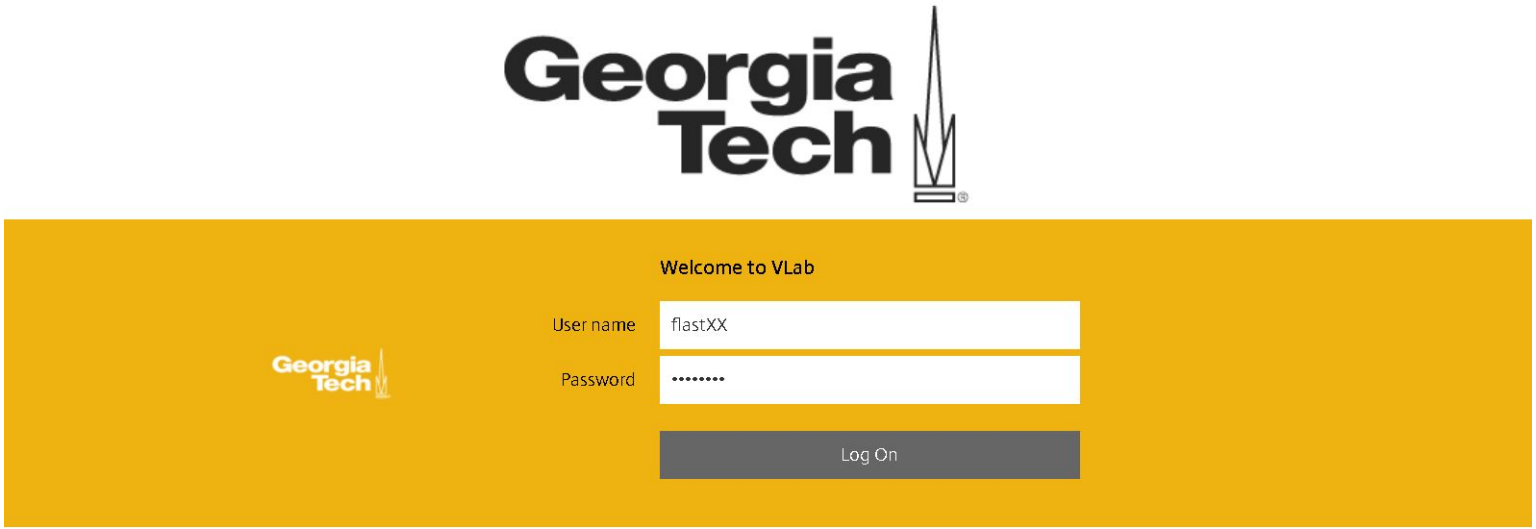
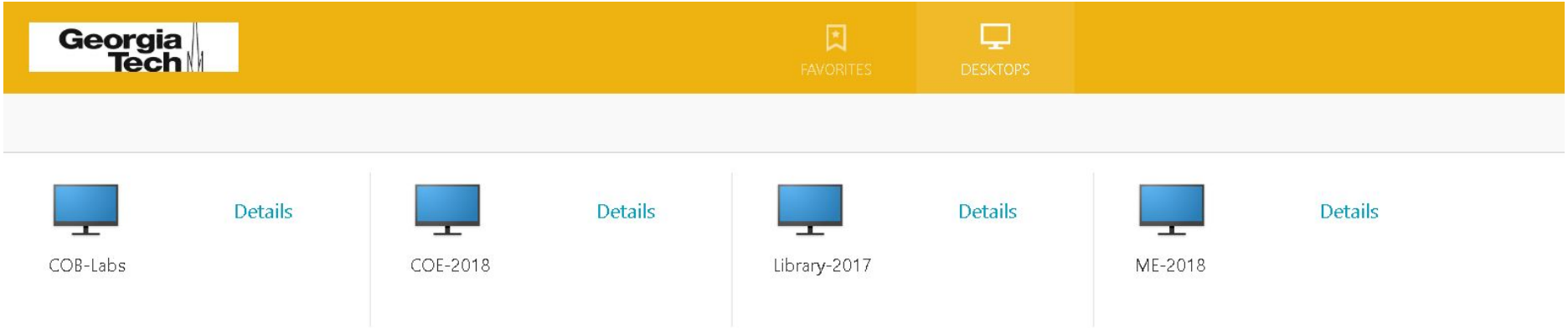
Autodesk Inventor

