Fusion 360 Guide

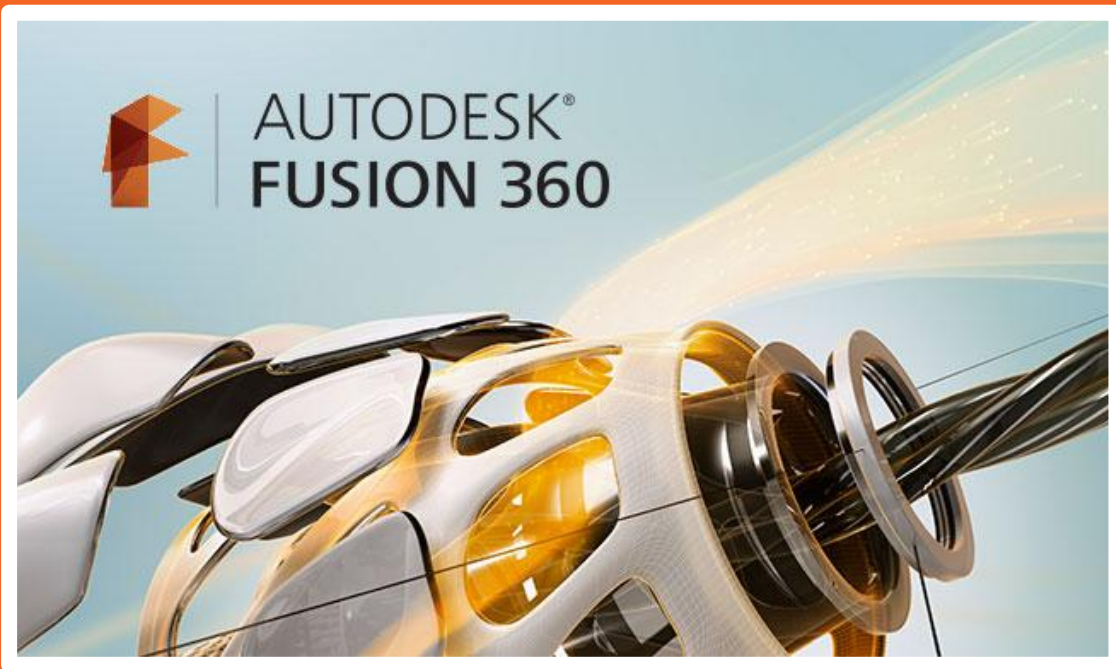FeMaidens Design
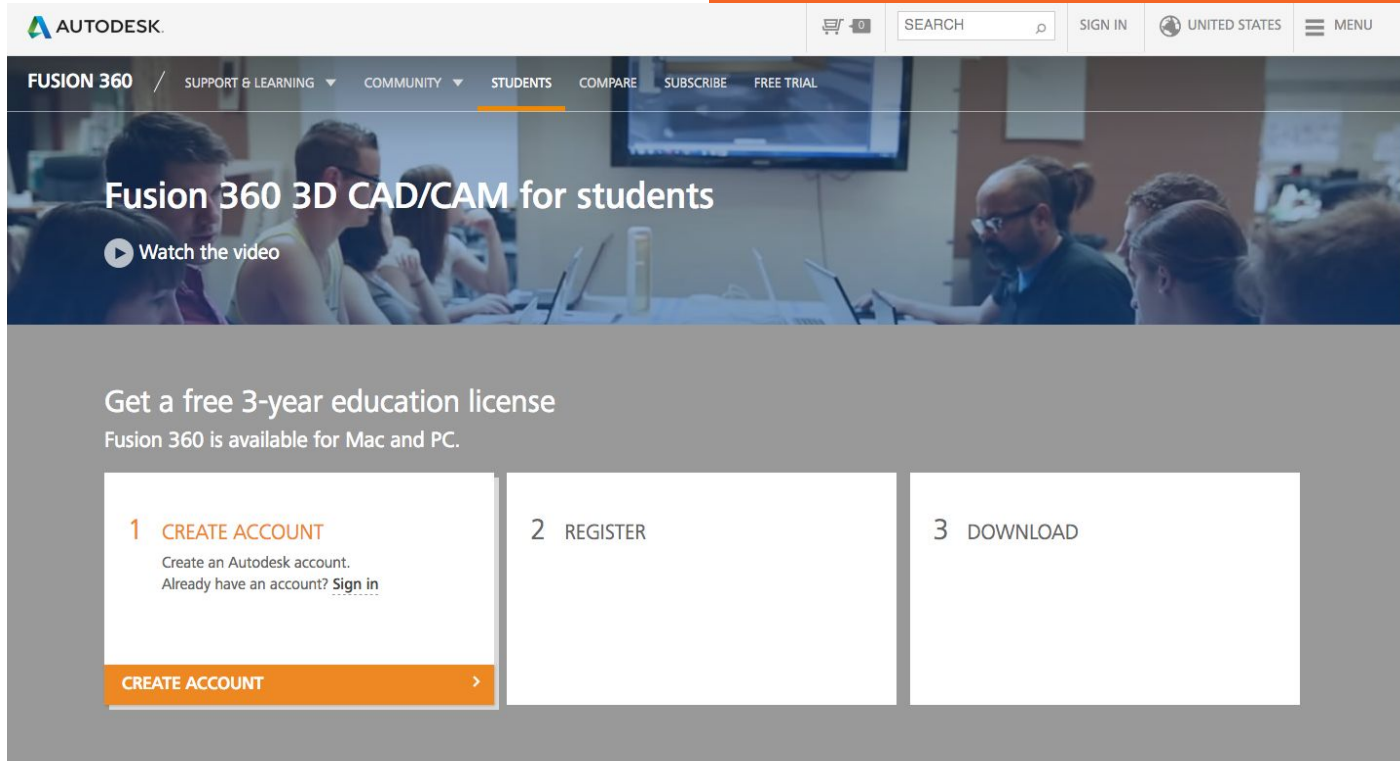
https://www.autodesk.com/products/fusion-360/blog/august-8-update-whats-new/

