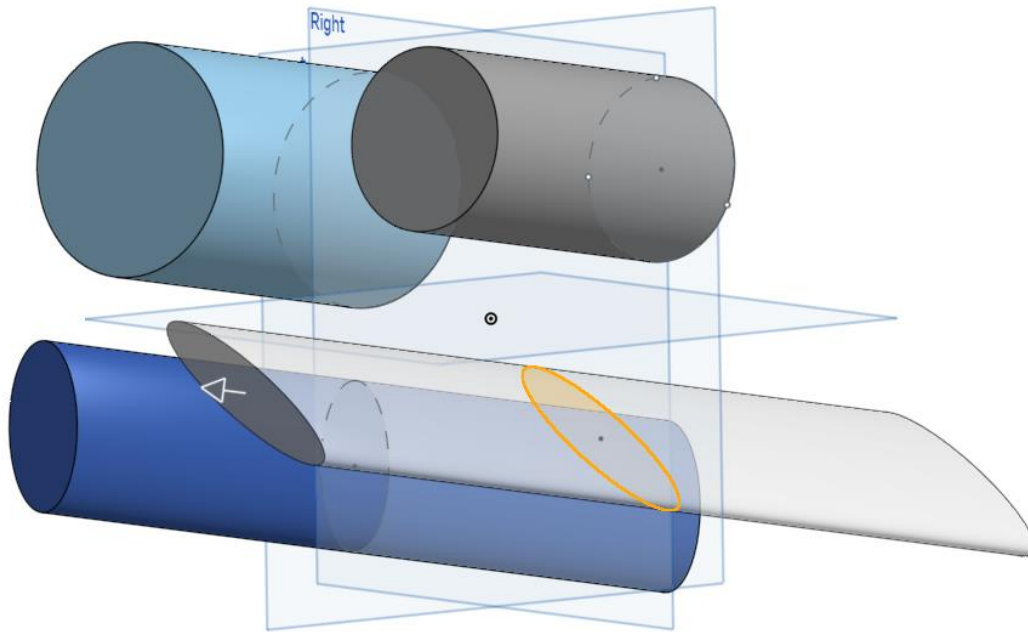


The extrude box has additional options to control the direction of the extrude or make it symmetrical.



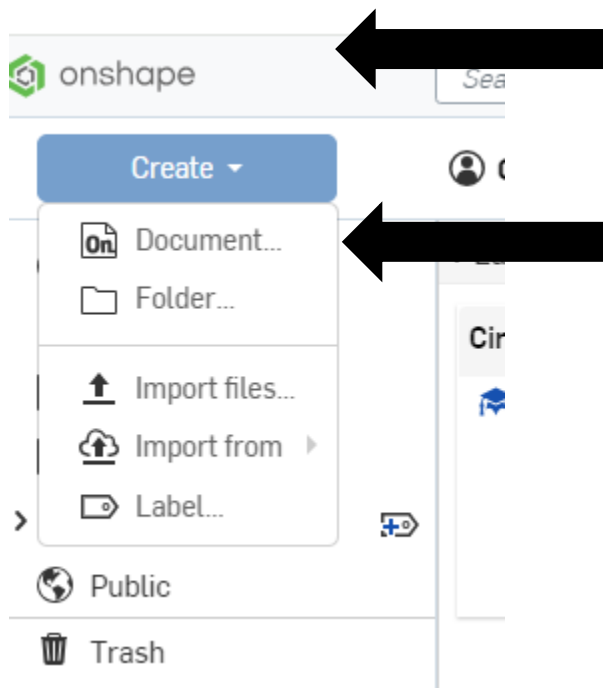
If you click Symmetric the extrude will go both directions from the sketch. Practice this and the other options in the box on your objects.