- Launching Inventor
  - VLAB
  - Local
- Introduction
- Basic use

Using VLAB



Log on to mycloud.gatech.edu



Select “COE-2019” Desktop and launch Inventor

Installation

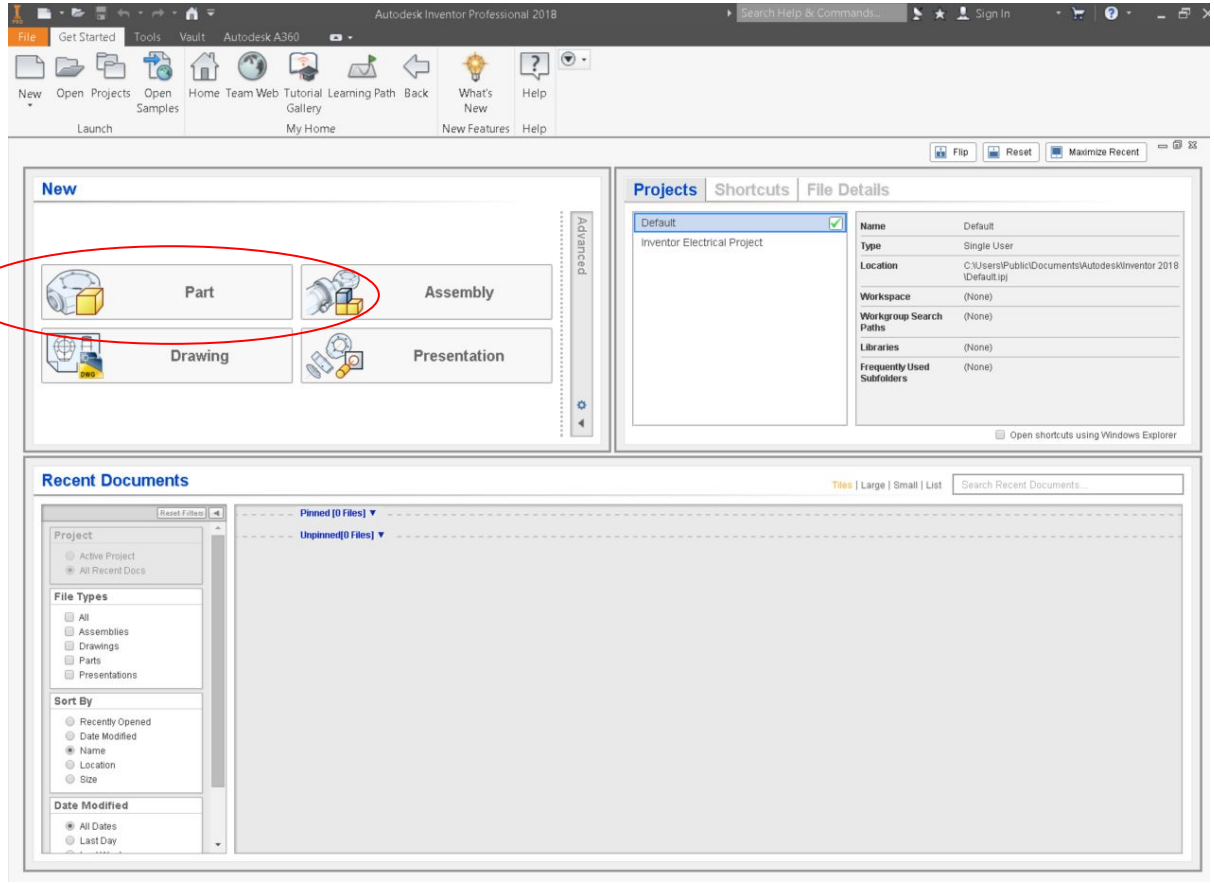
Installing Inventor

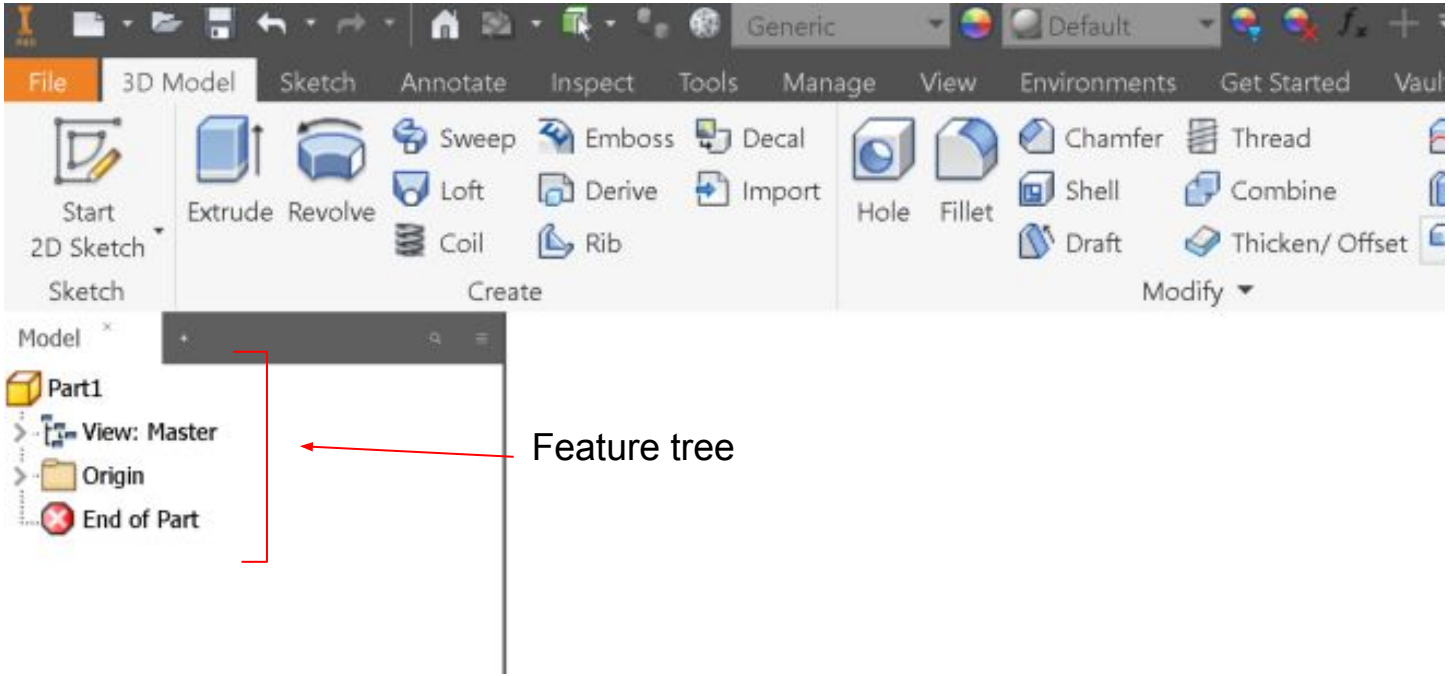
- Visit https://www.autodesk.com/education/free-software/inventor-professional
- Create an account (can be your personal email, doesn't have to be .edu)
- Download educational license per your system
  - We'll be working out of Inventor 2019
- Allow several hours to install

For Help: contact #it-helpdesk on RoboJackets Slack

Introduction to Inventor

- Default launch screen
- For now, we'll be working out of the Part environment, which can be opened by clicking Part