new stuff to check out from the latest update

What is Fusion 360?

- Fusion 360 is a platform that allows us to design and create 3D renderings of separate robot parts and mechanisms to finally join them together in 1 whole robot CAD.

- Fusion also allows us to prototype and run models under physics tests

Downloading Fusion 360

Fundamentals

1. Application Bar

⊞	**Data Panel** – Used for data management and collaboration.
▤	**File** – Create a New Design, Save, Export, and 3D Print.
💾	**Save** – Save an untitled design or save the changes to a design as a new version.
↩ ↪	**Undo/redo** – Undo/redo operations.

2. Profile & Help

	Profile – In your profile you can access your own personal settings.
?	**Help** – In the help menu you can access online learning content, help, forums, step-by-step tutorials, or link to community content.

3. Toolbar

SCULPT	The Sculpt workspace is used to create organic shapes by manipulating faces, edges, and vertices.
MODEL	The Model workspace is used to create solids with hard edges and flat faces.
PATCH	The Patch workspace is used to create open surfaces to stitch into solid bodies.
RENDER	The Render workspace is used to set up the environment and create photo-realistic renderings.
CAM	The CAM workspace is used to create and simulate tool-paths then generate g code for subtractive manufacturing.
ANIMATION	The Animation workspace is used for to create exploded views of an assembly and control over unique animations of parts and assemblies.
	The Drawing workspace is used to generate 2D manufacturing drawings.

Fundamentals

4. User Interface:
The layout of Fusion's controls is established directly on the screen that opens upon starting Fusion.