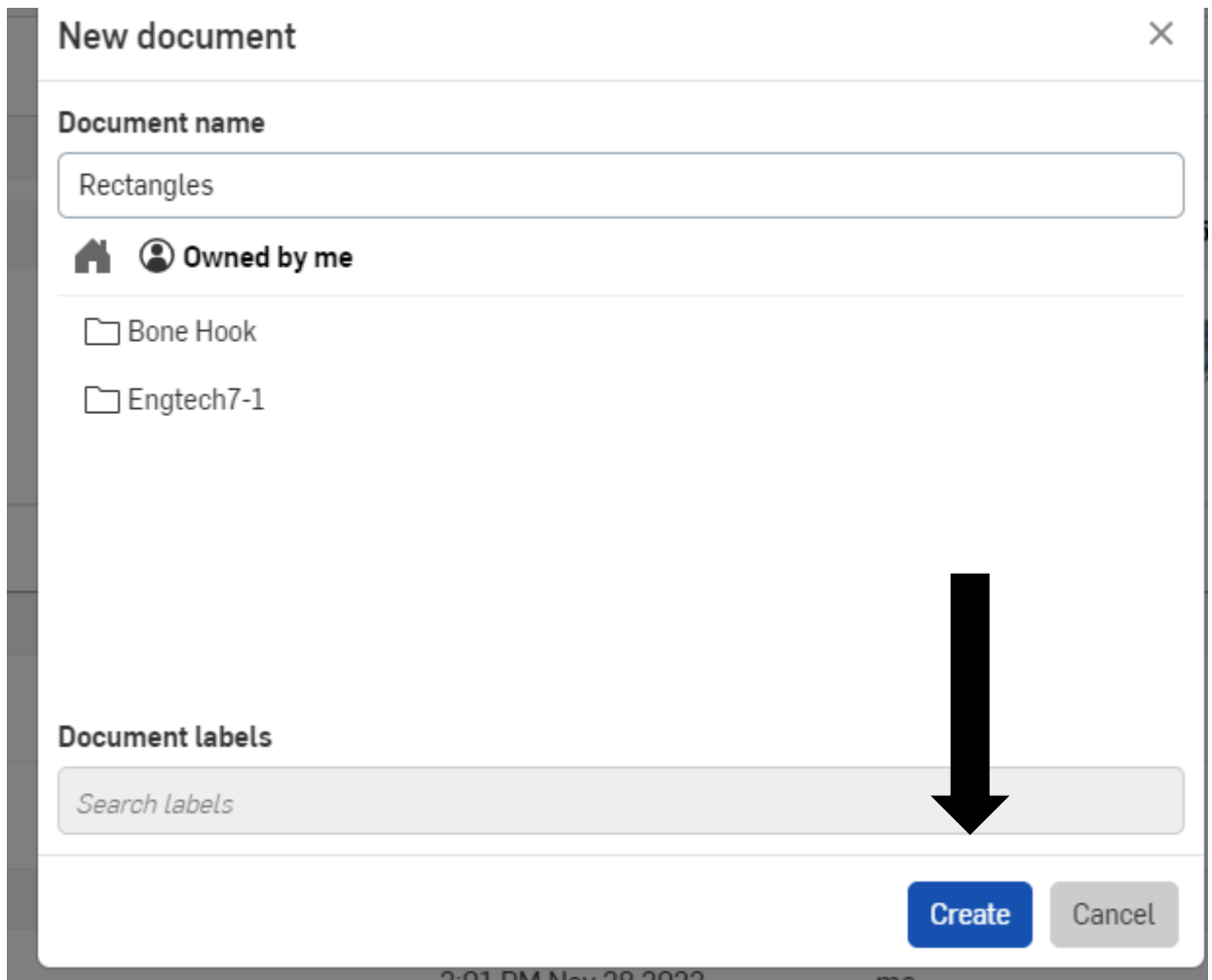




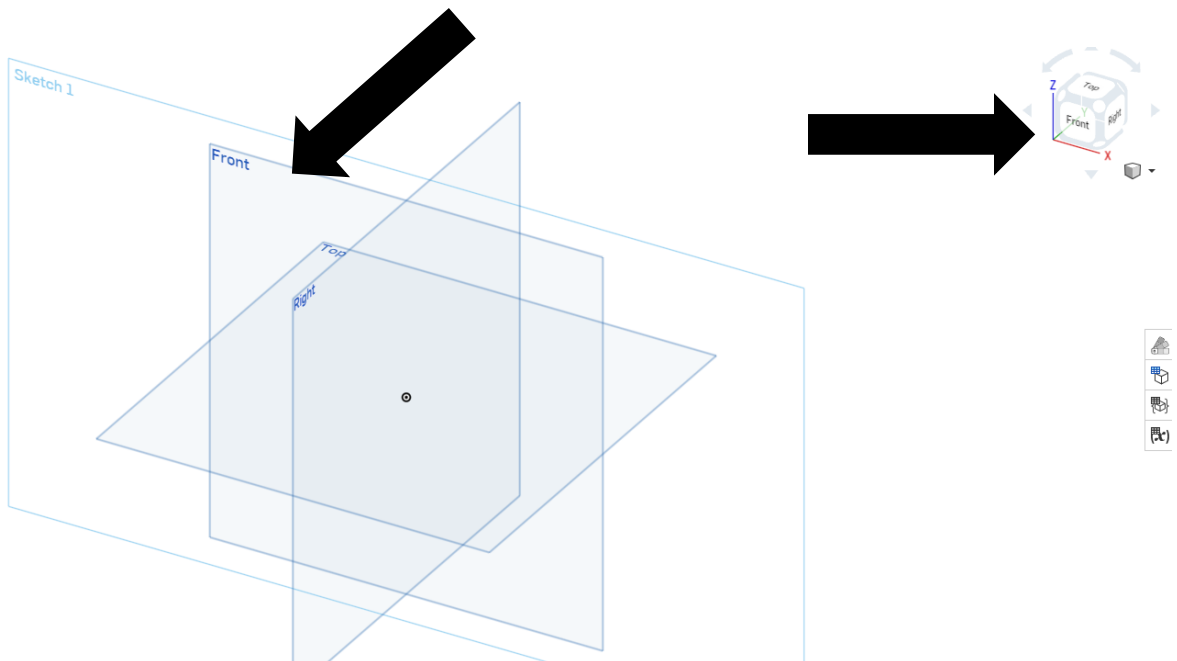
There are many what to extrude so you will need to practice all the options until you are confident in controlling the process. Click the green checkmark to finish the extrudes.

Left click on the Onshape icon in the top left corner to create a new document and name it retangles. Then click on the Create button.

