**Fusion 360 for Beginners:**

This tutorial is meant to help those new to Computer Aided Design learn how to navigate the basics of Fusion 360 as well as to learn some of Fusion 360’s most useful operations.

**Getting Started:** Before we begin to create our project lets familiarize ourselves with the different tabs and drop downs on the screen. Knowing how to change our view of the project makes it easier for us to make specific changes without making mistakes.

When we first open Fusion 360 we will see a blank workspace for a new untitled project. To start a new project or change the name of the current project look towards the top left corner of the window. The + symbol allows the user to create a new project and going to file “save” will allow the user to change the project name from “untitled” to something else.

Below the project name the user will see a tool bar with the tabs (SKETCH, CREATE, MODIFY, ASSEMBLE, as well as many others). These drop downs contain most of the operations we will be using in Fusion 360.

**SKETCH-** This drop down can be used to create 2 dimensional shapes including rectangles, ellipses, and circles. We can also draw lines and connect them to create more complex 2 dimensional shapes

**CREATE-** Allows us to edit an existing sketch or quickly create a basic 3d object such as a box or cylinder.
MODIFY- The modify drop down allows us to make larger changes to different bodies.

On the left side of the screen we see a drop down with the top tab being our project name. Currently the only thing we can see below this is a tab marked origin, but as we create new bodies and sketches they will be shown as separate tabs on the left side of the screen.

All tabs present on this side of the screen have a lightbulb icon that allows us to hide specific pieces of our project. This is useful when we only want to edit one specific sketch or body of our project and don’t want to make unintended changes.

(Notice how the origin disappears when the lightbulb is off)

Notice that having the origin on shows us three yellow squares. These squares represent the XY, YZ, and XZ planes.

*In fusion 360 it is important to realize that most of our work begins as a 2 dimensional sketch that we then bring into the 3rd dimension at a later point*

Now let’s look toward the bottom of the screen. Here we can see a bar that gives us the ability to view our workspace from different perspectives.

The “Pan” option represented by the hand icon allows us to drag and move the workspace.

The “Look at” option represented by the rectangle icon allows us to view one 2 dimensional plane in our workspace. Simply click the icon and then click one face on either the origin or an existing body.

The “Orbit” option represented by the leftmost icon allows the user to spin or rotate the entire workspace by clicking and dragging it. (This can also be achieved by clicking and dragging on the cube in the top right corner of the screen)
**Time to build:** Below I will discuss how we can make a very basic house as a project. I’ll go through how to make new sketches and as well as how to use some of the most useful operations in Fusion 360.

Recall that when we want to create something new in our project that we start by working only in 2 dimensions. To do this we start by selecting one of the planes on our origin or on an existing body and then clicking on the “SKETCH” drop down menu. This will bring us into a 2 dimensional point of view.

Let’s start by creating a box. Within the SKETCH drop down move to rectangle and then select 2-point Rectangle. After doing this click once to set the location of the first point. Then either drag or begin typing to set the dimensions of the rectangle (Begin typing to set the first dimension and click tab to be able to set the second dimension).

At this point select “stop sketch” on the right side of the top toolbar. Now we have a new rectangle sketch, which we can hide or show using the “Sketches” drop down on the left side of the screen.

Now that we’ve successfully made a 2 dimensional sketch let’s bring that sketch into the third dimension. We will achieve this by using the **Extrude / Press Pull** operations, which are found in the CREATE and MODIFY tabs respectively. These operations can also be accessed by right clicking on the desired sketch. Using extrude allows us to assign a third dimension. You can either drag the arrow or type a positive value into one of the fields marked 0.00mm.
On the above example I assigned a Distance value of 35.00mm. Now I will show how we can use extrude to hollow out pieces of our project.

Above I created a new rectangle sketch on top of our existing box. I then accessed the extrude operation and assigned a negative value resulting in the image on the right.

*Extrude is one of the easiest ways to add or cut away material from your project. You can always create a new sketch and use extrude to modify an existing body or create a new one*

By repeating the same process of making a new sketch and using extrude I was able to add windows and a door to our house.
Now let’s create a roof for our house. We do this by creating a new sketch and then using extrude. This time though we will adjust the “taper angle” so we have less material as we extrude. Note that a negative taper angle causes the extrusion to narrow while a positive one will cause it to widen.

Using only extrusion and sketches we have created a basic model of a house.

**Other important operations:**

**Fillet:** The fillet operation allows us to curve the edges of the bodies we create. This can be done by either going to the MODIFY drop down, selecting fillet and then the desired or simply by right clicking on the edge you want to add a fillet to.

It is often important to use fillets on many of the edges in a project because the 3d printers often have trouble printing at 90 degrees.

**References:**

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