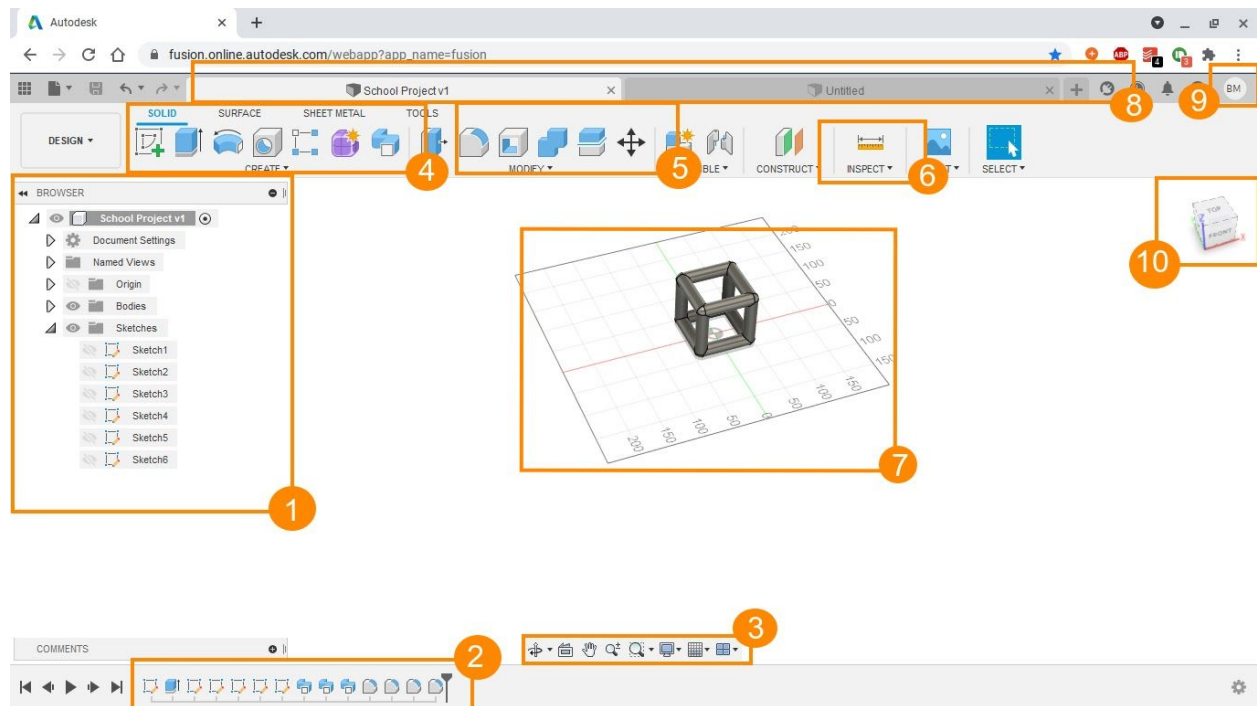# Fusion 360 Workspace



This is your fusion 360 Workspace. This is where you will be making your designs.