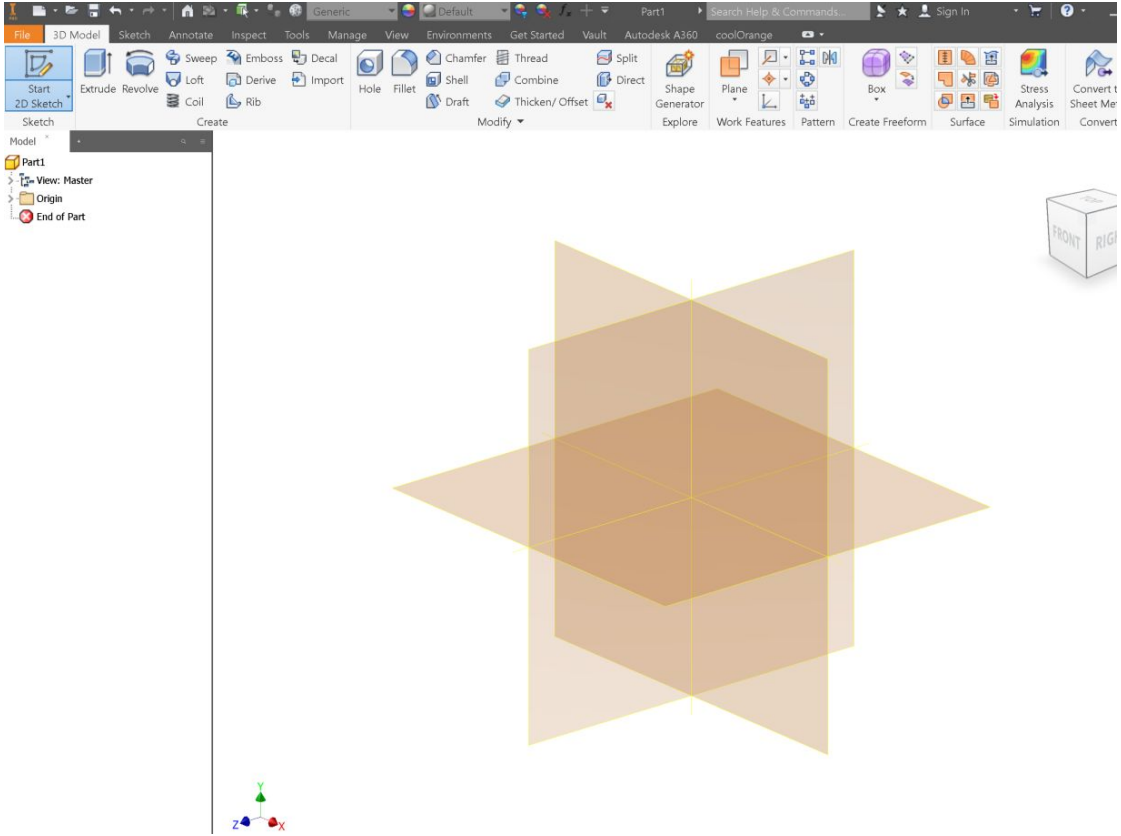


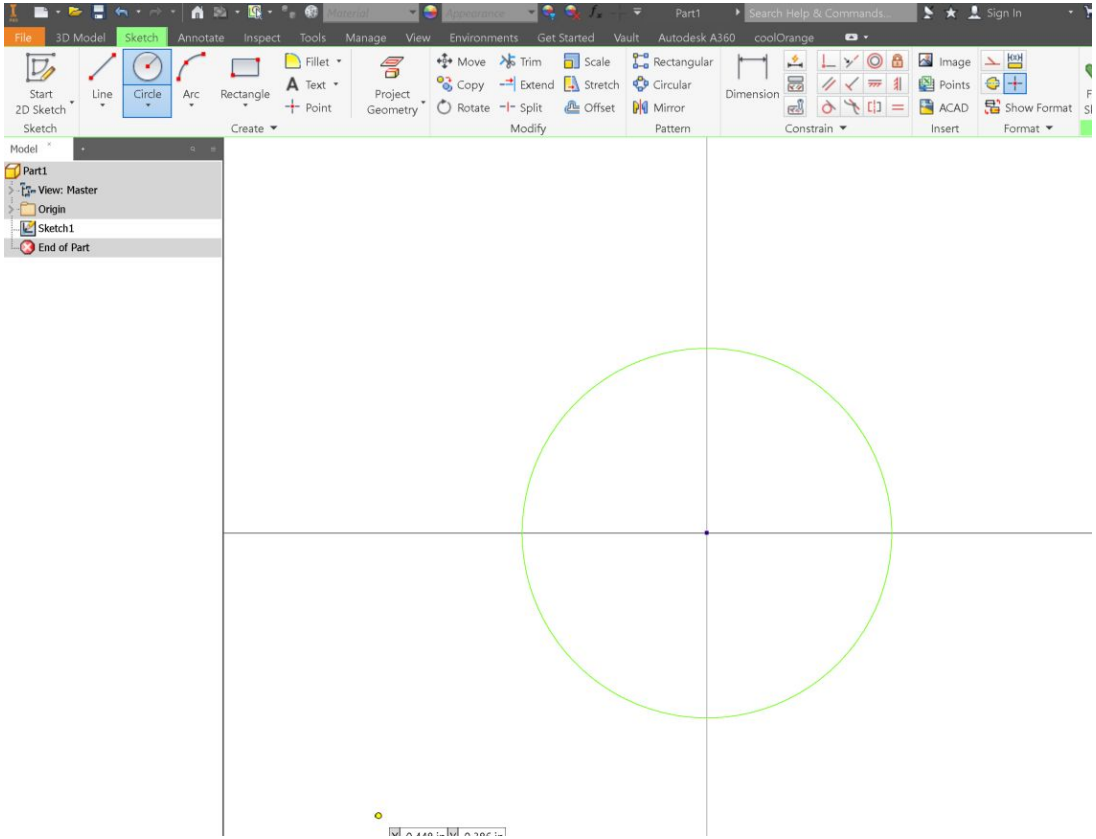


The feature tree displays created objects, allowing easier manipulation and management