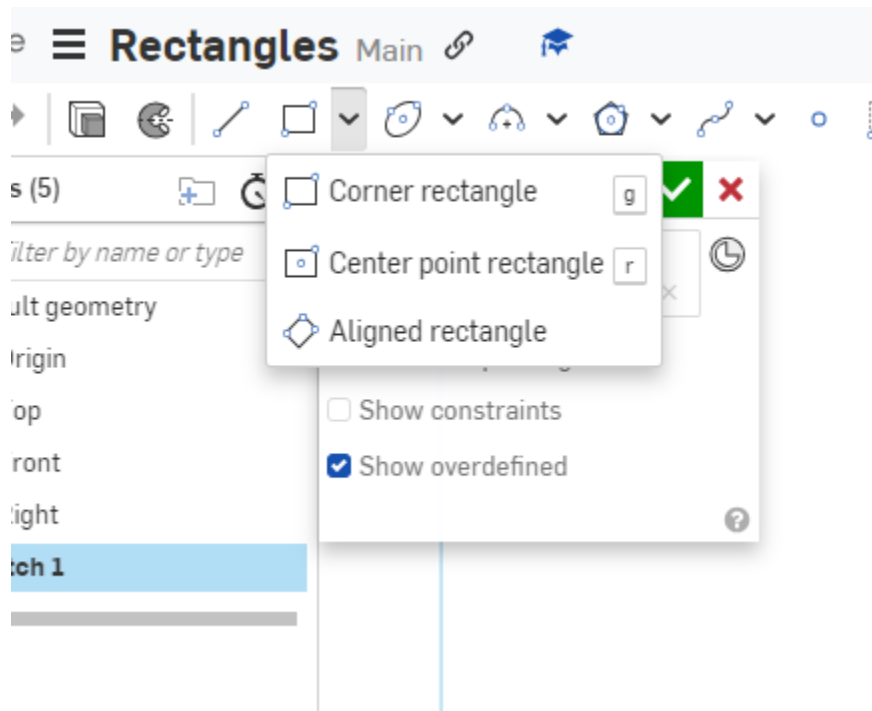




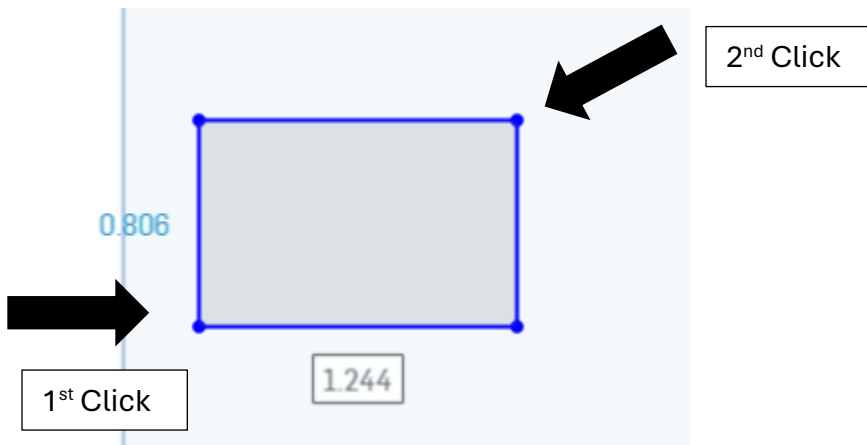
A new drawing screen will show up and now left click on Sketch in the ribbon to start the rectangles. Left click on the Front plane and then left click on the Front button on the view cube.



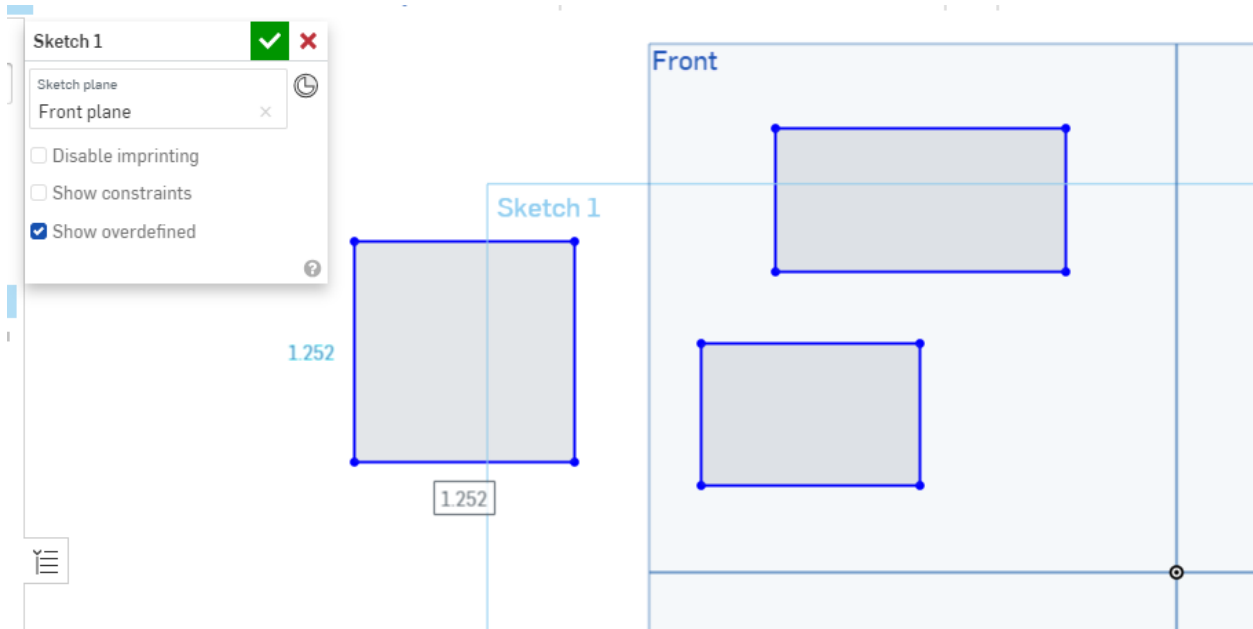
Left click on the Rectangle drop down to see the three ways to make rectangles.



