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Autodesk Fusion 360

Tutorials

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DESIGN FACTORY

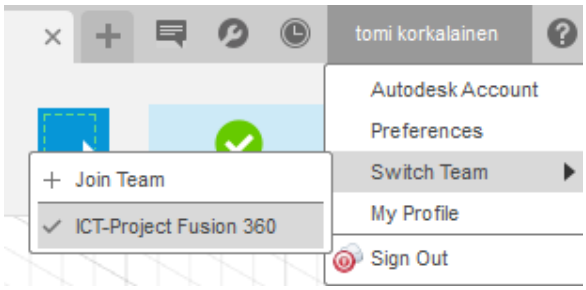
GENERAL INFORMATION

OVERVIEW

This document unfolds the basic functions of Autodesk Fusion 360 and goes through three small tutorials. They will include the essential tools for beginners for them to have a quick start on it.

PROJECT MANAGEMENT

Fusion Team is a centralized source for all your project collaboration. Projects and designs are always created within a team. Before you can start modelling, you need to create a team if you have not already. Active team can be changed from the top right corner.

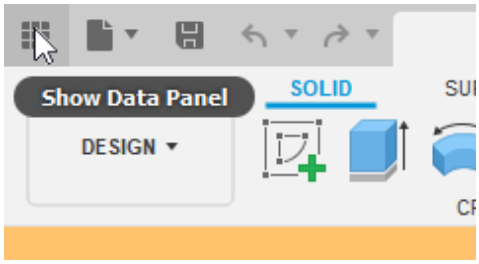


Team can be switched from the top right corner

Projects can and often will consist of multiple designs. Projects are the intended way of keeping the related designs organized in the same place. You can share projects with other Fusion 360 users by being in the same team and giving the other person privileges to the project.

DATA PANEL

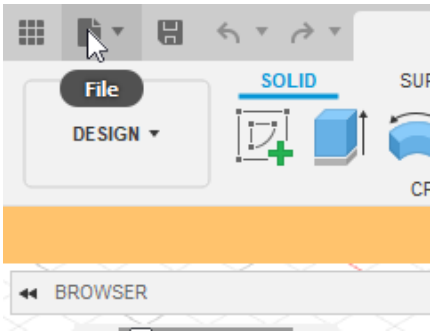
Open the Data Panel to see your team's projects, create new and open existing ones. The Data Panel button can be found from the upper left corner of the program. The active team can be switched under your username from the upper right corner. Preferences are also found here.



Data Panel button location

FILE FORMAT

Export and other basic functions can be found under the file menu. File Menu button can be found next to the Data Panel button. For the design to be compatible with most of the 3D printing software you should export the file as STL-format.



File button location

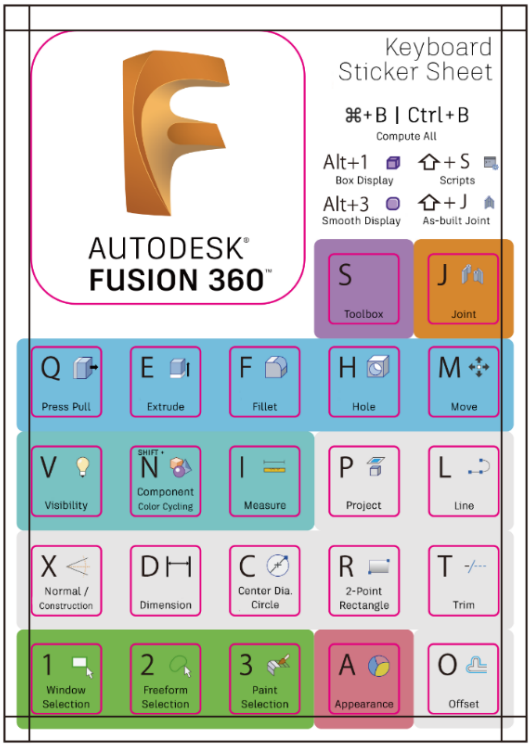
TIMELINE

You can move the timeline marker to return to different points of the project. Each step is shown in the timeline and can be distinguished by the tool icons. The steps can be edited from the timeline by right clicking the step and clicking *Edit Feature*. Pressing the play button showcases every step of the project like a slideshow.

Timeline / History

HOTKEYS

Some tools can be hard to find from the menus, so learning the hotkeys for the basic tools can be useful.



Keyboard sticker sheet

CUP – TUTORIAL

OVERVIEW

This tutorial contains instructions how to 3D-model a simple cup.

