

SOLIDWORKS 101:

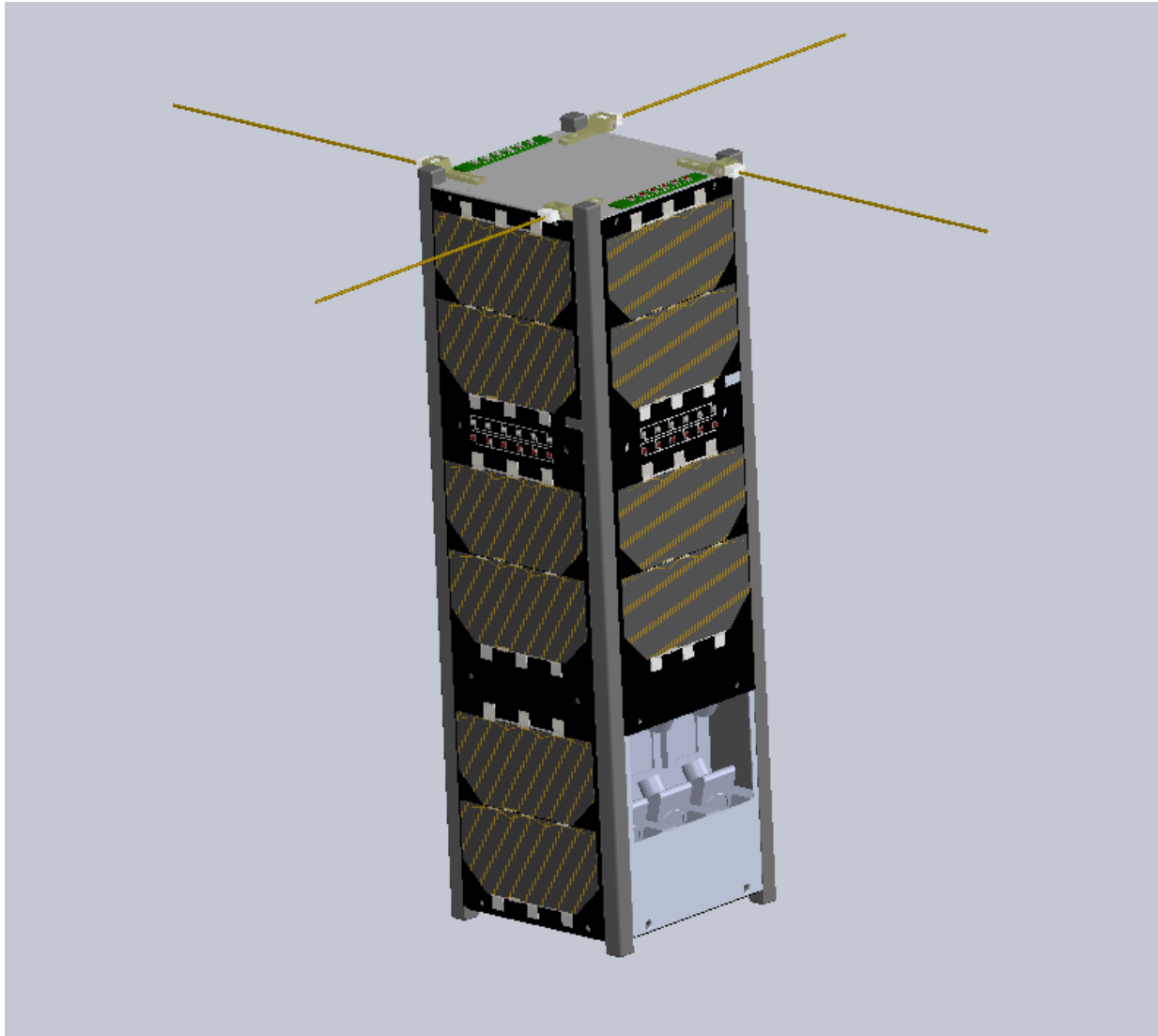
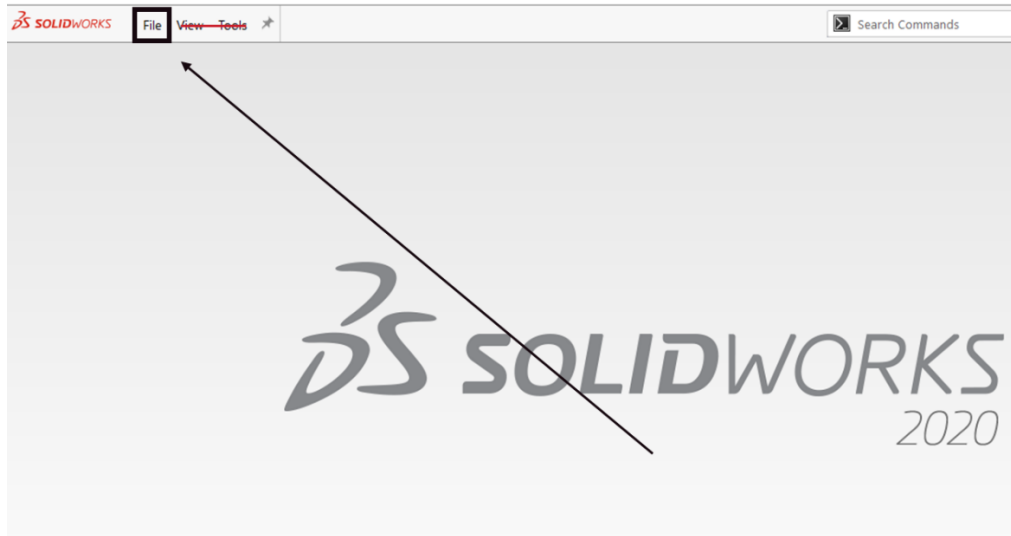


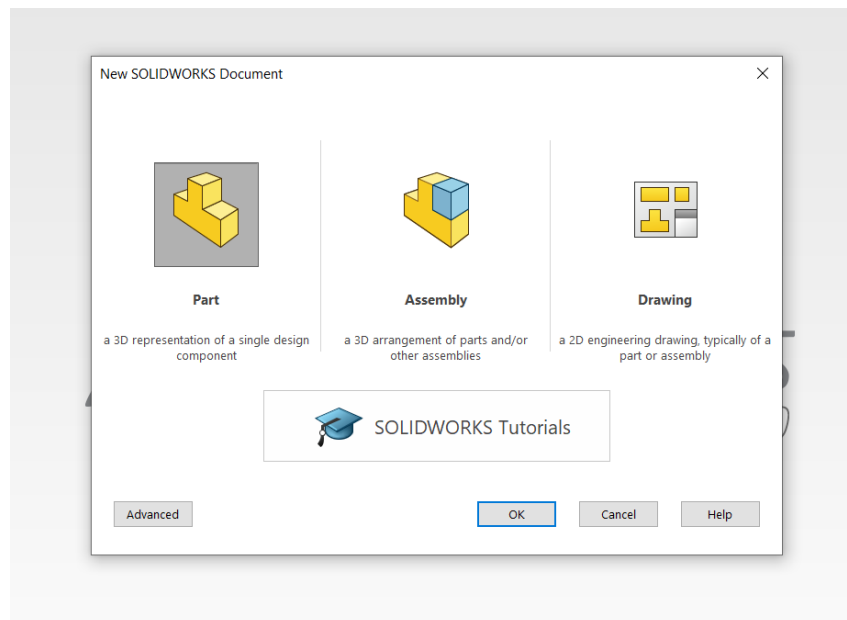
Figure 1 Full SOLIDWORKS CAD model assembly of QSET's CubeSat for the 2017 CSDC.