4. ViewCube:
Alters perspective and orientation of the model
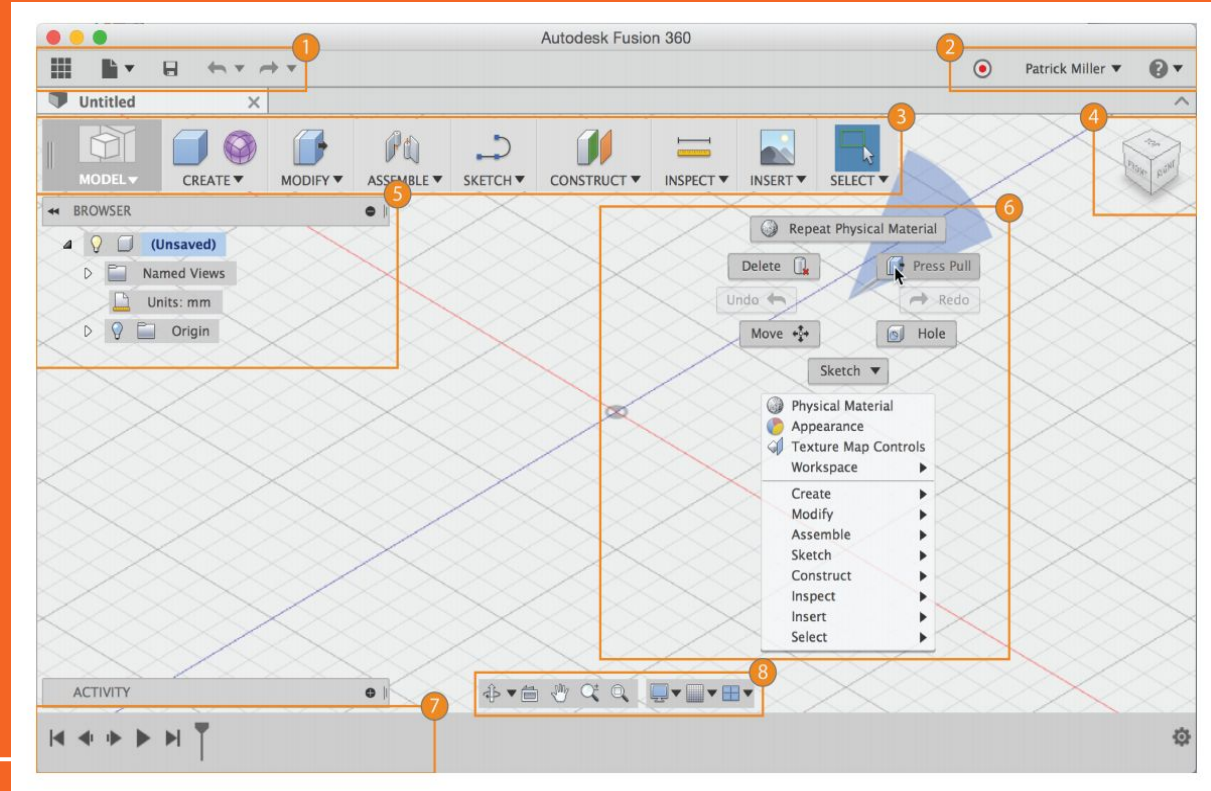
Fundamentals

5. Browser:
Is the family tree for your design. Every component within is listed here. It also acts as a history timeline.

6. Canvas & marketing:
Left click to select objects in the canvas. Right-click to access the marking menu. The marking menu contains frequently used commands in the wheel and all commands in the overflow menu
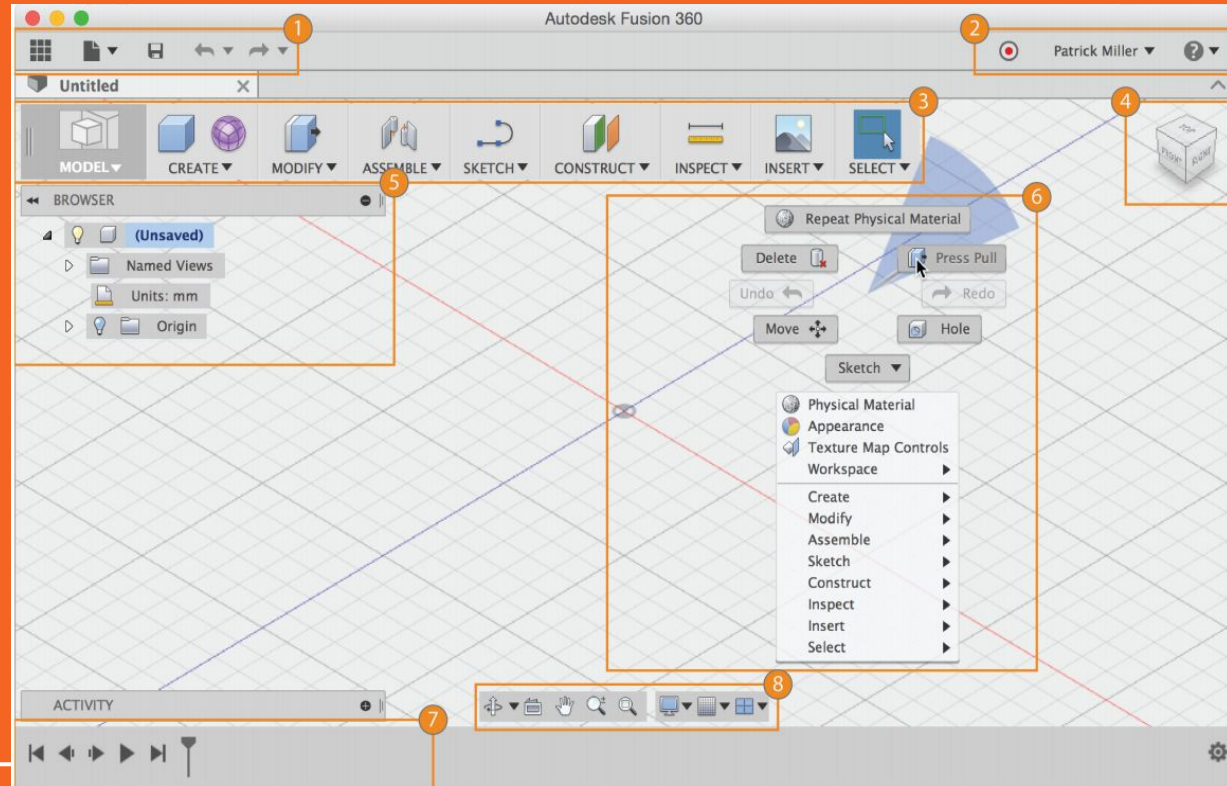
Fundamentals