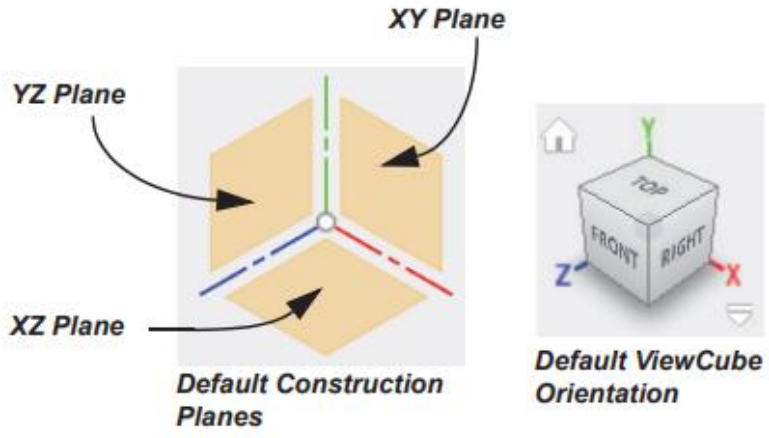
Tools included in this tutorial:

- Plane
- Sketch
- Extrude
- Fillet
- Shell

3D modelled cup

SKETCH

Before making a sketch to the project, a plane must be selected. Planes can be thought as working surfaces. Created sketches will be placed on the selected plane, so they will be on the same face. At start of each design you are given 3 planes to work with.



Plane Selection and the View cube

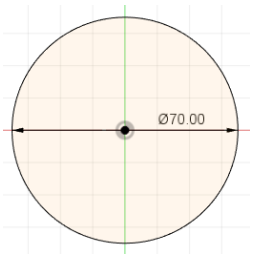
Click on the *Create Sketch*  **button**. The button can be found in the toolbar at the top of the view, in the *solid* tab.

Now, select the *XZ Plane* *by clicking it.* If you find any trouble selecting the correct *plane*, press the *top* face of the *View Cube*. The *View Cube* is located on the upper right corner of the view.

After pressing the top face, you should only see the *XZ Plane.* After the plane selection, *sketch* tab and *sketch palette* will appear on the toolbar.

Select *Center Diameter Circle*  **tool**. The tool can be found in the toolbar at the top of the view, in the *sketch* tab. With this tool, a circle can be drawn by placing two points, one on the center and one on the circumference.

Click to place the center point and **specify diameter of the circle** with mouse or by entering a value. After entering the value, **press *Finish Sketch*** from the *Sketch Palette* menu. The circle sketch has now been finished.

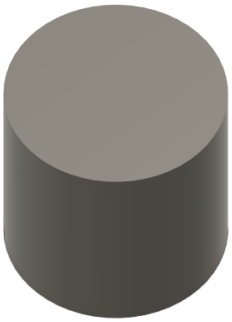


Finished Circle Sketch

EXTRUDE

After creating the *sketch*, it can be extruded into a 3D object. **Select the *Extrude***  **tool** to create a cylinder out of the circle *sketch.* The tool can be found in the toolbar at the top of the view, in the *solid* tab.

C*lick on the circle sketch that was drawn during the previous step**. To adjust the extrude distance, **drag the blue arrow or enter a value** to the *distance* field in the extrude menu. **Then press *OK from the *Extrude menu.*



Circle extruded 70mm

FILLET

Select the *Fillet*  **tool**. The tool can be found in the toolbar at the top of the view, in the *solid* tab. When the tool is activated, **select the bottom edge of the body**. Any edges will be highlighted when the mouse hovers over it. After an edge is selected, **drag the blue arrow or enter the value** to the radius field to add a fillet to the object. **Then press *OK*** from the *Fillet menu.*

15mm Fillet added to the bottom of the cup

SHELL **Select the *Shell***  **tool**. The tool can be found in the toolbar at the top of the view, in the *solid* tab. When the tool is activated, select the top face (flat surface) of the object. **Adjust the thickness of the cup's walls by entering a value on the shell menu or by dragging the blue arrow**. Now **press *OK*** from the *Shell* menu.

HINT - Fillet can also be added on the upper edges to make them smoother.

Finished cup after shell and fillet on the top side corners

DOORKNOB – TUTORIAL

OVERVIEW

This is a step by step tutorial, on how to 3D model a doorknob.