User Interface Basics:

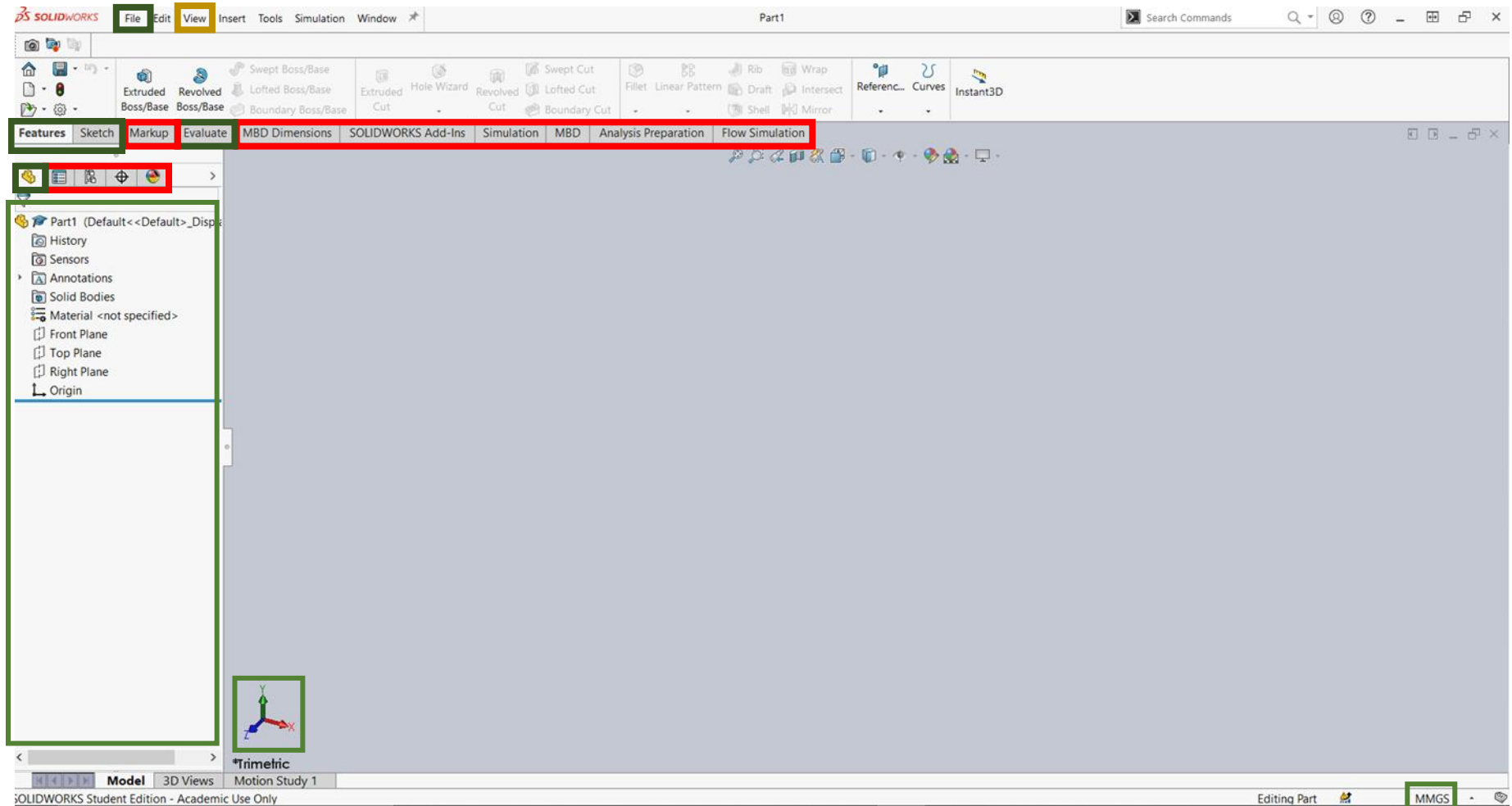
Here we will briefly try to outline some aspects of the user interface you should become familiar with and some aspects that are rarely used, so can be avoided. If you are using the 2020 version of SOLIDWORKS, when you first click on the program you should be greeted with the following screen:



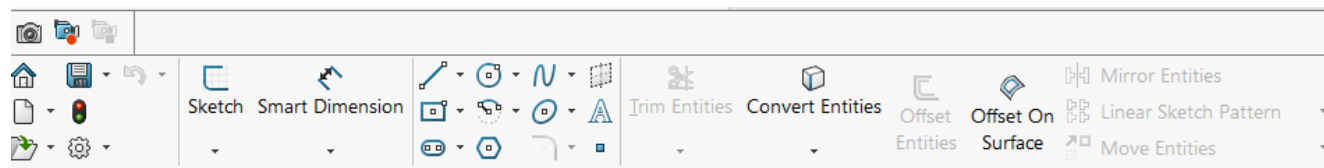
Click on **“File”**, **“New”** and **“Part”**. This will open a new part file. In SOLIDWORKS there are three types of files you can begin. A SOLIDWORKS parts used to make the individual parts has file formatter .sldprt. Next is a SOLIDWORKS assembly and sub assembly, these are both considered just an assembly, but for more complex designs it is useful to have both classifications. A assembly consists of multiple parts and has file formatter .sldasm. Finally, is a SOLIDWORKS drawing, which are useful for making technical schematics or displaying you design with measurements these carry the file format of .slddrw.



At first the user interface may seem slightly daunting, but there really is only a small set of buttons that are commonly used. We will begin by breaking all these buttons down into three categories: what is used in green, what is not used as much in yellow, and items that can be ignored (negating rare cases) in red. All green and yellow items will be discussed further below.