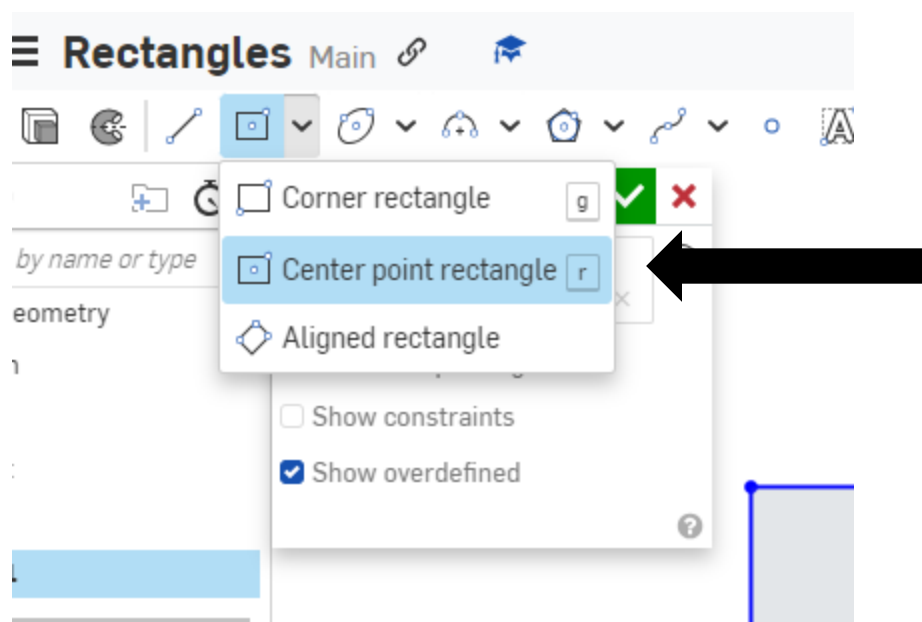
Left click on the Corner rectangle to make rectangles by clicking on the two opposite corners.



Make a couple more rectangles in the top left area of the workspace.