7. Timeline

Lists operations performed on the design. To make changes in the timeline simply right-click operations

8. Navigation bar & display

Zoom, pan, & orbit your design as well as controls how the interface looks and how designs are displayed
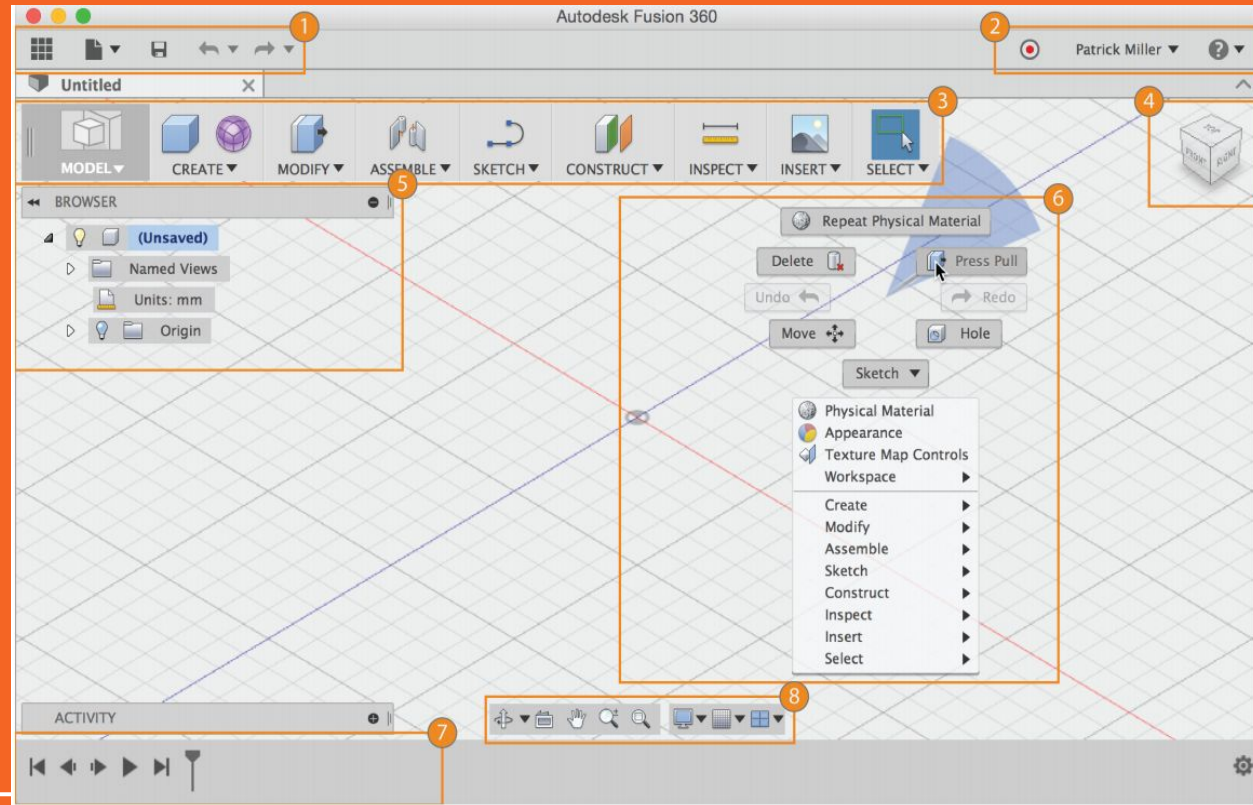
Sketch Constraints

- The constraint commands are in the **Sketch Palette** on the right side of the screen.
- You may use these geometric constraints to create persistent relationships in your sketch.