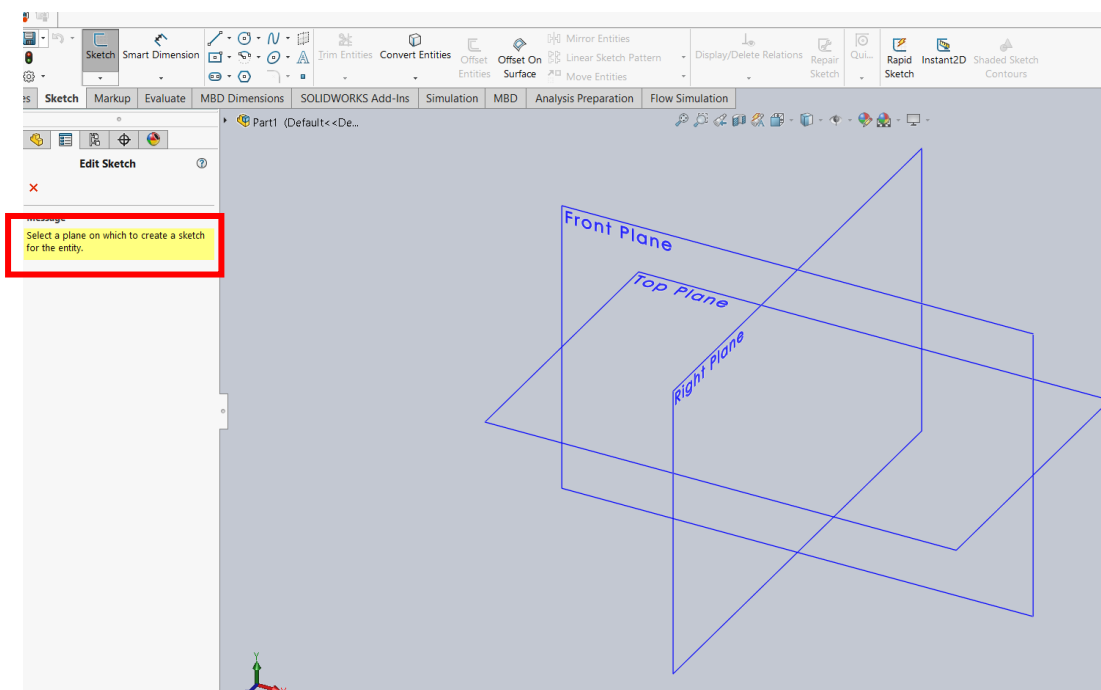


Sketch

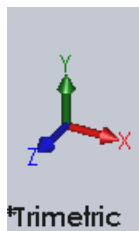


Beginning with what will probably be your first step in all CAD designs will be defining a primary sketch of the general shape of the object to be created. Once you have this general 2D outline drawn in SW, you can extrude it to 3D.

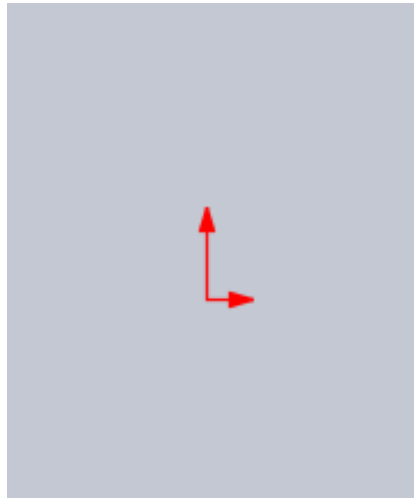
Beginning with defining a sketch. This can be done by clicking on the first button entitled 'Sketch'. With the sketch button selected it will ask you to select a plane to define where you will be sketching on.



The choice of plane doesn't matter in simple parts but will become more important in designing complex parts and assemblies. This is a good time to draw your attention to the coordinate axis. The three arrows in the bottom right corner display your current alignment with respect to these and the three primary planes.



Once you have selected a plane you will see two red arrows appear on your screen.



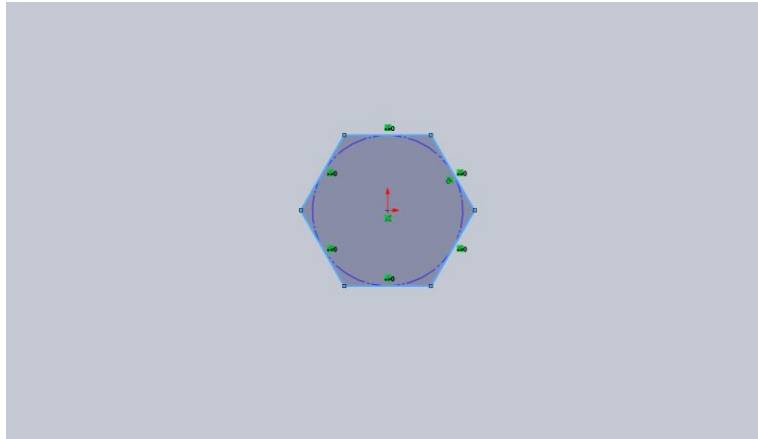
This represents the origin for the selected plane. It will be useful to define relations of this point. We will begin by making a simple hexagon in the 'Front Plane'. To do this we can use the many shapes from the shape library in the top toolbar.



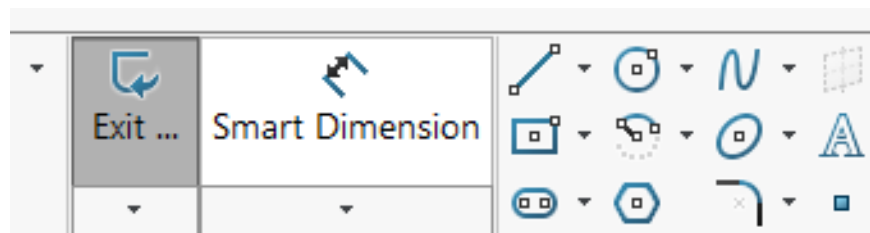
From top left we have,

- lines,
- circles,
- splines curves (which are not commonly used unless you are trying to create something aerodynamic or curved),
- rectangles/squares,
- arcs,
- ovals,
- stadiums,
- hexagons.