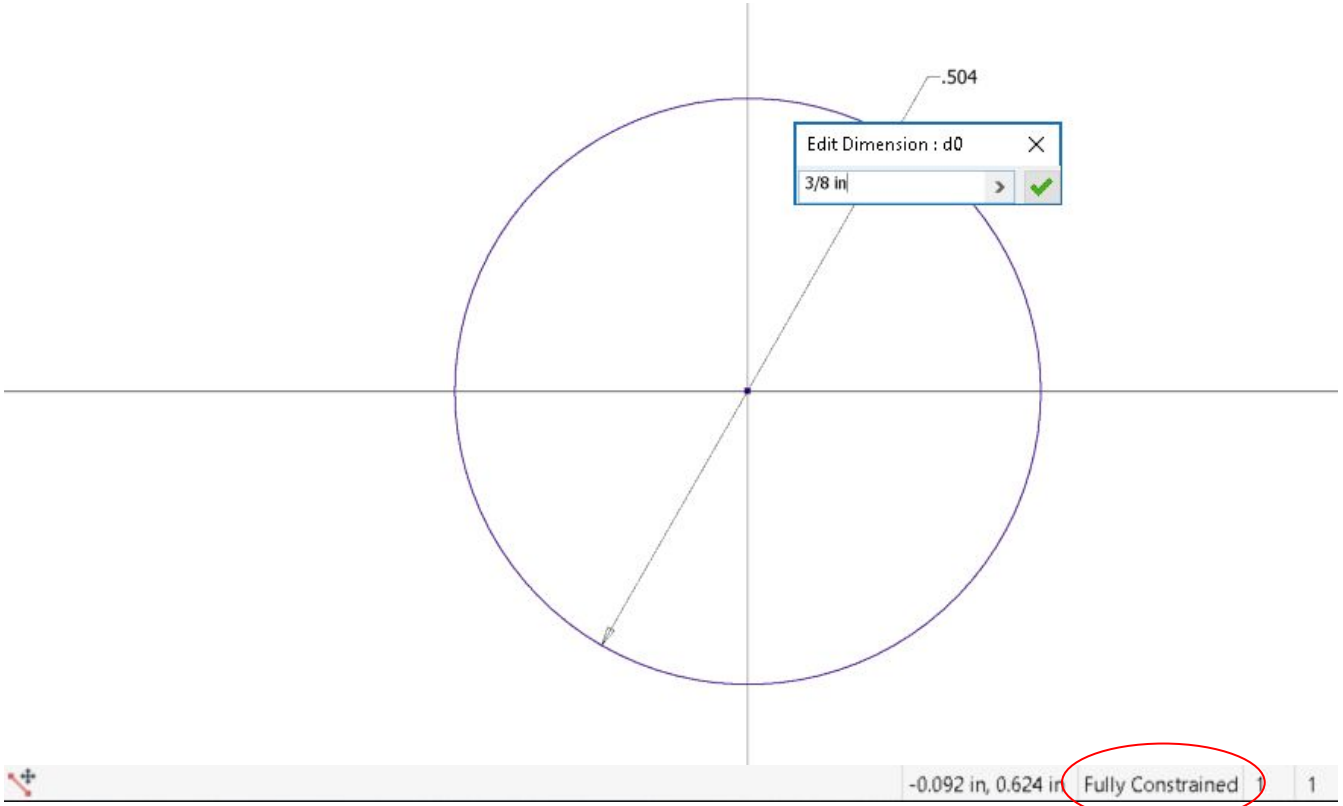
Starting a Sketch

- Select Start 2D Sketch
- Select a plane to begin sketch (XY Plane used in example)





Draw a circle by selecting the circle tool



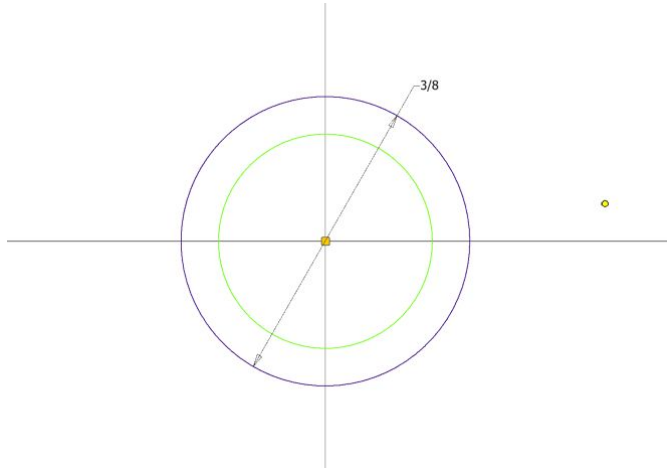
Select dimension tool and click circle to add dimension.
Set the diameter to ⅜ in

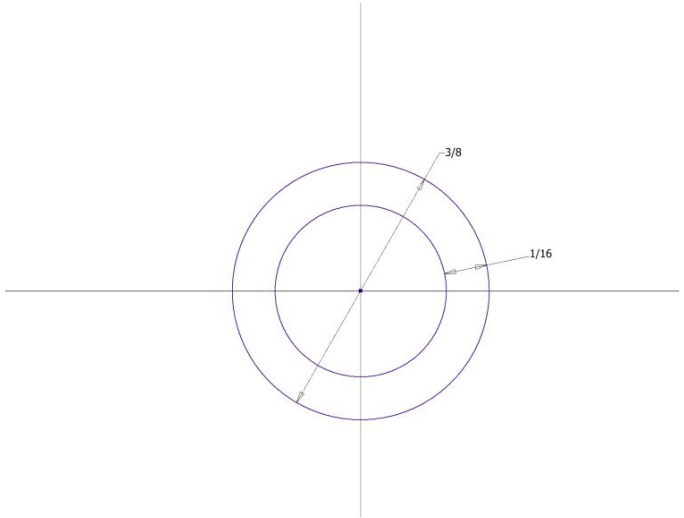
The circle turns blue when properly dimensioned (fully constrained)

