

SolidWorks Protocol

- Open SolidWorks, CTL+N to open a new Part, save as .SLDPRT
- In the toolbar on the left, select the plane where you wish to begin your drawing (Front, Top, or Right plane)
 - If you want to design your part from the bottom-up, choose Top Plane
- **Begin drawing (2D)**
 - Note: start with an initial shape, then add more shapes to “build” your 3D part
 - Sketch Toolbar > Sketch > pick shape > draw > green checkmark to finish
- **Add a third dimension to the 2D shape**
 - Make sure the sketch is highlighted (blue) in the toolbar on the left
 - Features Toolbar > Extruded Boss/Base > set End Condition to Blind > type in desired height > green checkmark
- **Make adjustments to the sketch**
 - In the toolbar on the left, select the sketch you want to edit (you may need to click on + to expand the shape)
 - Sketch Toolbar > Sketch > Sketch Dimensions > click on sides/points > type in the desired size (in mm) > green checkmark > Exit Sketch
- **Add protrusion**
 - Click on surface on which you wish to draw > Sketch Toolbar > Sketch > pick shape > draw on the initial shape’s surface
 - Features Toolbar > Extruded Boss/Base > type in desired height > green checkmark > (exits the sketch automatically)
- **Add hole**
 - Click on surface on which you wish to draw > Sketch Toolbar > Sketch > pick shape > draw on the shape’s surface
 - Features Toolbar > Extruded Cut > set End Condition to Blind > type in desired depth > green checkmark > (exits the sketch automatically)
 - OR: set End Condition to Through All > automatically cuts through entire shape
- **Move sketches**
 - In the toolbar on the left, select the sketch you want to move > Sketch Toolbar > Sketch > Move Entities > Select all parts of the sketch > From/To > Select Start Point > Click where you want to move the sketch > green checkmark > Exit Sketch
 - OR: Select all parts of the sketch > X/Y > type in the distance in the X and Y direction you want to move your sketch
 - You can also **Copy, Rotate, and Scale sketches**
- **Adding Fillets**
 - Features Toolbar > Fillet > click on any edge(s) on the shape (inside or outside) > adjust radius > green checkmark
- **Viewing your sketch**
 - Zoom in/out by scrolling on the mouse
 - Press space bar to set the orientation of the part automatically
 - Press the scroll button on the mouse and drag to change the orientation manually
- Note: save as .STL file before opening the file in MakerBot

Convert SolidWorks part into a print file in MakerBot

- Open MakerBot > Add File > Open .STL file
- **Lay part flat**
 - Click on part > Press T (change rotation) > Click on +90 until the shape is face up > Press Lay Flat
- **Move part to control where it's printed**
 - Click on part > Press M (change position) > click on the object and drag to desired position
- **Change Settings**
 - Click Settings > High quality > Infill Density at 10% > Make sure Raft is checked off > OK
- **Export**
 - Export Print File > Export Now > Save as .MAKERBOT file on a flash drive (make sure the file name isn't too long)
- Plug in flash drive into 3D Printer and set up to print