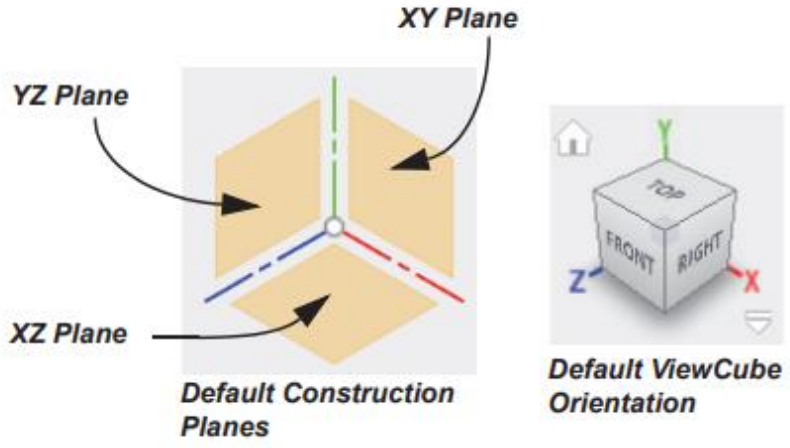
Tools included in this tutorial:

- Plane
- Sketch
- Line
- Conic Curve
- 3-Point Arc
- Sketch Dimension
- Revolve
- Hole
- Circular Pattern

3D modelled Doorknob

PLANE

Select *front view* from the *view cube* and **click *Create Sketch***  icon and then **select *YZ Plane or XY Plane*** for the sketch.



Default Planes and the View Cube

LINES

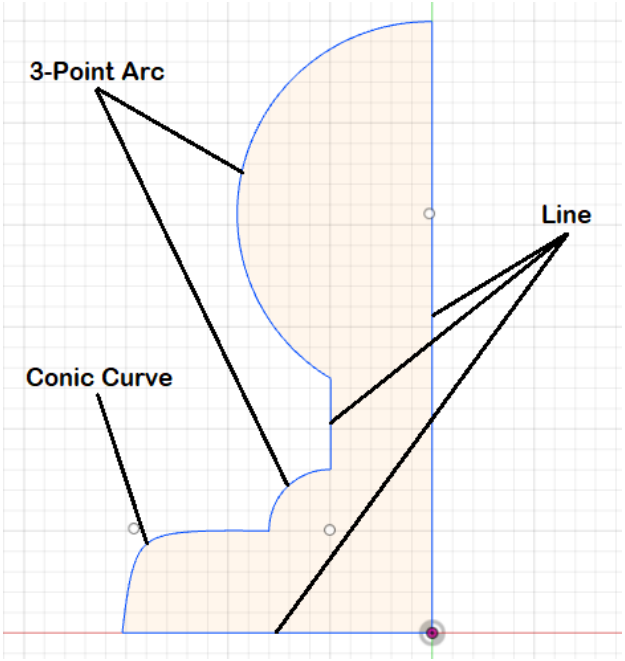
The *sketch* for the cross-section of a doorknob can be drawn easily with couple of lines. This tutorial demonstrates the use of three of the line tools: *Line*  *, 3-Point Arc*  and *Conic Curve*  . Example photo of the *sketch* is located underneath.

First draw straight *horizontal* and *vertical lines using Line*  ***tool***.

Then draw the other lines using Conic Curves and 3-Point Arcs.

Conic Curves  are drawn by selecting three points. First two points determines start and end points for the line. Then place vertex point to modify the curve for the line.

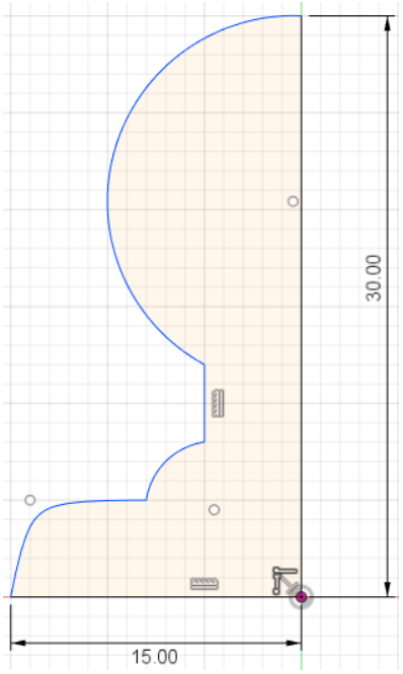
3-Point Arc *works essentially like the Conic Curve but maintains a circular shape*. Select beginning, end point and then adjust arc.