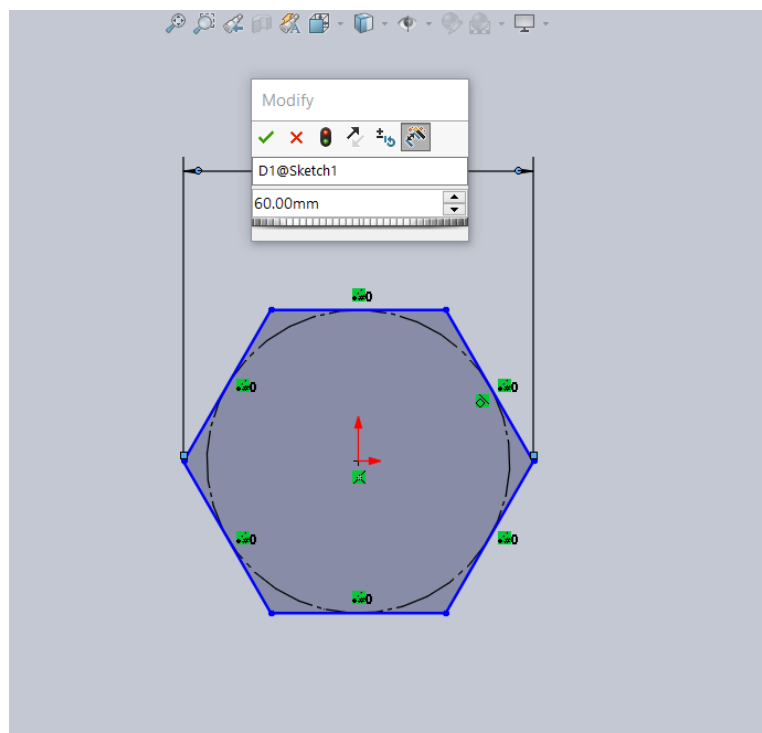
With these drawing tools and a combination of linear, circular patterning, and features (which will be touch on very shortly), almost any geometry can be generated, potentially any geometry. To sketch a hexagon first select the tool by clicking on it then come down to the display window and place its center at the origin this defines a coefficient relationship between the origin and the center of the hexagon so it cannot move.



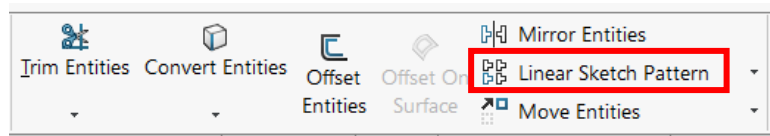
We may wish to define the dimensions of the shape to do this we will use the smart dimension tool.



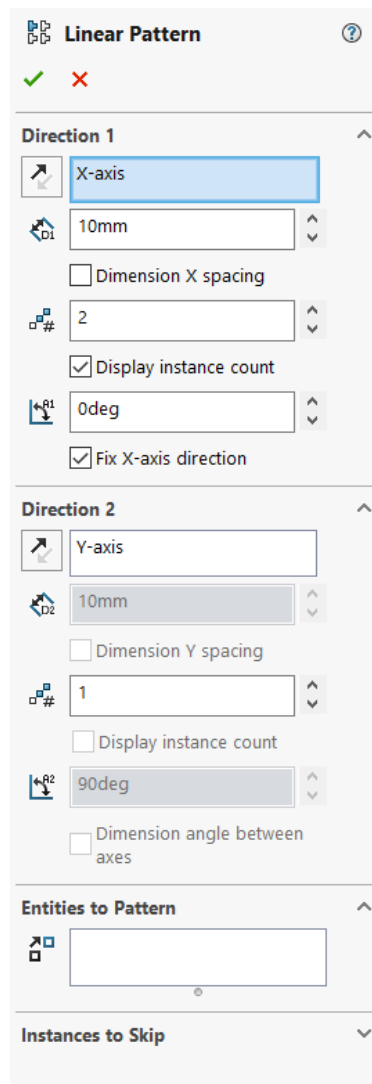
Select the tool then select the two vertices of the hexagon and define it as 60 mm.



We can easily create duplicates of this sketch by selecting linear sketch pattern in the top toolbar of the sketch menu.

