Basics of SolidWorks Tutorial
This is your screen when you start a new part
The Design Tree lets select and modify sketches and features.

Units can be changed in the bottom-right corner.
To begin designing your part, select a plane using the Sketch tool.
Tools within the Sketch Tab

- Dimensions
- Modify Lines and Create Patterns
- Lines, Curves, and Shapes
Start with basic shapes using the line, rectangle, circle, and arc tools.
Shift-click on lines to add constraints
Use the Fillet tool to round your corners
Using the Smart Dimension Tool, add measurements to your lines
Under the Features Tab, select Extruded Boss/Base to add depth to your sketch.
Tools within the Feature Tab

Extrusion Tools used to add volume from sketches

Cutting tools used to remove volume

Specialty Tools
Use the Fillet Tool to round the edges
Right-click on a surface to begin a new sketch
Use the view selector to rotate the part for a better view
Use the Convert Entities tool to add lines to the sketch surface
You can use Trim Entities to remove unwanted lines
Fillet the corners and add dimensions to the sketch.
Use Extruded Cut to remove part of the object
Select the Chamfer tool under Fillet
Using the chamfer tool, select either the face, or individual lines.
Create a new sketch
Use the Text tool to write your name on the part

Don’t forget to uncheck this box
Select Font to adjust the style and size of your text
Use Extruded Boss/Base to add depth
To add a picture into your sketch, use the Sketch Picture Tool.
Resize and place your image

The Cube
Trace over your image using lines, arcs, and shapes
Then extrude the sketch.
Your finished part should look something like this
Save a copy of your part as a .STL file for 3-D Printing