Note: Some of these constraints may already be created so you won't need to bother recreating them!
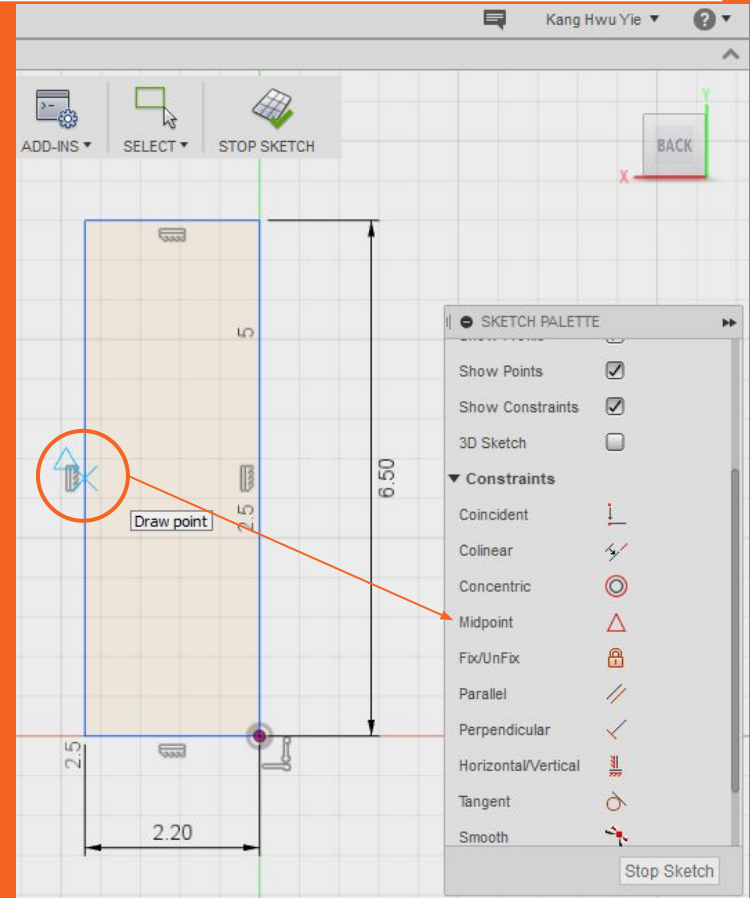
List of constraints

Coincident
Point to point, or snap point to entity.

Collinear
Add manually only.

Concentric
Circle, arc, or ellipse have the same center point.

MidPoint
Point to point, or snap point to entity.

Fix/UnFix
Locks the sketch entity in the current X, Y position.

Parallel
Second line parallel to first line.

Perpendicular
Second line perpendicular to first line.

Horizontal/Vertical
Line parallel to horizontal or vertical sketch axis, whichever is closer to the current position.

Tangent
Inferred to circles, arcs. Manually added to circles, arcs, lines

Smooth
Creates a curvature continuous (G2) condition between a spline and another curve.

Equal
Sets the selected entities to have equal size.

Symmetry
Sets the selected entities to have equal size and position across a selected center line.

Sketch

Sketch Plane

Every time you create a sketch, you will have to choose a **sketch plane** to sketch on. Its importance is that it'll determine **which way your 3D features can be created**.

Select a plane or planar face
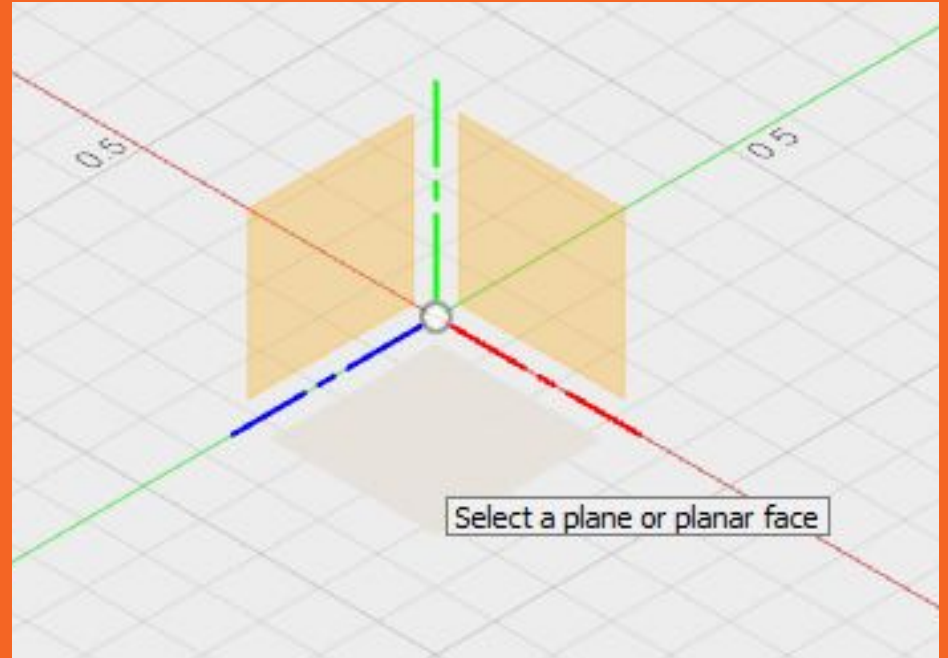
Create Lines

1. Select Sketch → Line OR simply

press L.