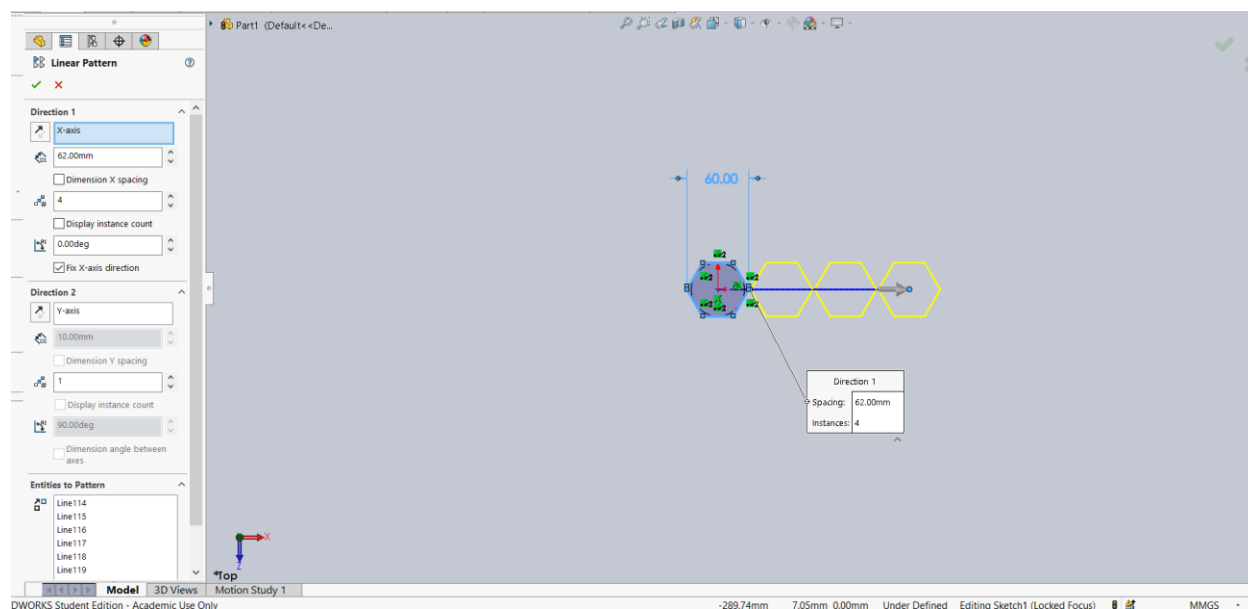


Once this option is selected multiple parameters appear in the left menu,

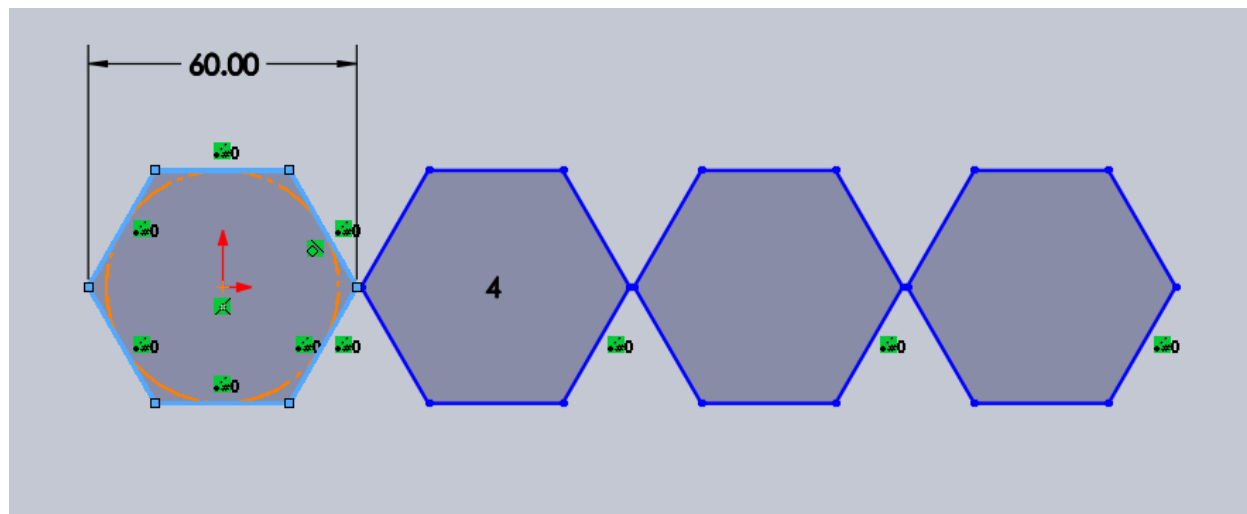


From top to bottom we have 3 main options Direction 1, Direction 2, and Entities to Pattern. Direction 1 can be selected based of a sketch line or construction line. The distance between each copy is given next then the number of instances you wish to create followed by the angle from the axis. Selecting the

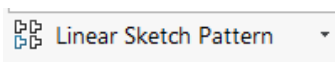
hexagon as the entity to pattern then creating 4 instances in the x-direction with a spacing equal to the width of the hexagon plus two yields the following result.



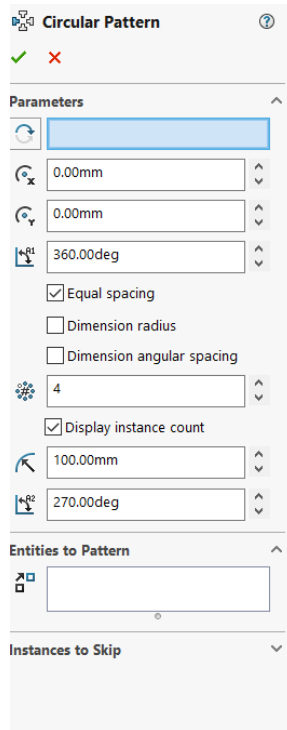
We may also wish to pattern vertically also, to do this just increase the instance count in Direction 2. Click the green check in the corner to generate these sketches show below. Sketches are defined in blue lines. The green squares are the lines relations to each other.



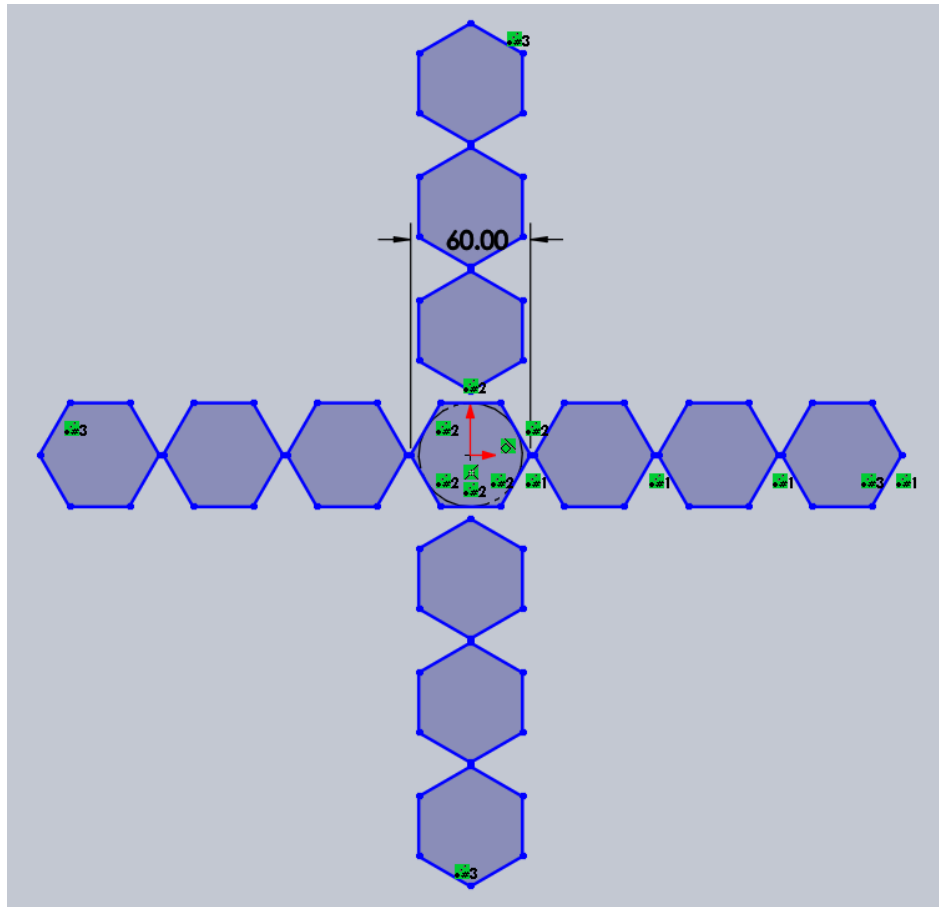
Moving on we shall look at circular patterns then move into features. In the same spot you selected the linear pattern tool there is a small drop down to obtain more patterning tools.



Select the circular pattern tool and it will bring up the following menu.

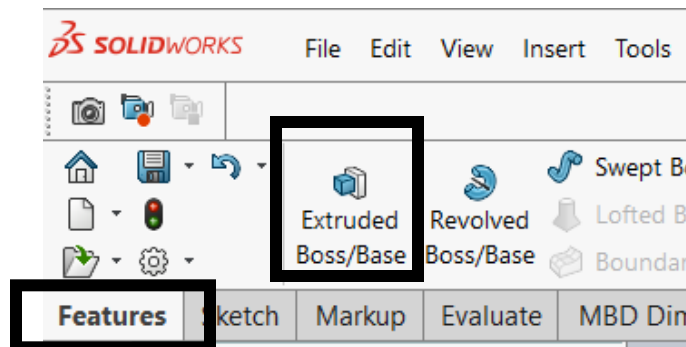


The only difference is now an angle and the origin as well as being able to adjust the radius of the circular pattern and the angle. Leaving all settings and selecting the previously patterned hexagons generates the following pattern. You will have to deselect the center hexagon from the contours, so it is not patterned as well.

