

• LESSON 01 •

Autodesk Fusion 360: An Introduction to 3D Modelling

Lesson Overview

You might have done some basic models in 2D in AutoCAD in the Engineering Graphics course. In this lesson, and throughout this module, we focus on mechanical aspects that are immediately relevant to the design and construction of mechatronics devices. You will learn the following:

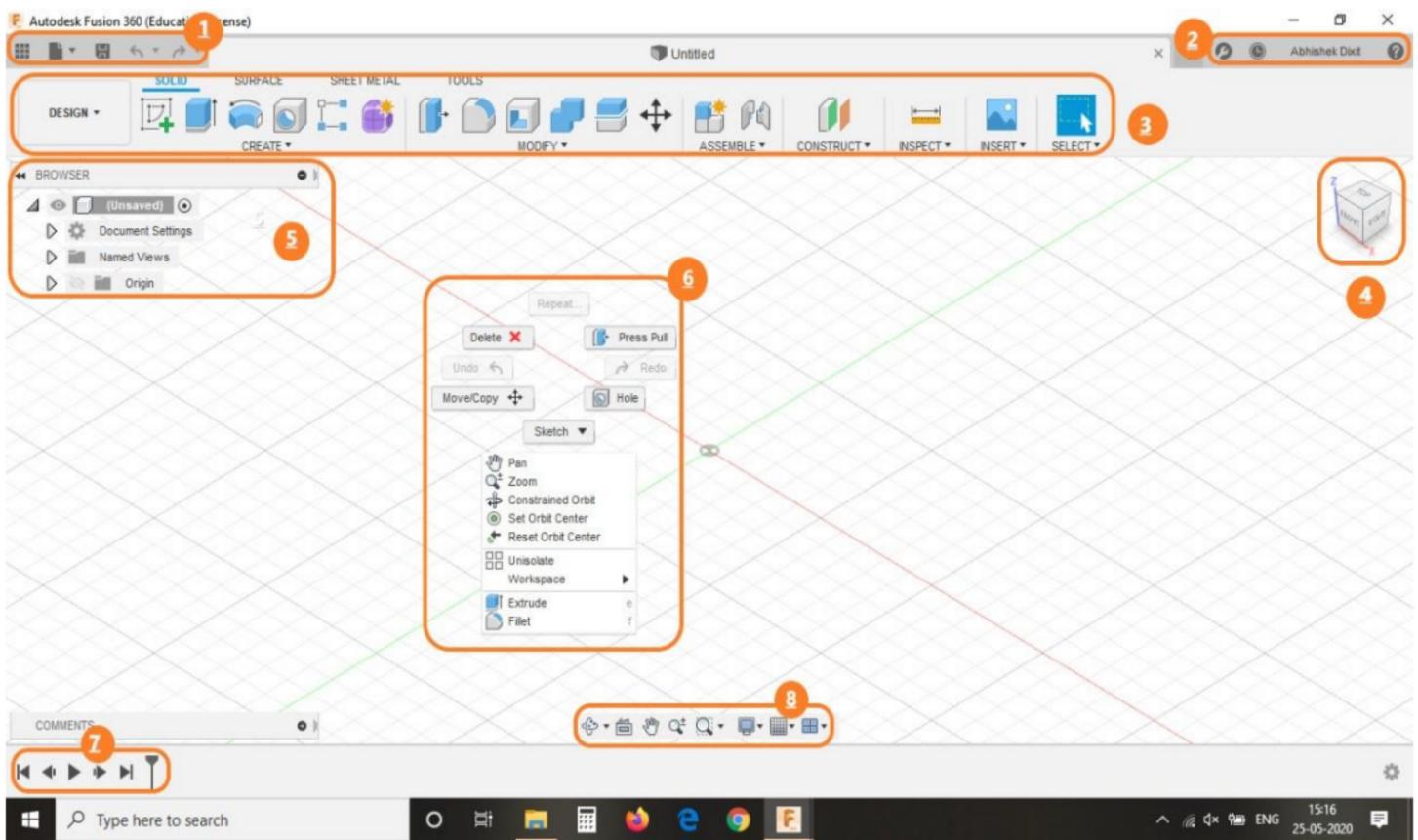
- Introduction to **Fusion 360** and basic UI.
- Modelling concepts in **Fusion 360**.
- Designing basic models in 3D.