2. Select the sketch origin.

3. Click to end the line

4. Continue sketching lines as follows

5. Press esc to exit the command

Create Circles

1. Select Sketch→ Circle → Center Diameter Circle

2. Now hover over the origin   (center) of the sketch. You should see the cursor "snap" to this location.

3. Click once to begin placing the circle.

4. Move the mouse away from the center to define the size.

5. Click again to place the circle.

Create Rectangles

2-Point Rectangles

1. Select Sketch → Rectangle → 2-Point Rectangle
2. Click on origin to specify first corner
3. To specify opposite corner:
   a. Click another point on the graphics window
   b. Enter values for length and width

   (Enter one value and press TAB to lock value and jump to other value)
4. Press ENTER to end action

Create Rectangles

3-Point Rectangles

1. Select Sketch → Rectangle → 3-Point Rectangle
2. Click on origin to specify first corner
3. Specify 2nd point
   a. Click a point in the graphics window
   b. Enter value, press ENTER to lock value.
      Click a point to specify the angle of line
4. Specify 3rd point
   a. See 3a
   b. Enter value, press ENTER to lock value. Click a point to specify the position of the rectangle.

Create Rectangles

Center Rectangles

1. Select Sketch → Rectangle → Center Rectangle
2. In the graphics window, click to specify the center of the rectangle.
3. To specify the corner:
    a. Click a point in the graphics window.
    b. Enter values for length and width, and then press Enter. Click a point to define the position of the rectangle.
    c. Enter one value, and then press Enter to lock the value. Click a point to define the other value.
4. Click a point to begin another rectangle, or right-click and then click Cancel to end the command.

Create Dimensions

1. Select Sketch → Sketch Dimension OR simply press D.

2. Select the line

3. Click again to place the dimension.

4. Type in a value of the dimension.

5. Press Enter to accept the value.

Create Angle Dimension

1. Select Sketch → Sketch Dimension.

2. Select the bottom line

3. Select the angled line

4. Place the dimension

5. Type angle value.

6. Press Enter to accept the value.

New Component

- Components can contain bodies, sketches, joints, planes, and other objects. You can drag bodies and other objects from one component to another in the browser.

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Extrude

1. Select Modify → Press Pull OR simply press Q
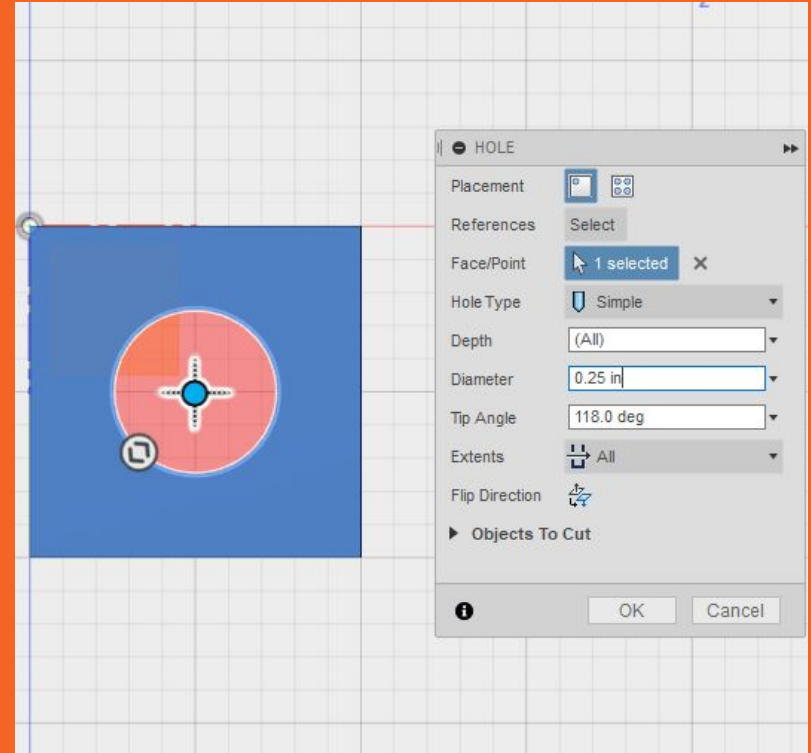
2. Select the Profile.

3. Press OK to finish.

Everything about Holes

A. Creating a hole:

1. Click → Create→ Hole.

2. Select the top face of the box.

3. Drag the center of the hole to the center of the box.

4. Set **diameter** value.

5. Change Extents to **All**.

6. Click OK.

B. Editing a hole:

- Select Edit Hole from the browser context menu or use Press Pull.