1. Browser. This is where all of your sketches & Bodies go.
2. Timeline. When you create a sketch, Use any operation (EX: Sweep, Fillet).
3. Navigation bar. This is where you move your camera to see more of your model. (See #10)
   Toolbar:
4. Create. This is where you create sketches, Sweeps, etc…
5. Modify. This is where you modify existing bodies.
6. Inspect. This is where you can measure bodies
7. Cartesian plane. Your models go on to this plane.
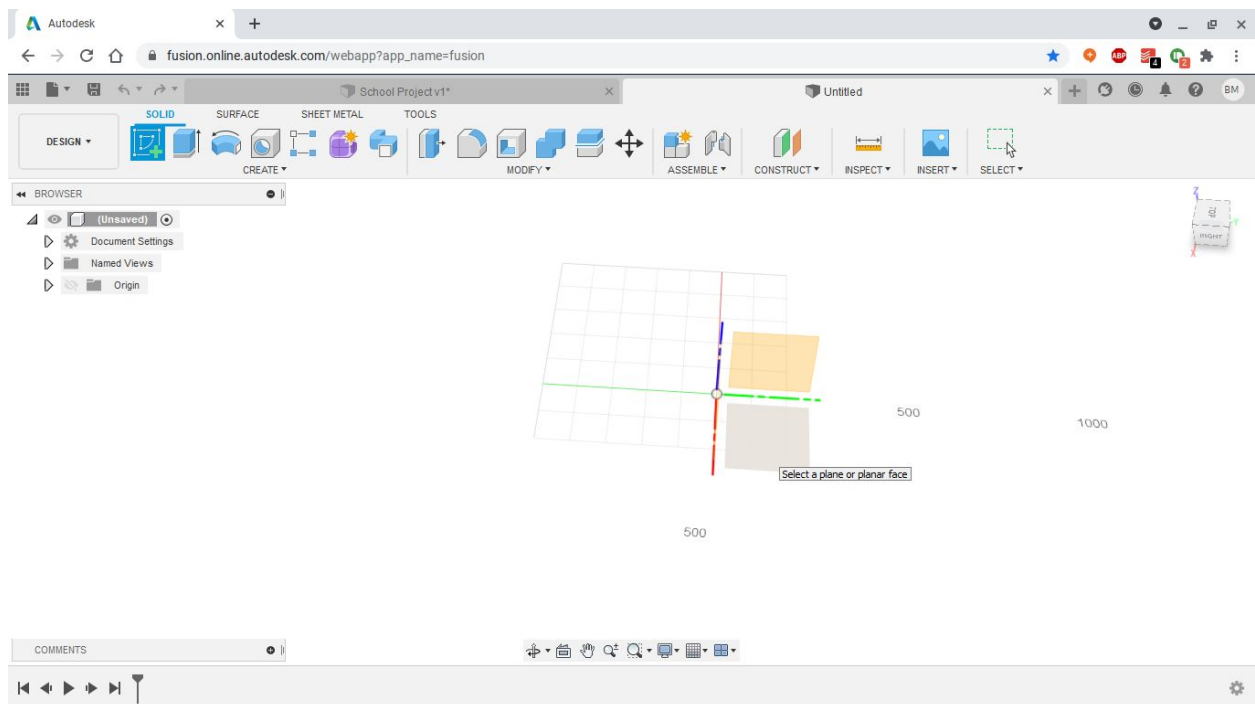8. Tab-bar. This is where your open files go. (When you are finished click the save button)
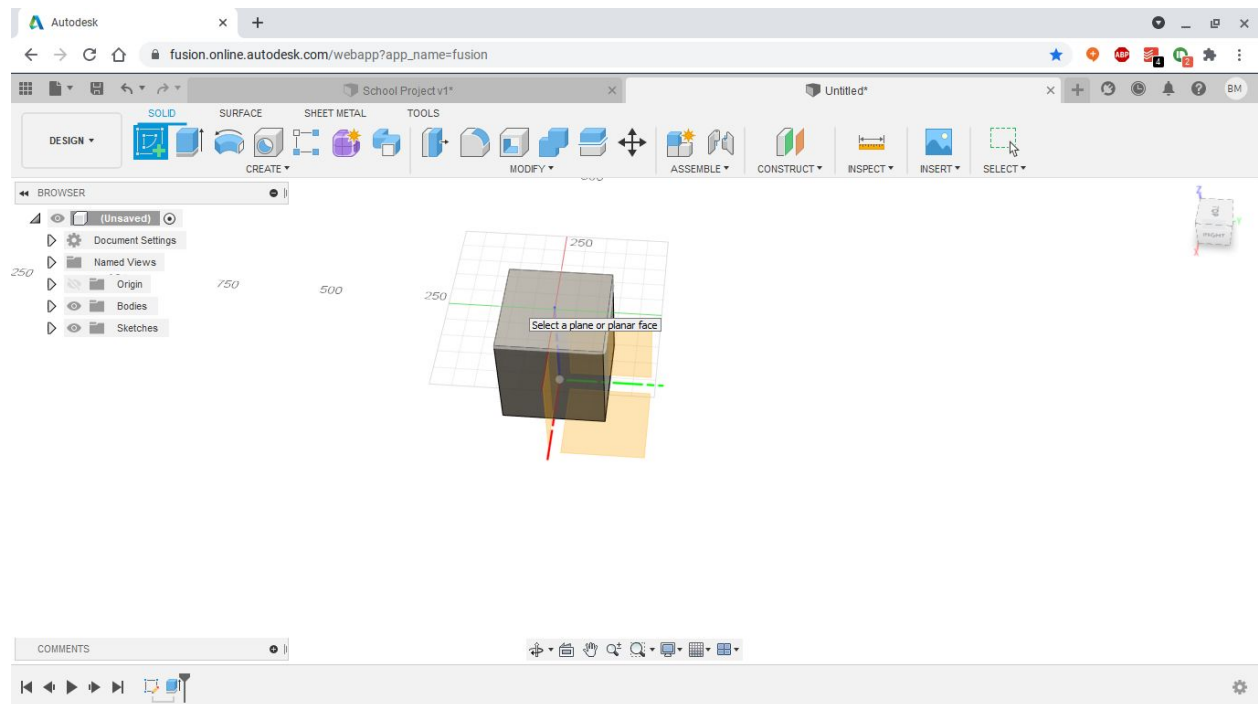
9. Settings menu
10. View Box.