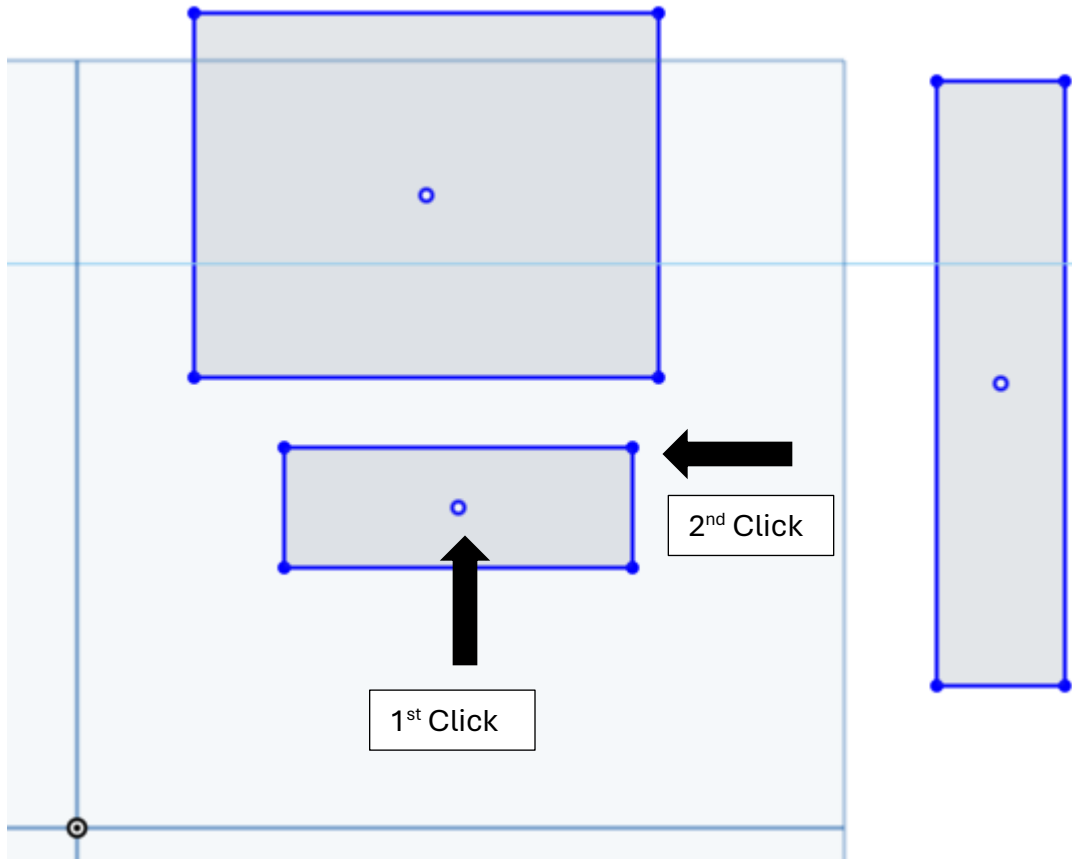


Left click to escape making rectangles. Select rectangles and then click on Center point rectangle.

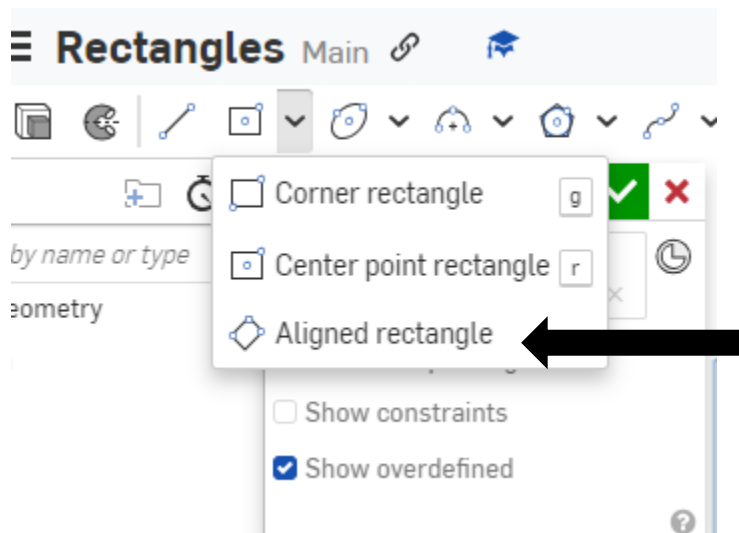


Left click in the top right section to start a new rectangle, the second click will be a corner. Make a couple of center point rectangles and then right click to escape making rectangles. Notice these rectangles will only go horizontally or vertically.