C. <u>Adding threads to a hole:</u>

1. Select the hole that you want to apply threads to, and then choose **Create →Thread**.

2. In the Thread dialog, click **Modeled** to preview the thread.

3. Specify Thread Type, Size, and Design. If desired, click Remember to copy your dimensions.

4. Right-click a cylinder.

Note: To use the previous thread dimensions, click **Repeat Thread.** Drag the arrow to the desired thread length, or click Full Length.
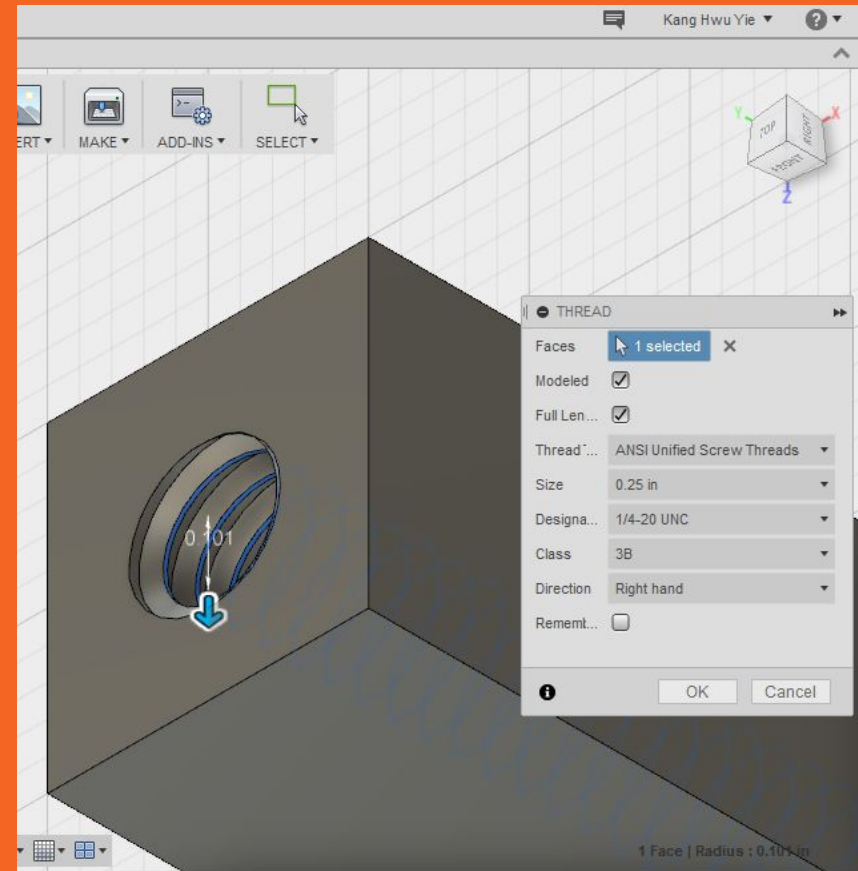
Threading

1. Go to the thread command under the Create option in the toolbar.

2. Select a body to put thread on.

3. Check "Modeled" option to get physical threads instead of visual threads.

4. Select "OK."

Modify

Use the modify commands to add or remove geometry to progress your design.

Fillet/Chamfer

1. Select Sketch > Fillet/Chamfer
2. Select the intersection point
3. Select the other intersection point
4. Enter a value you desire.
5. Press Enter to confirm

Assemble

1. Open data panel.
2. Right click on on file you wish to insert or assemble to your current file.
3. Click on "Insert"

Joint

	Joint Type	Description	Motion Allowed
	Rigid	Locks components together, removing all degrees of freedom.	None
	Revolute	Allows the component to rotate around joint origin.	1 rotation
	Slider	Allows the component to translate along a single axis.	1 translation
	Cylindrical	Allows the component to rotate and translate along the same axis.	1 translation 1 rotation
	Pin-slot	The component can rotate about an axis and translate about a different axis.	1 translation 1 rotation
	Planar	Allows the component to translate along two axes and rotate about a single axis.	2 translation 1 rotation
	Ball	Allows the component to rotate about all three axes using a gimbal system (three nested rotations).	3 rotation

Creating Planes

1. Go to "Contruct"
2. Click on "Plane Along Path"
3. Hover over "Construct"
4. Click on "Offset Plane"
5. Click on face you want to offset.
6. Type in the distance you want to offset.