# Creating a sketch.

Select the plane that you would like to build on: (2.1)



(2.1)

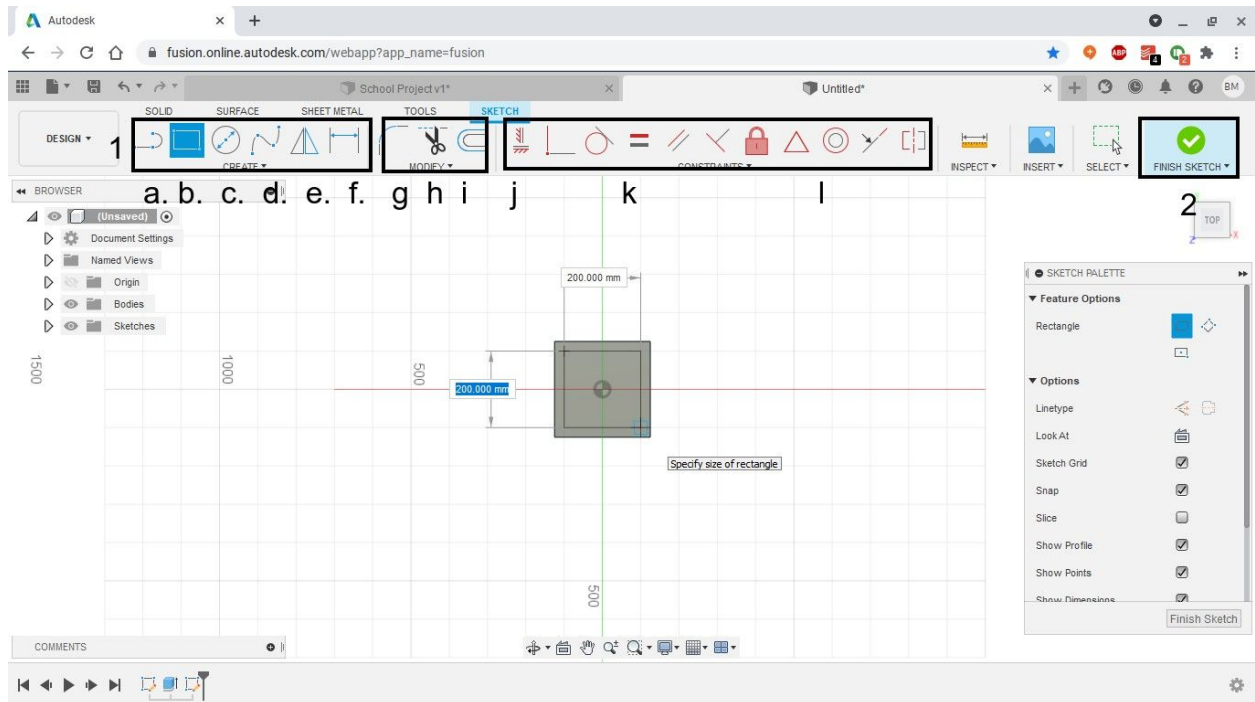You can also select a face of an existing body. (2.2)

(2.2)

Editing sketch.

(2.3)

When you create a shape you will see some dimensions popup. To edit the dimensions of the box that is highlighted you start typing. To go to the next box you press the tab key.

1. Toolbar
    a. Line tool. (Create line)
    b. Two-point rectangle (Create rectangles)
    c. Diameter circle (Create circles).
    d. Fit pint spline. (Create curved lines)
    e. Mirror
    f. Dimensions. (Used for editing dimensions)
    g. Fillet. (Curve points)
    h. Trim. (Trim shapes)
    i. Offset. (Create a new shape in the selected shape)
    j. Horizontal/vertical tool
    k. Equal. Will make one thing the same as the other.
    l. Midpoint (Triangle.) (Move shapes to midpoint)
2. Finish Sketch. Save & Exit sketch. (You can also use "E" to Save, Exit & Extrude).

Sketch creates a 2d Shape. When you use the extrude the shape you add height, Making it a 3d shape

## More Info:

https://www.instructables.com/3D-Printing-101-3-Intro-to-3D-Modeling/

https://www.youtube.com/channel/UCiMwMz3RMbW5mbx0iDcRQ2g

Want to try on your own? Try this project to start with:
https://www.instructables.com/Stamp-Project/