Just like the previous module, every topic here will have an example or two, followed by a few practice problems at the end of the chapter – submission for which is not required.

Getting started with Fusion 360

1. User Interface

The UI can be broken up into 8 areas. To help you become familiar with the product we will describe each of these areas and go into more details with how to use it in the following lessons.



1) Application Bar

The application bar is where you'll find and use the following.

- 1.1)  **Data panel** - Used for data management and collaboration.
- 1.2)  **File** - Create a new design, save, export and 3D print.
- 1.3)  **Save** - Save an untitled design or save the changes to an existing design as a new version.
- 1.4)  **Undo/Redo** - Undo or redo operations.

2) Profile and help

In profile, you can control your profile and settings, or use the help menu to continue your learning or get help in troubleshooting.

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

3) Toolbar

Use the toolbar to access the workspace you want to work in, and the tools you want to use in the selected workspace.

SCULPT

- 3.1) **Sculpt** – used to create organic shapes by manipulating faces, edges and vertices.

MODEL

- 3.2) **Model** – used to create solids with hard edges and flat faces.

PATCH

- 3.3) **Patch** – used to create open surfaces to stitch into solid bodies.

RENDER

- 3.4) **Render** – used to set-up the environment and create photo-realistic renderings

CAM

- 3.5) **CAM** – used to create and simulate tool-paths then generate g code for subtractive manufacturing.

ANIMATION

- 3.6) **Animation** – used to create exploded views of an assembly and control over unique animations of parts and assemblies.

4) ViewCube

Use the ViewCube to orbit your design or view the design from the standard view positions.

5) Browser

The browser lists the objects in your design. Use the browser to make changes to your objects and control their visibility.

6) Canvas & Marking menu

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 the commands in the overflow menu.

7) Timeline

The timeline lists operations performed on your design. Right-click operations on the timeline to make changes. Drag operations to change the order they are calculated.

8) Navigation bar & Display settings

The navigation bar contains commands used to zoom, pan and orbit your design. The display settings control the appearance of the interface and how designs are implemented in canvas.

Now that we are familiar with the UI, let's create a 3D model of a box with a hole.

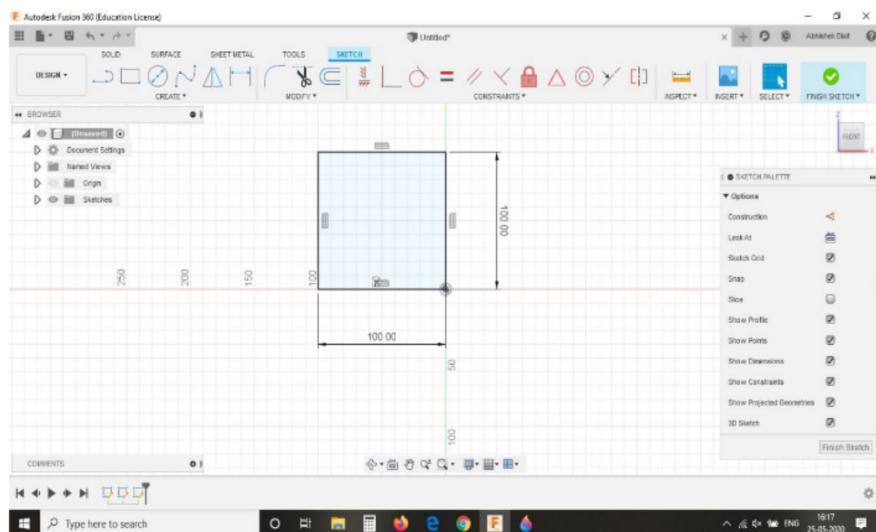
2. Designing 2D sketches

Step 1: Create a new design

1. On the “Create” tab on the toolbar, select  “Create sketch” option.
2. Select the plane (XY, YZ or ZX) on which you want your 2D sketch upon. Generally, XZ plane is preferred.