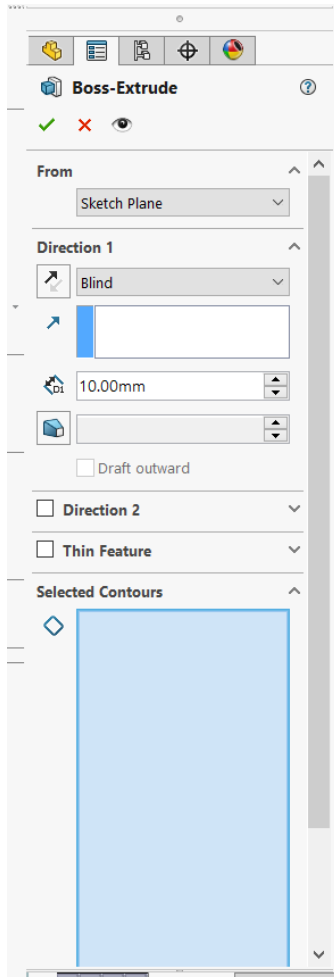


Features

Now to extruding the sketch we just made. Go to Feature up in the top toolbar, and click extrude boss base.

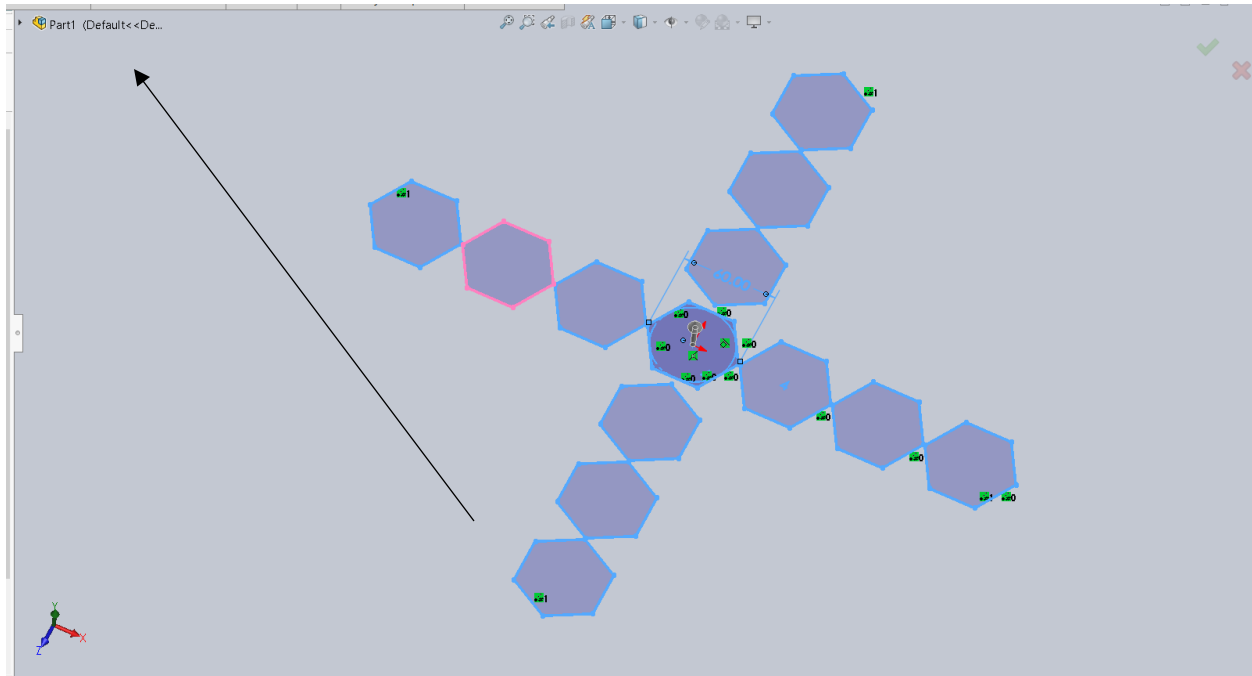


After you click this another menu will appear on the left of the interface.

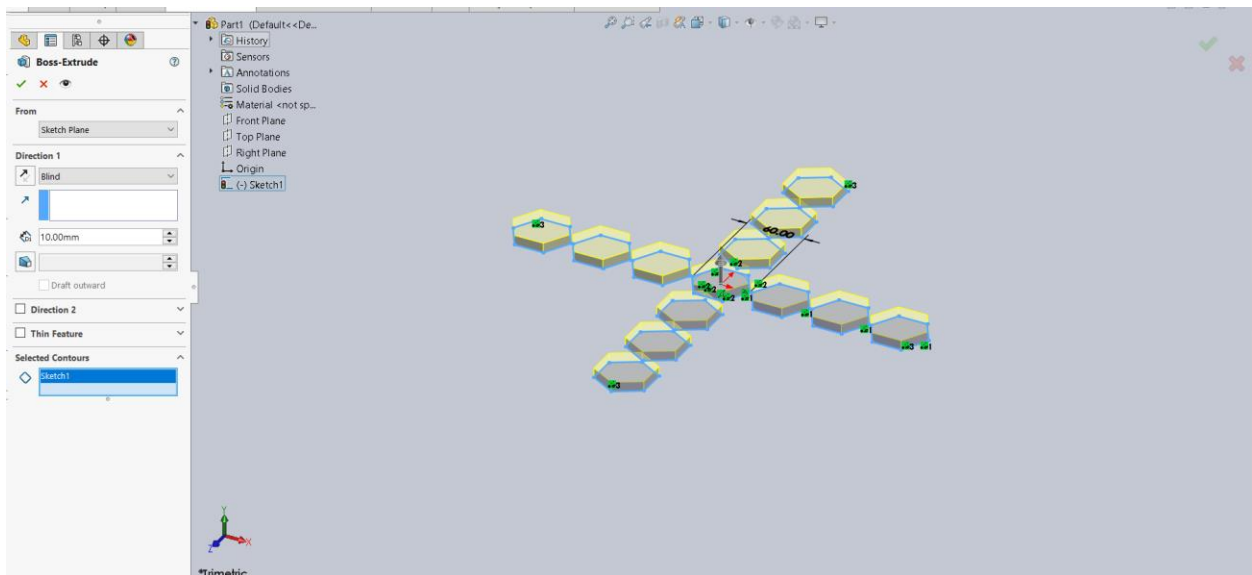


Now you must choose the length that the object will be extruded under Direction 1. You must also select the contours to be extruded.