The result after drawing the lines

SKETCH DIMENSION

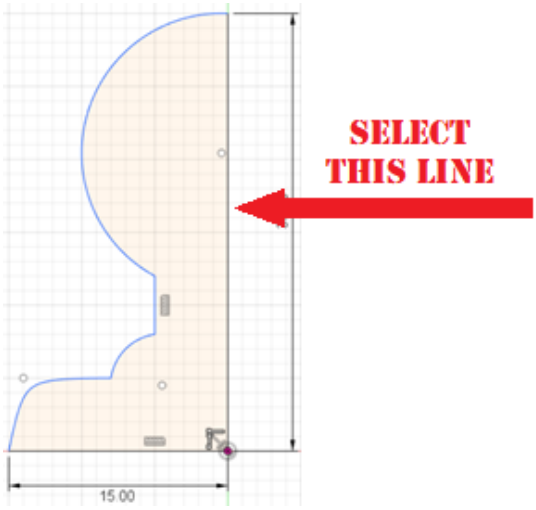
With the *Sketch Dimension* tool, points can be measured and locked to a specific length. **Use it to lock vertical and horizontal line lengths.** In the example photo below, vertical and horizontal lines have been locked with the dimension tool and their length is visible and can be changed. Modifying something in the project will not affect the length of these lines anymore. **Finish the *sketch* by pressing *OK*** from the *sketch palette* menu.



Dimension tool used between horizontal points and vertical points

REVOLVE

Revolve the sketch into a 3D object using *Revolve*  **tool**. The tool can be found in the toolbar at the top of the view, in the *solid* tab. **Then select the sketch** made in the previous step as the profile. **Select the long vertical line as the *axis***. Check the picture below for reference.



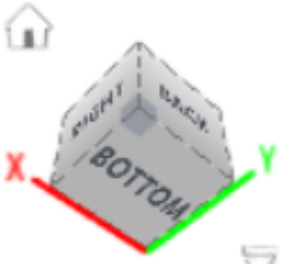
The sketch should be revolved around this line

After selecting the *axis*, **press *OK*** from the *Revolve* menu.

The sketch extruded into a 3D object

HOLE

This step instructs on how to make a screw hole to the doorknob. **Select bottom view from the *View Cube***. **Then select *Hole***  **tool**. It can be found in the toolbar at the top of the view, in the *solid* tab.



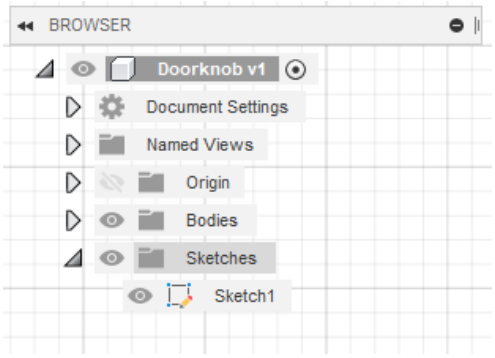
View Cube

Click on the bottom of the body to make a hole. **Adjust the hole size, location**, and finally **press *OK*** from the *Hole* menu.

A hole added to the doorknob

CIRCULARN PATTERN

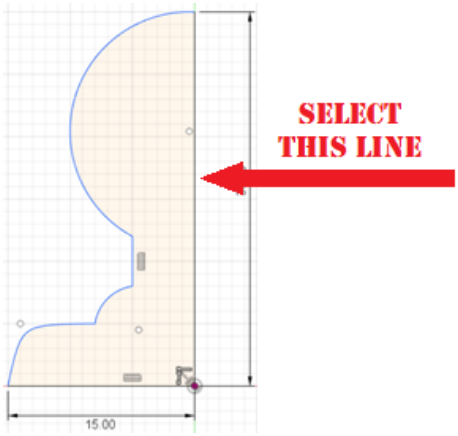
The sketch that you have created might be hidden, so make sure it is visible. It can be made visible by clicking the eye icon from the Browser.



The Browser

Select *Circular Pattern*  ***tool*. Select the previously made hole as the *Object*** in the circular pattern menu and **make sure that the pattern type is set as *Faces*** as the created hole is not an object, but merely a *face*. *Faces* could be described as object's surfaces in 3D modelling. So, we are basically copying the hole surface appear in multiple positions.

Finally select the long vertical line from the sketch as the axis.



The line that should be select as the axis

After selecting the axis, new fields appear on the circular pattern menu. **Enter quantity of the screw holes** and **press *OK*** from the *Circular Pattern* menu.

Finished doorknob with screw holes.

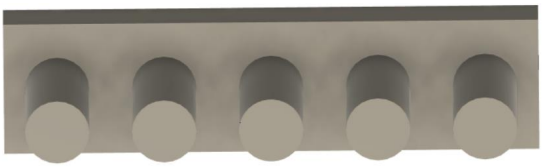
WALL MOUNTED COAT HANGER – TUTORIAL

OVERVIEW

This tutorial gives a step by step guideline, on how to make a wall mounted coat stand.