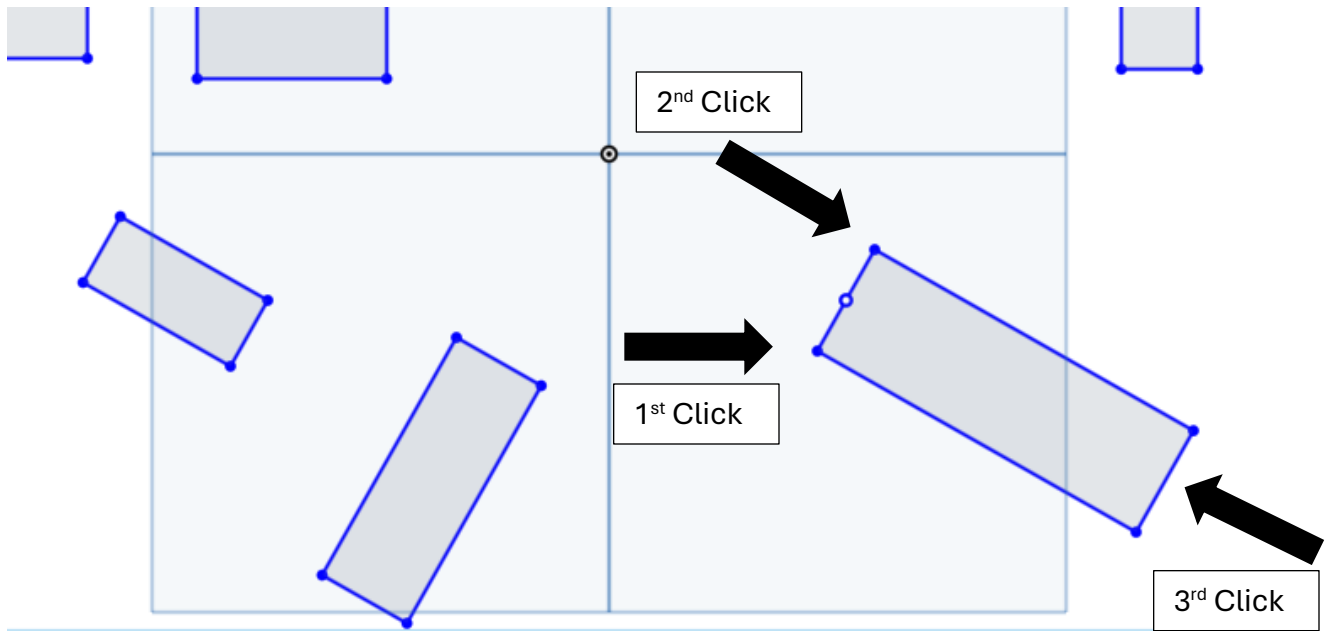




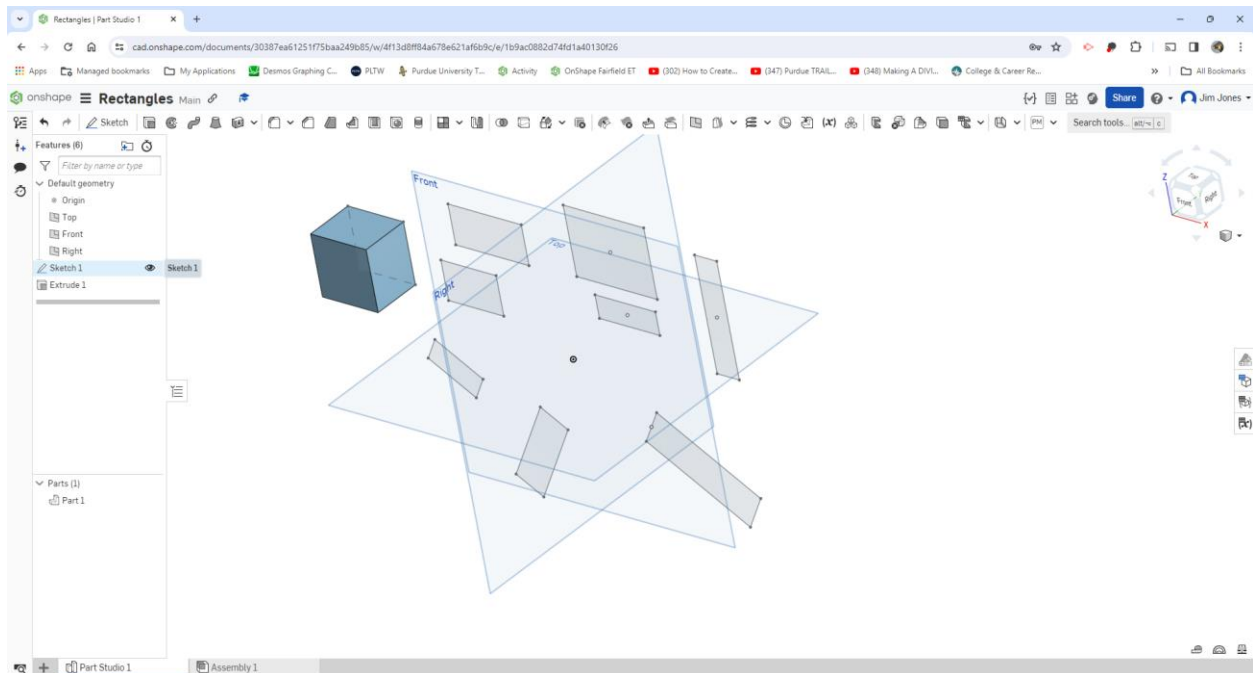
Left click on the Rectangle drop down and left click on Aligned rectangle. Now draw a couple of rectangles on the bottom of the workspace.