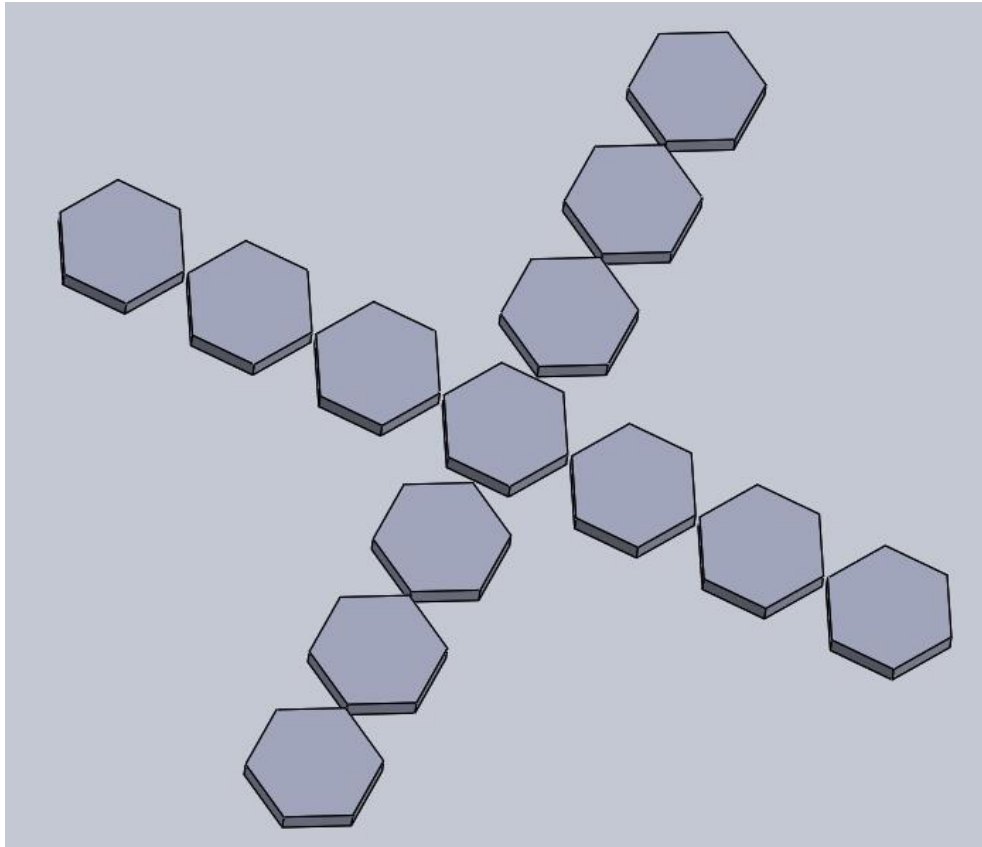
You have two options for selecting the contours. First is to go through and select each hexagon individually, making sure you have selected the “Selected Contours” box before hand (i.e. it has turned blue like above). This method is slow and very tedious for large linear and circular patterns. The better method would be to go into the design tree located in the upper left corner of the design menu shown below.



Opening this menu well u still have the contours box opened you will see the one and only current sketch in your part. Select this sketch and you will see a preview of the extruded bodies.