Tools included in this tutorial:

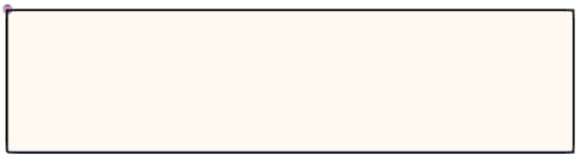
- 2-Point Rectangle
- Construction Line
- Center Diameter Circle
- Rectangular Pattern
- Extrude



3D modelled Wall Mounted Coat Stand

SKETCH

First select the Sketch  **tool** and click on one of the *planes*. **Use the *2-Point Rectangle***  **tool** to draw the base of the coat stand. **Press *OK*** from the *2-Point Rectangle menu.*



Finished rectangle sketch

CONSTRUCTION LINE

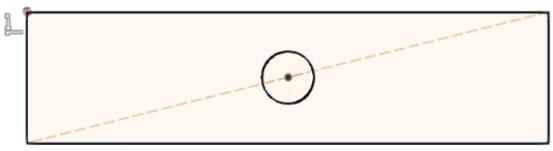
Select the *Line*  **tool**. When the *line* tool is active, the sketch palette opens on the right. Under the sketch palette's options, **enable the *Construction***  **tool**.

Draw a *construction line* diagonally across the rectangle as shown in the picture below.

Construction line across the rectangle

CENTER DIAMETER CIRCLE

Draw a Center Diameter Circle  **in the middle of the rectangle**. The cursor will be on the center when the triangle indicator appears. **Click there to set the circle center and then finish it by giving it a radius**.

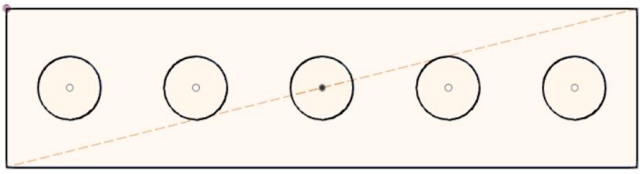


Circle drawn in the middle of the rectangle

RECTANGULAR PATTERN

Go to the *sketch* tab and **select *Rectangular Pattern***  **tool**. While the rectangular pattern tool is active, **select the edge of the circle**.

Drag the arrow to duplicate the circle or enter a value to the distance field in the Rectangle Pattern menu. The quantity of the circles can also be changed on the rectangular pattern menu. **Change direction type to symmetric** to duplicate on both left and right sides of the initial circle. Now the sketch is finished, **press *OK*** from the *Sketch Palette* menu.

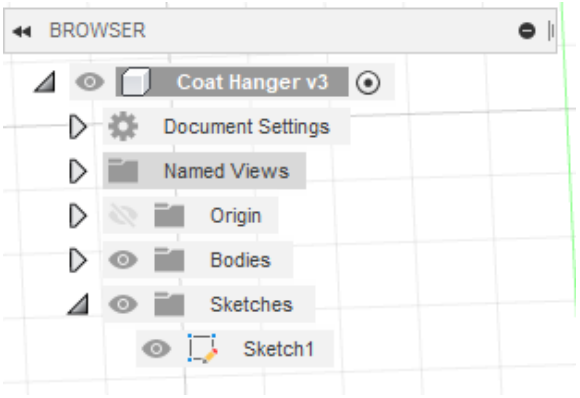


Circle duplicated with Pattern tool

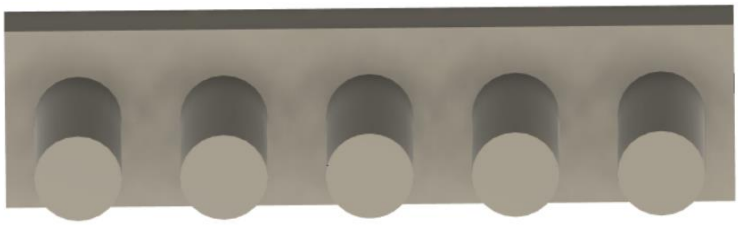
EXTRUDE

Select the *Extrude*  **tool**. Now that the extrude tool is active, **select all the circles and enter a value to the distance field** in the extrude menu and **press OK** from the extrude menu.

After extruding the circles, **extrude the base of the hanger**. The sketch might not be visible, so make sure that eye icon is active in the *Browser* menu. You can change the visibility by clicking the eye icon.



Browser menu



Finished coat hanger