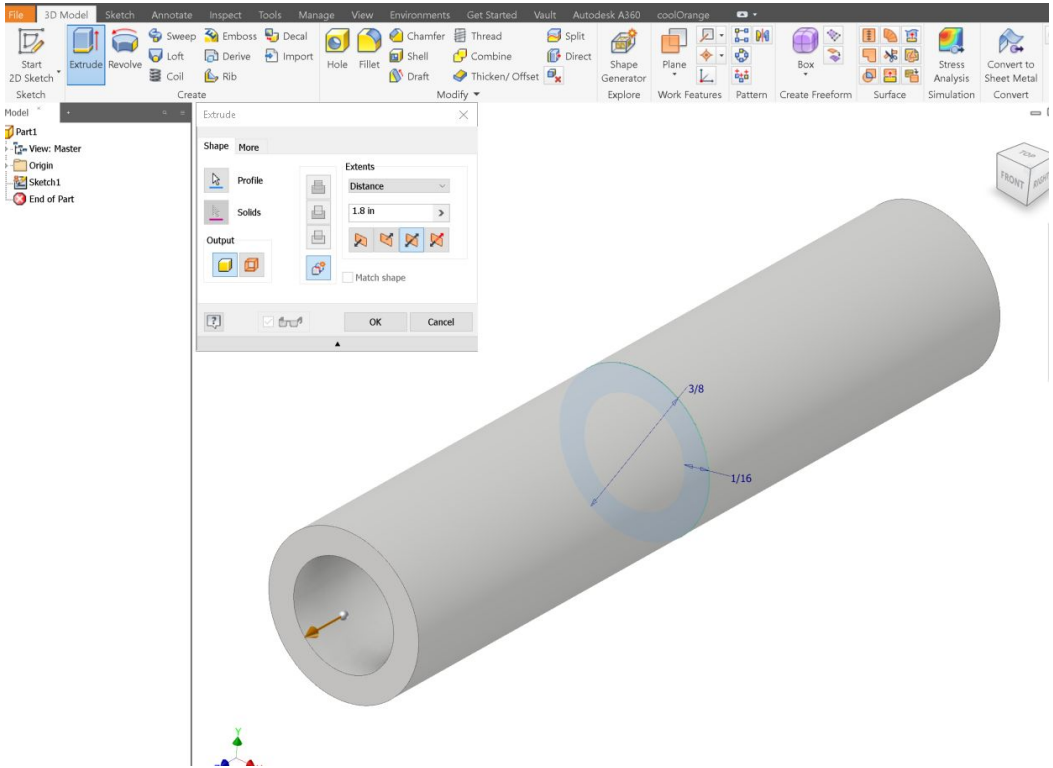
- Inventor must have all information to completely describe an object
  - Note how Fully Constrained is displayed in the status bar (see previous slide)
- Things that are green are not fully constrained
- Do not use “Auto-Constrain” or “Fix”
  - Fix - makes object’s current position permanent
  - Auto-Constrain - makes some assumptions for what it thinks you want and fixes everything else

Create another circle within the previous circle

- Dimension this circle as an “offset” by clicking the larger circle followed by the smaller circle
- Enter an offset distance of 1/16 in

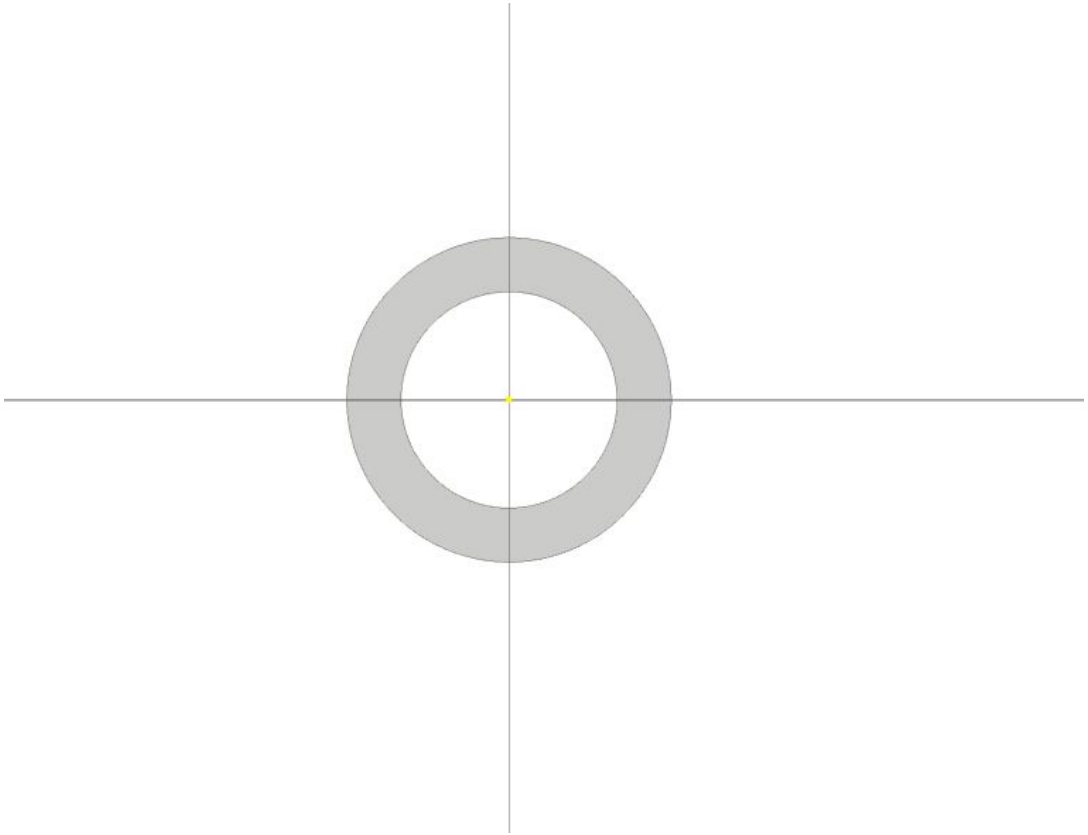






Select *Finish Sketch* and click *Extrude* under the create palette. In the dialog, enter 1.8 in

- Sometimes Inventor will use your last sketch and make a coherent extrusion. A sketch must have an enclosed area for extrusions.