Robot Assembly

Importing STEP Files

Numerous types of files can be uploaded into Fusion 360 through the Data Panel, but we will usually deal with STEP files(*.ste, *.step, *.stp)

1. Download the file you want to import
2. Extract All the file
3. Click the grid sumbol to access the Data Panel.
4. Click Upload and select the STP file you want to import.
5. Click Upload.
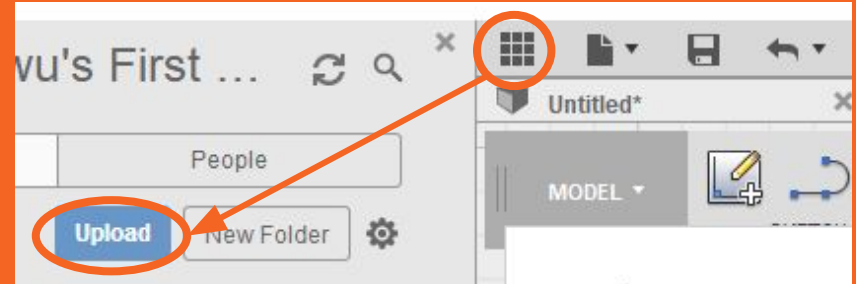
Publishing a Design to Construction

1. Name & Save Design to Correct Folder First (be mindful of edit or master)
2. Toolbar: Model: Drawing: From Design
3. Choose a first perspective to dimension the design
4. Drawing views allows you to add more views to the drawing as well as views of different perspective
5. Modify allows you to rotate and move the drawings
6. Dimensions allows you to select lines, radius/diameter, and distances from a line or point to the middle of a circle
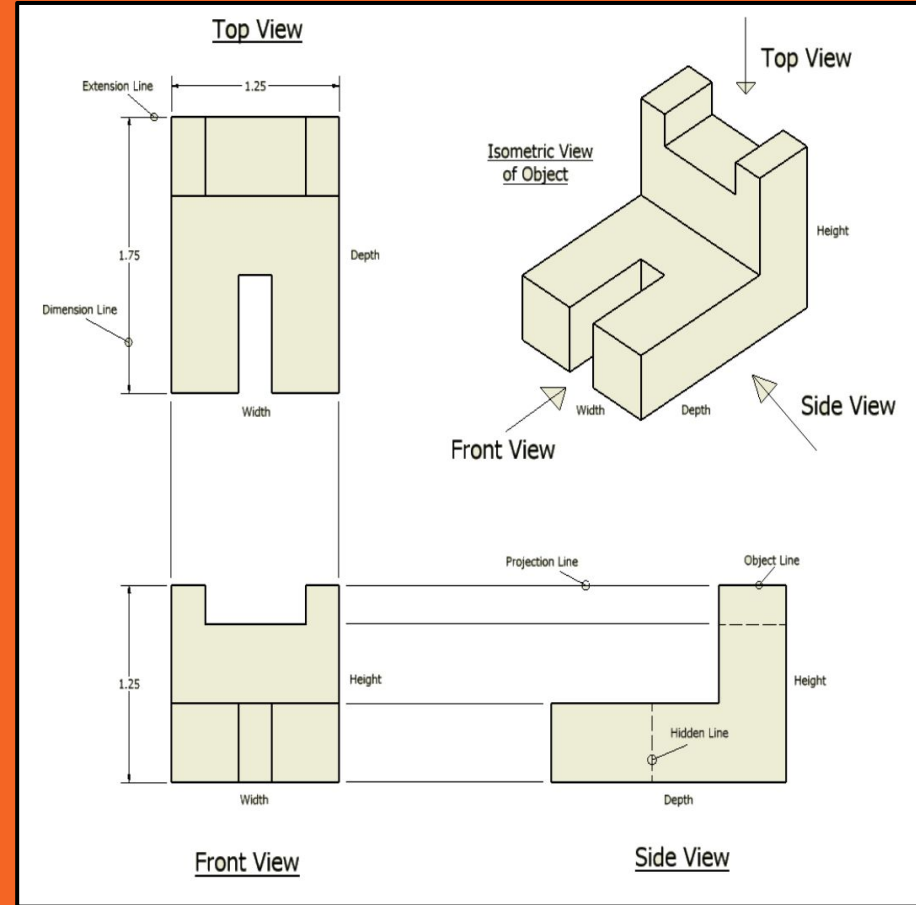
Naming

7. Text allows you to add on descriptions under project and title, or measurements in places that otherwise cannot be made from dimension

Annotating

8. Here the navigation bar & display shows annotation settings which can easily be toggled over to expose possible modifications to font, dimension precision, fraction/decimal format, and unit display