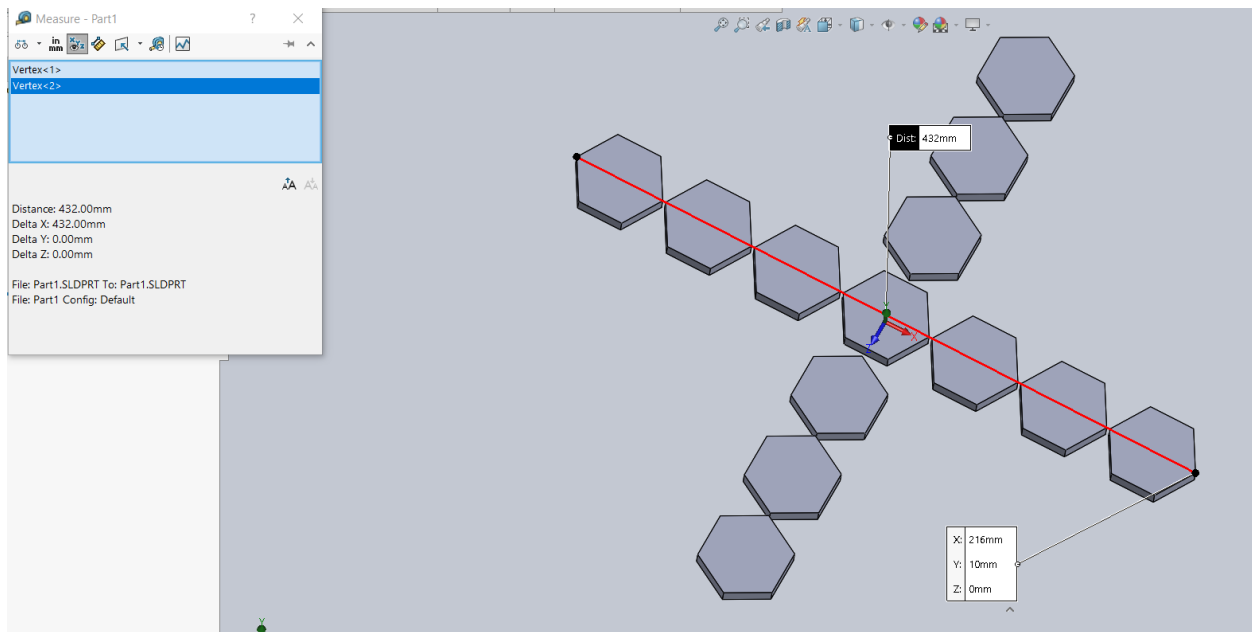


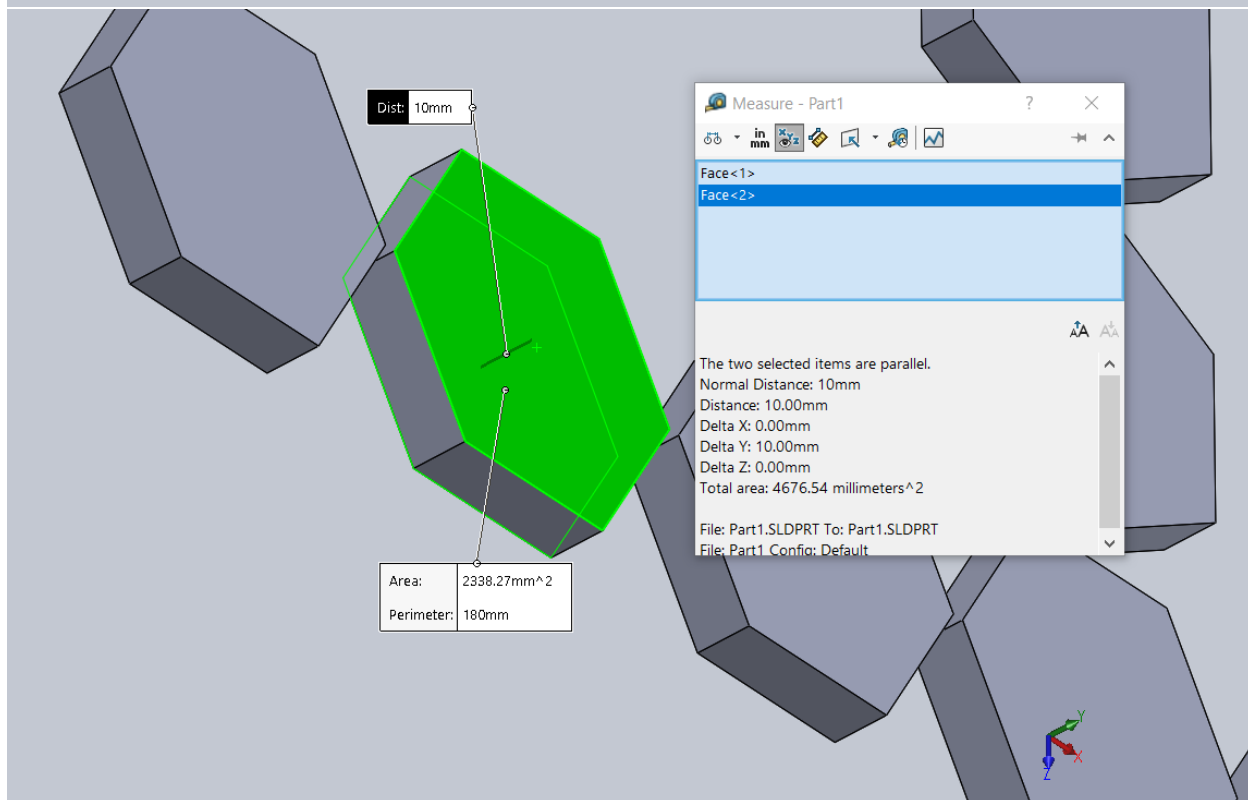
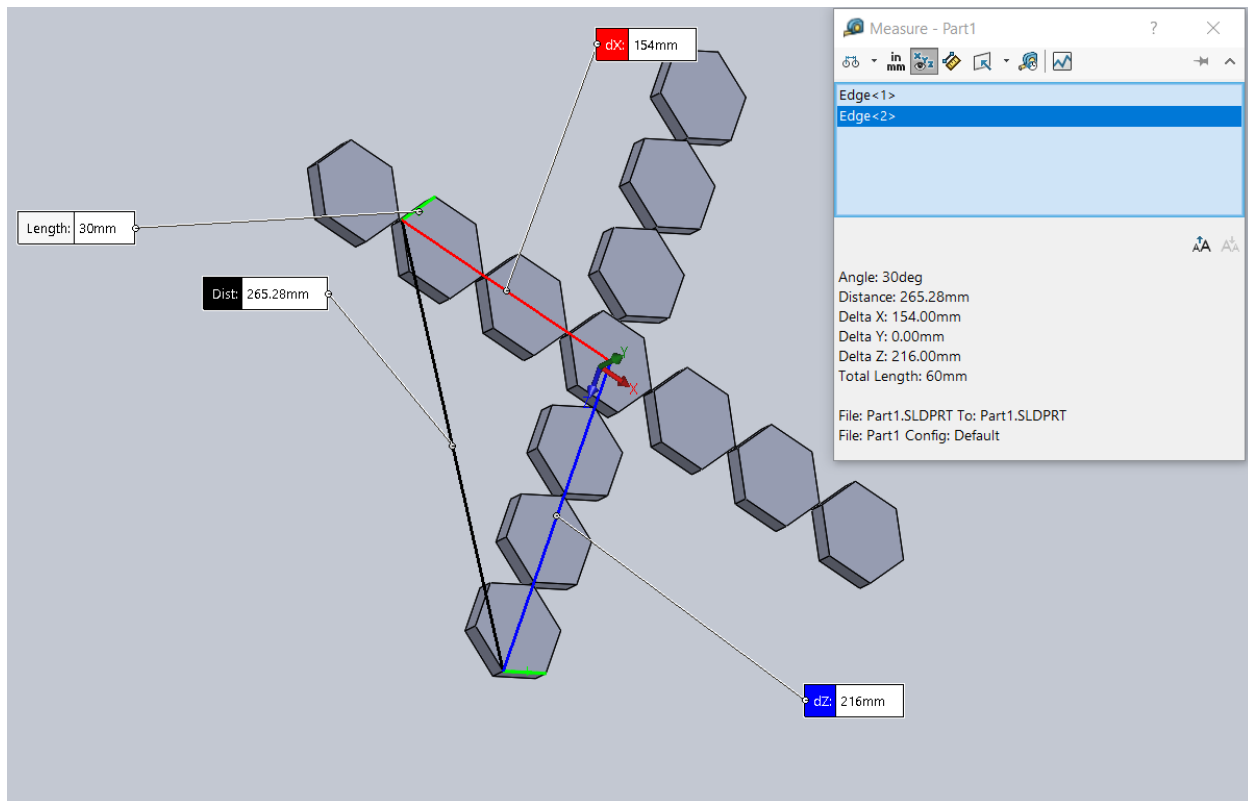
There exist other methods of extrusion such as revolving. This can be used to make circularly symmetrical objects with a single sketch. Many of the options given to sketches can be also done to features or bodies. Things like linear and circular patterning.



Evaluate-Measure

A handy tool in the Evaluate menu on the top toolbar is the measure tool. This allows you to determine measurements within a model that you didn't need to define to create the object. The measure tool can measure between two points, lines, and planes/surfaces.





File-Pack and Go

Quick mention of this feature. Pack and Go allows you to save solidworks files in such a way that other people can open them with a very limited chance of issues. It can be found in the very top tool bar under File-Pack and go,

0 1 0 4 Total: 5 Select / Replace...

☐ Save to folder: C:\Users\abdla\OneDrive\Documents Browse...

☒ Save to Zip file: Browse...

☐ Add prefix:

☐ Add suffix:

☐ Email after packaging

☒ Flatten to single folder

☐ Flatten to minimal folders

☐ Keep full folder structure

Save Cancel Help

Make sure to select the save to zip file option and you should be good to go!

Appendix of Hotkeys:

File Hotkeys	
Ctrl + N	New
Ctrl + O	Open
Ctrl + D	Makes new drawing from part or assembly
Ctrl + A	Makes new assembly from part or assembly
Ctrl + S	Save

Edit Hotkeys	
Ctrl + Z	Undo
Ctrl + Y	Redo
Enter	Repeats last command
Ctrl + X	Cut
Ctrl + C	Copy
Ctrl + V	Paste
Delete	Delete
Ctrl + B	Rebuilds
Ctrl + Q	Forced rebuild
Ctrl + R	Redraws screen
S	Opens shortcut bar
Ctrl + click items	Selects multiple items
Shift + drag	Moves item

View Hotkeys	
Arrow keys	Rotates model
Ctrl + arrow keys	Pans model
Ctrl + R	Redraws the screen
Spacebar	Orientation
F	Zooms to fit
Shift + Z	Zooms in
Z	Zooms out