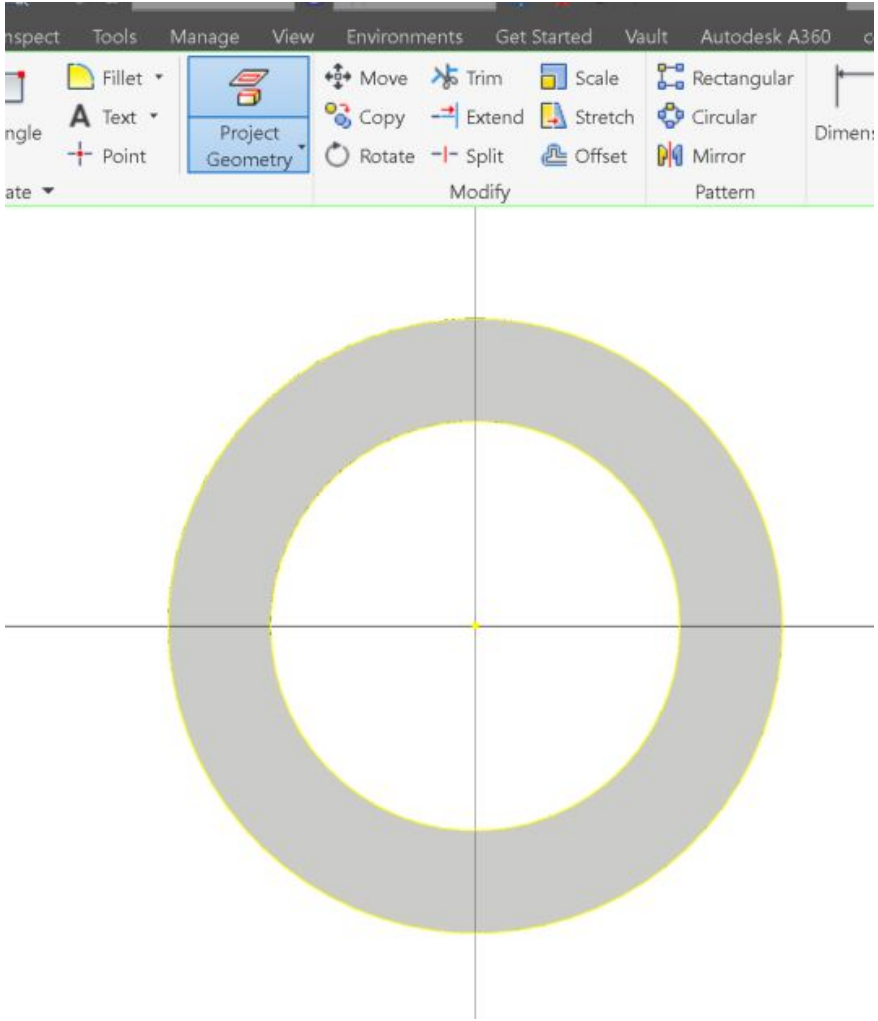


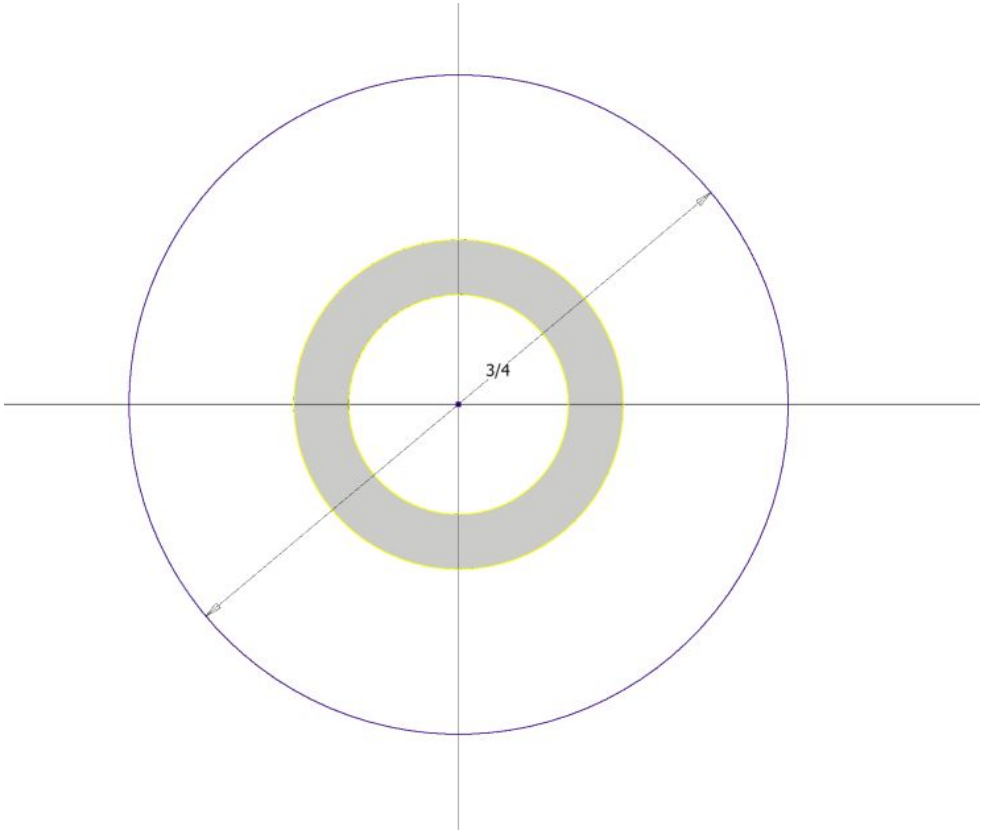
Start a 2D Sketch once again, but this time, click on an end face of the extrusion