Step 2(a): Create a box

1. Sketch palette appears on the right side of the canvas. To get an orthographic view of the plane, select the “Look At” option.
2. In the create drop down box, select rectangle. Now, it's your choice as to how you want to construct the rectangle.
3. Start the sketch from the origin and select the vertices, 2 of them on the axes. You can type in the dimensions also. For now, consider the box to be a square of side 100mm. (Note: Units can be changed in Document settings of Browser)
4. Click OK.

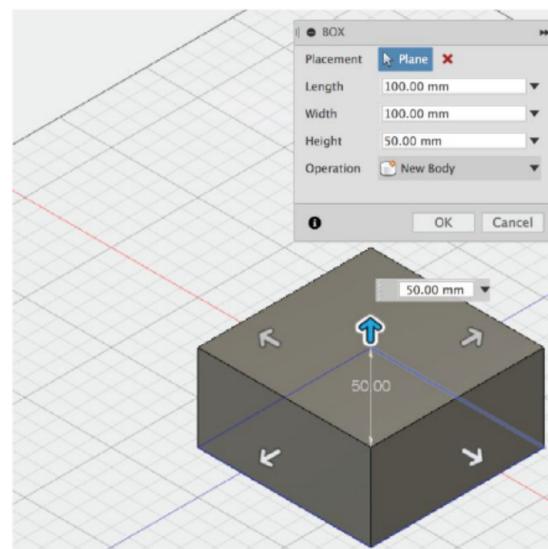


3. Converting 2D sketch into 3D design

You can make a 3D model of the box directly, convert a 2D sketch into 3D model. For the former, follow step 1, step 2(b) and then step 3. For the latter, continue from step 2(a) to step 2(c) and step 3.

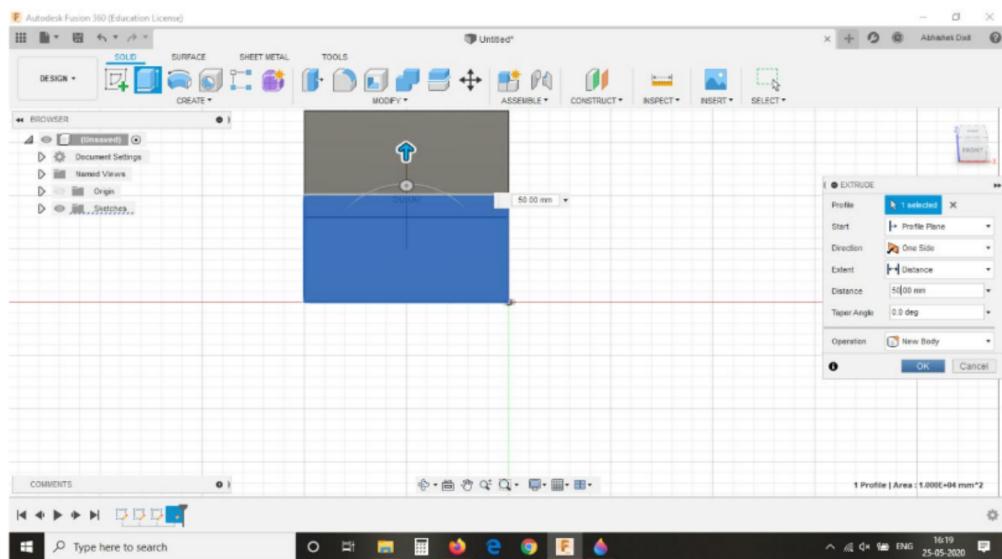
Step 2(b): Creating a 3D box directly

1. On the toolbar, go to the “Solid” section. From the “Create” drop down list, select box.
2. Select origin as the 1st vertex, then you can create a box of length and width of 100mm.
3. In the box dialog box, input the height to be 50mm.
4. Click OK.



Step 2(c): Converting 2D sketch to 3D

1. In the create toolbar, select Extrude.
2. Select the 2D sketch we just created.
3. In the Extrude edit box, enter 50 as the extrusion distance. Or you can just drag the arrow mark.
4. Click the OK button to proceed with creating the 3D part.