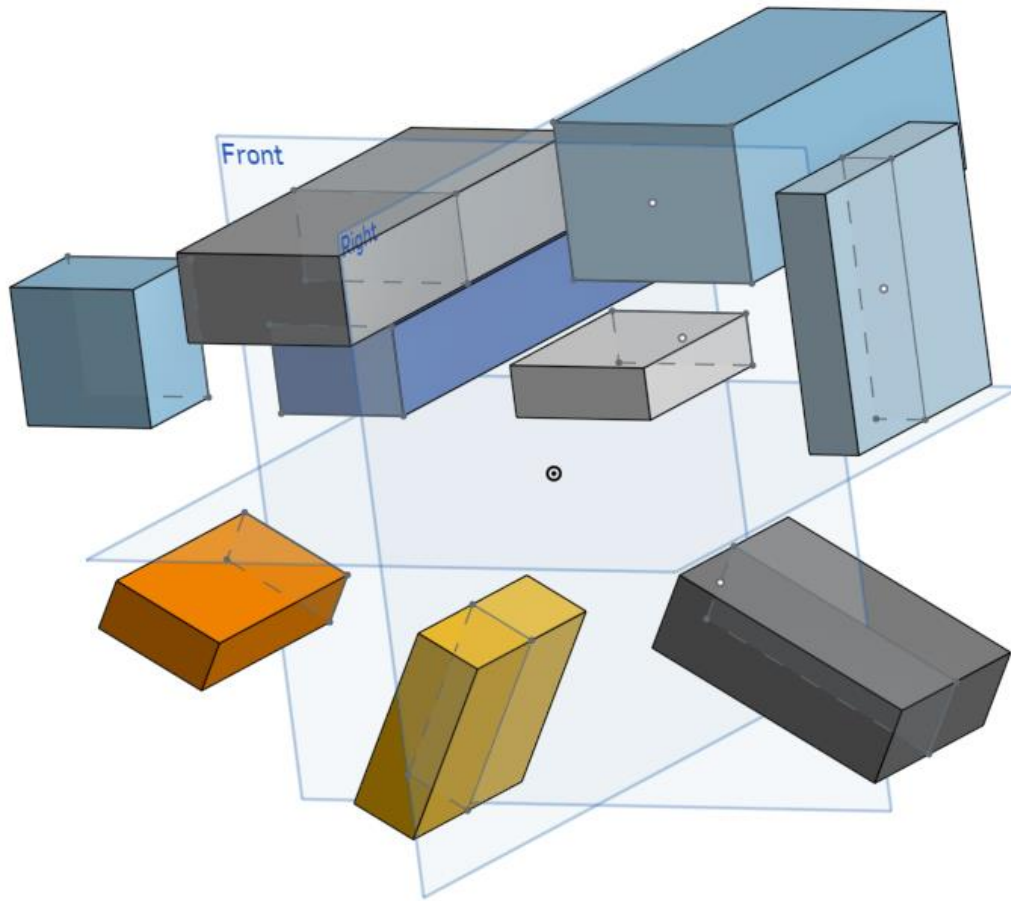
These rectangles take three click with the first two clicks setting an endline and angle for the rectangle. The third click will set the distance for the rectangle.



Left click on the green check mark to finish the sketch. Now click on the extrude icon to extrude each rectangle. Practice making each one a different length and or direction. Use the right mouse button to rotate the screen to make the extrudes easier to see. After the first extrude you will need to left click on sketch one in the browser to turn on the eye so you can see all the sketches.



This is just practice so if you make a mistake it is not a problem. Just try and do your best.



Further instructions will follow on separate documents. Including Polygons, lines, arcs, text, splines and then additional instructions on 3D modeling using Revolve, Loft and Sweep with additional tips on Fillets and Chamfers.