- You can start sketches on any plane, including flat faces on bodies

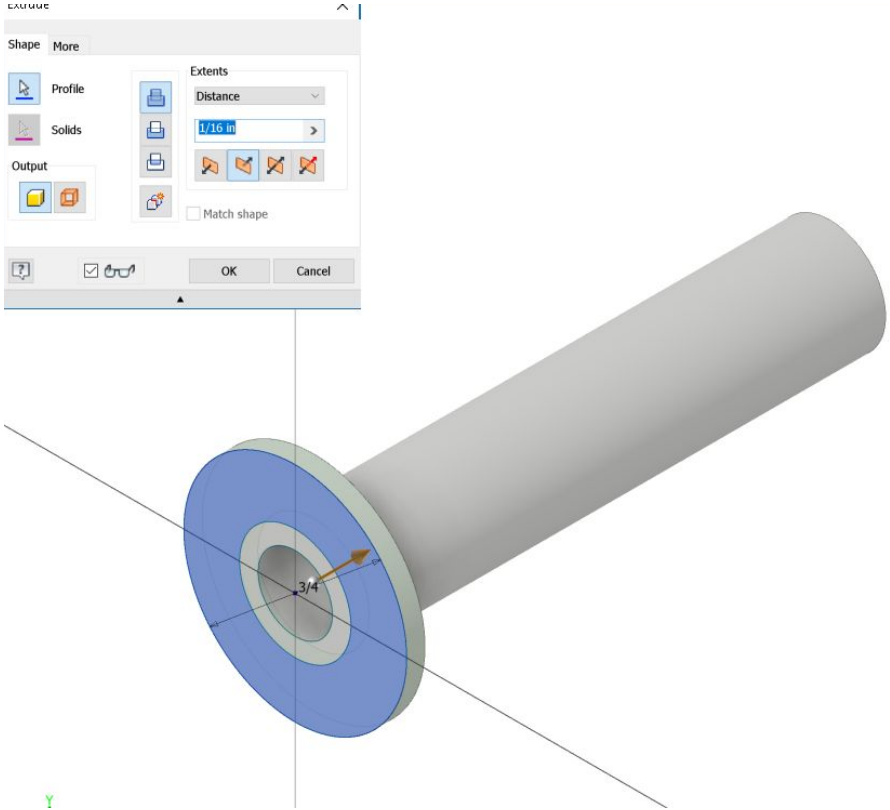
Select project geometry and click the outer edge of the face.

- This allows you to access entities outside the sketch’s scope

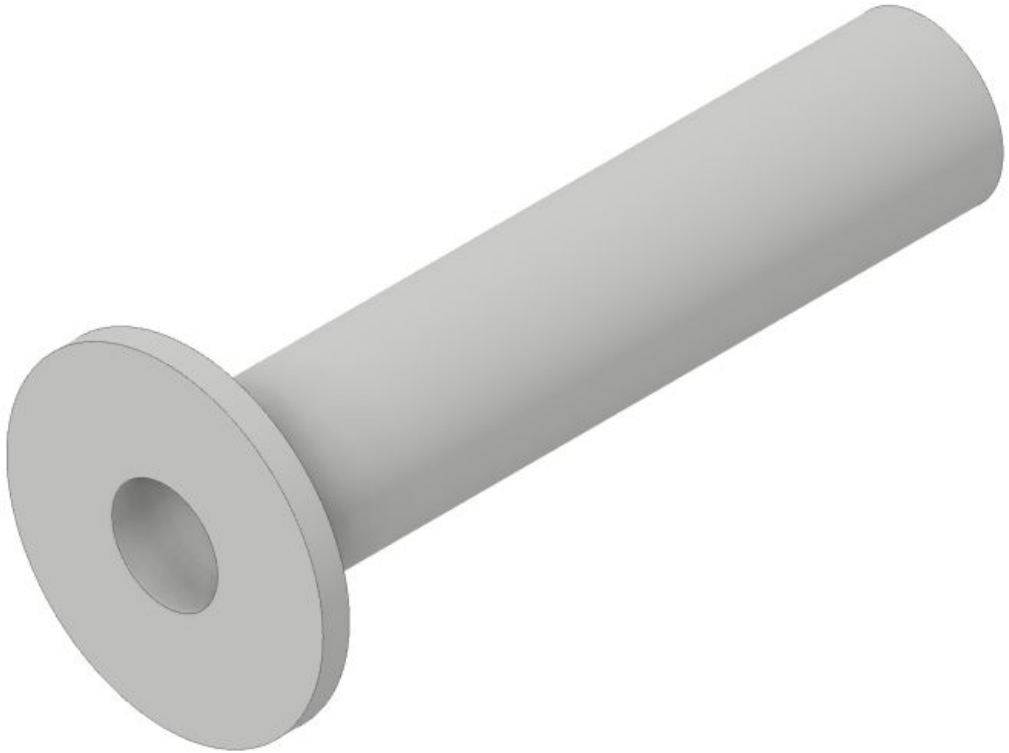




Create a new circle with ¾ inch diameter.