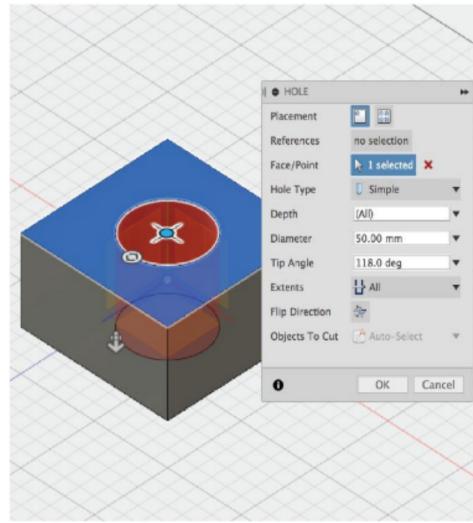


NOTE: Extrude option can also be used to create holes in a model. Sketch the shape of the hole you want on the 3D model, superimpose it on the model and use extrude for the superimposed sketch.

Step 3: Add a hole to the box

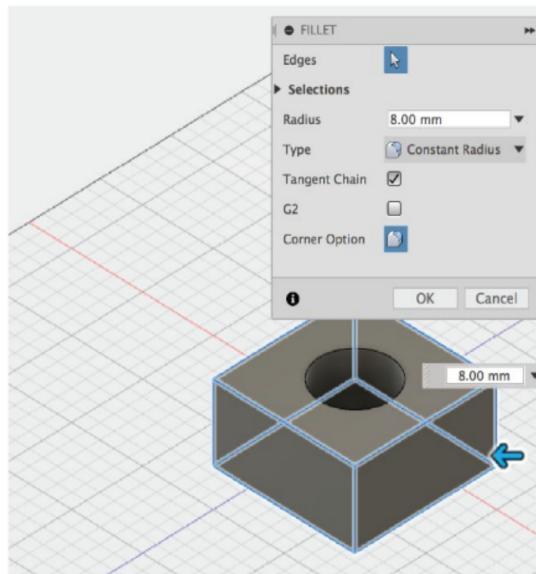
1. Click Solid > Create > Hole.
2. Select the top face of the box.
3. Drag the centre of the hole to the centre of the box.
4. Set the diameter of the hole to be 50mm.
5. Change Extents to All.

6. Click OK.



Step 4: Round the edges of the box

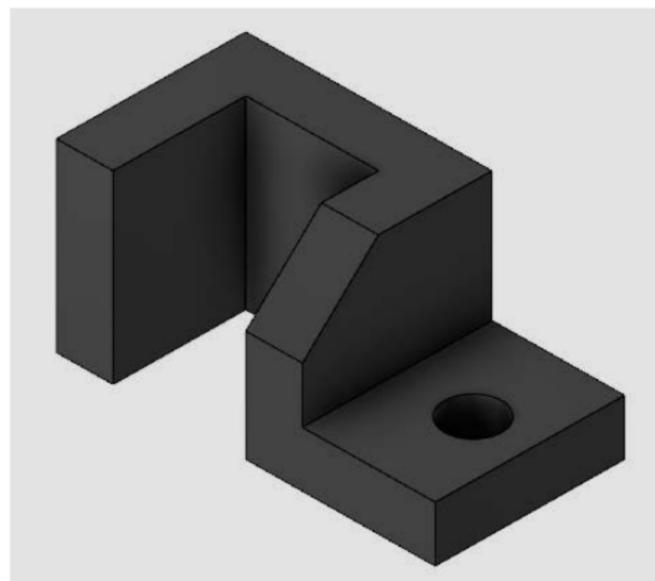
1. Right-Click an empty area in the canvas and select “Press Pull”.
2. Hold the left-mouse button then drag to windows to select the entire box.
3. Click the top and bottom edges of the hole to deselect the edges.
4. Enter 8mm for radius.
5. Right-click then select OK.



Step 5: Save the design

1. Click  to save the design.

Assignment:



Try this by yourself!

References:

<https://www.youtube.com/watch?v=A5bc9c3S12g> (Lars Christensen youtube channel)

<https://www.youtube.com/watch?v=qvrHuaHhqHI> (Product design online)