Finish the Sketch and Extrude the outer profile. (Check that your dialog settings match the example)

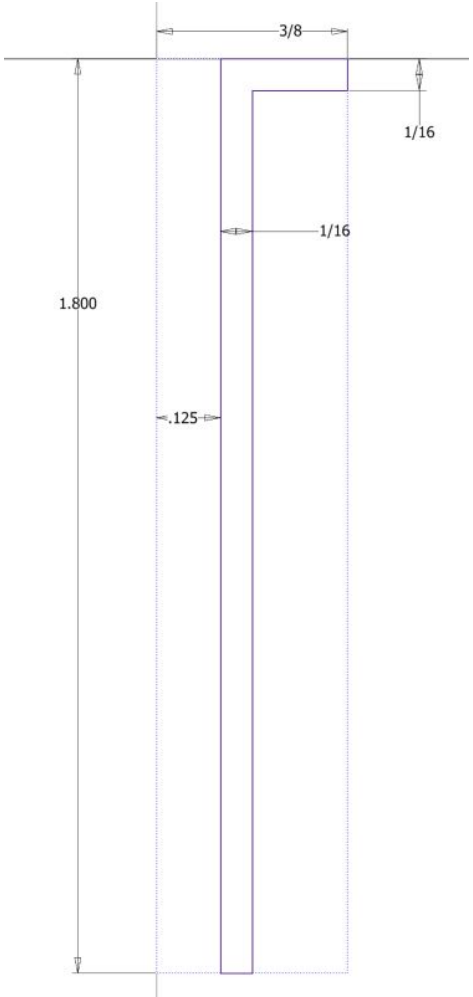


Great job! You've made a simplified rivet. Now, let's make it another way...

Create a new part, and draw a new sketch according to the picture.

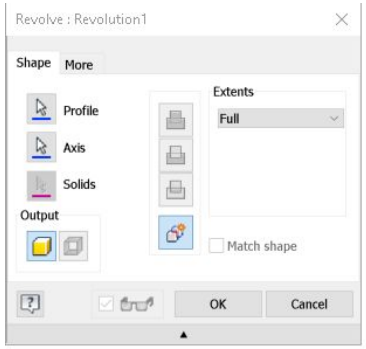
- The dashed line is a construction line (these will be elaborated on in a later session)
  - Any sketch entity can be converted into a construction line by clicking the construction button

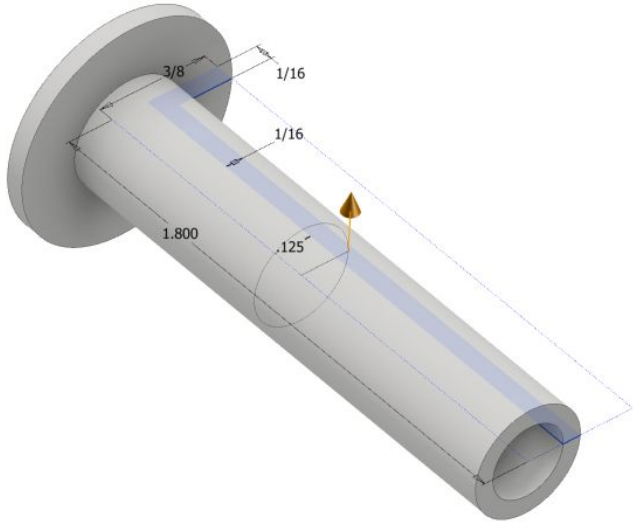
- For this example, draw a 1.8” x .375” rectangle and after selecting the rectangle, click the construction button
- Inventor will automatically snap to horizontal and vertical lines if you let it do so

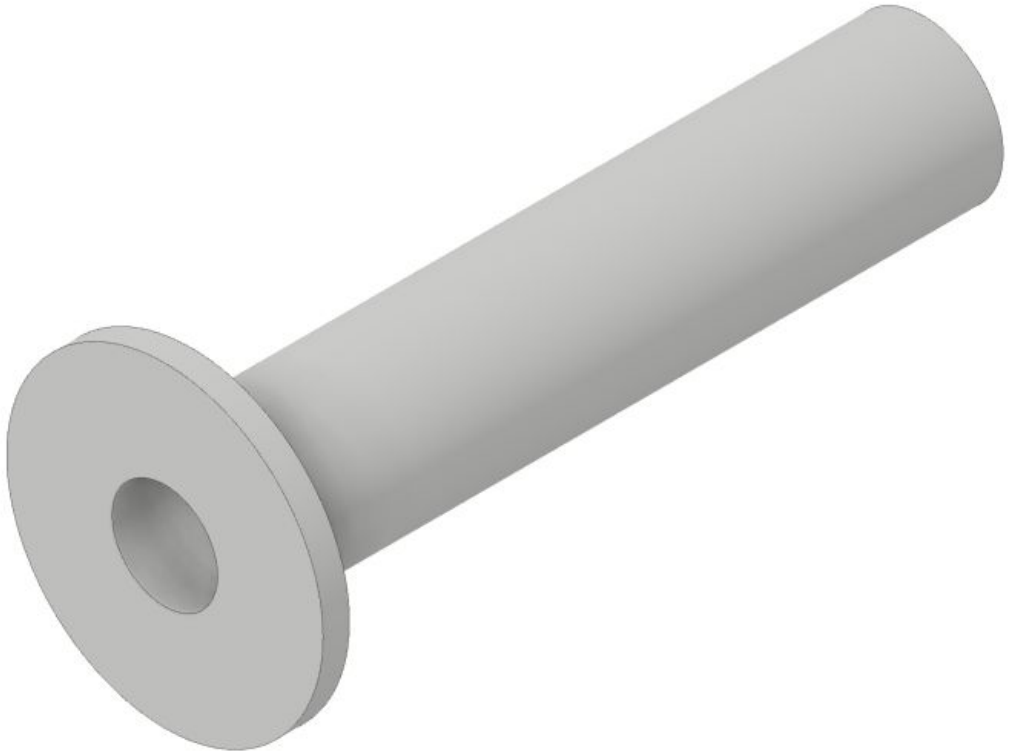


Finish the sketch, and select *Revolve* (Next to *Extrude*).

- You will be prompted to select a profile to **revolve** around an axis, creating an axially symmetric part
- Select the profile shown in blue in the picture and use the white line as your axis







Surprise! It’s the same part, created in two steps.

- This is why it’s important to think about your design beforehand