

MEEG202 LABORATORY SESSION - SolidWorks 'Sketching'

Basic Sketching: Lines and Arcs

Purpose:

The purpose of this lab is to demonstrate the basic functionality of the SolidWorks software. This includes opening the software, beginning a new drawing, and utilizing basic sketching functions such as creating lines, arcs, and simple extrusions.

Opening software:

Open the SolidWorks software; this should bring you to a blank screen with several menu bars at the top. To start a new drawing, go to the windows menu bar and select *File/New*. This should bring up a menu with several tabs and template options. Click on the tab that says Part (and it would be a good idea to be in Novice and not Advanced mode) and then select OK.

Basic notes on the SolidWorks interface:

Upon opening a new drawing, the following should be visible on the screen:

- A blank model space containing nothing but a set of axes. This is where the actual model is created.
- Several toolbars. The top most toolbar is the standard windows toolbar which can be used for all standard windows functions (File, Edit, Tools etc), as well as SolidWorks functions, options menus etc. Of most significance is the SolidWorks function toolbar. This includes the most commonly used design tools for creating geometric parts and is split into two subsets: *Features and Sketch*. Selecting one or the other (clicking on one of the two left-hand buttons on the SolidWorks function toolbar) opens a submenu to the right containing all the most commonly used tools for that subset.
 - Sketching: Basic sketching tools such as line, circle, rectangle, spline, offset, trim, etc.
 - Features: Used to transform 2d sketches into 3d solid geometries through extrusion, revolution, sweeping, subtracting, etc.
- A history window on the left-hand side of the screen (laid out in an explorer format) which contains all the information directly related to the design of the part itself. This includes the material used, principal planes, all sketches, features, etc. that have been used in creating the part. This window allows you to, at any point in the design process, go back and make changes at a particular stage of the design without having to step backward undoing many processes which may have already been applied.

Beginner sketching and part creation:

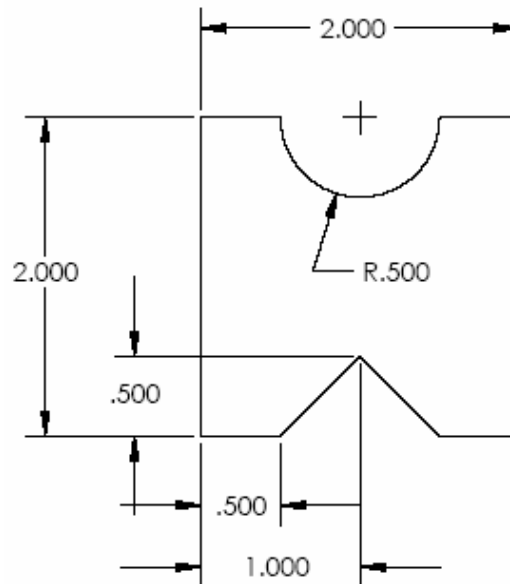


Figure 1:

1. **Sketch:** Having SolidWorks open with the basic Part template, start by making sure YOU know what some of the defaults are! Go to **Tools/Options** and then choose the tab that says Document Properties. Under Units choose the IPS radio button (inch, pound, second) for now; then OK. Now, make sure you select the **Sketch** library function on the left-hand side of the SolidWorks function toolbar. If it is not already open, you will see the function toolbar change to basic sketching commands (*line, rectangle, circle, arc, etc...*)
2. **Lines:** Select the **line tool** with your mouse and you will see the three principal planes show up in the model space. Select the one labeled “front” as the plane you would like to sketch on. With the **line tool** active, click on the origin to start drawing lines. Begin roughly sketching the geometry as shown above, ignoring the dimensions for now. You will notice that the line tool remains active, allowing you to place several lines in sequence until you either close the geometry, deselect the line tool, or hit the escape key on your keyboard (hitting escape will cancel any function/tool currently active). Also, if you are careful, you will notice that the software has built in relationship functions that will automatically snap a line horizontally, vertically, perpendicular to another etc. These can be very useful in roughly sketching out a geometric feature and can be added or subtracted manually if need be.
3. **Arcs:** To draw the arc section, select the **tangent arc** command in the Sketching menu and then proceed by picking the point on the drawing where you would like to begin the arc. (Hint: SolidWorks has an arc function built into the line command which you may find useful as well. With the line tool active you can place your first line point, draw the cursor away from the point, and then ‘mouse over’ that point again. If done correctly the software will automatically enable the **tangent arc** command until the second point is placed.)

4. **Dimensions:** Once you have roughly sketched out the geometry above, add all the “driving” dimensions shown above. Begin by first selecting the *smart dimension* tool in the Sketch toolbar. To dimension an edge, circle, or arc you can simply pick the edge and drag the dimension outward. You can also place a dimension by selecting the end points in which you want to dimension. Once you have placed a dimension, a text box will appear in which you can enter the desired value. Clicking the green checkmark will apply the dimension and SolidWorks will automatically update the geometry to match the dimension. You can change a dimension at any time by double-clicking on it and entering another value in the text box. Basically you want to continue dimensioning until the part is “Fully Defined” - we’ll talk much more about this! Note: how YOU created the geometry and how YOU lay out the dimensions will determine what is needed to fully define the part: in solid modeling (as in manufacturing), how the part is ‘made’ has implications!
5. **Simple Extrusion:** Up to this point, only the Sketch function library has been used. Once the geometry is defined as in Figure 1, select the *Features* command on the left-hand side of the SolidWorks toolbar. This should bring up a new menu of functions to the right which correspond to various features that can be created from the sketched geometry. Select the *Extruded boss/base* button on the Features toolbar, and you will see the history window on the left-hand side of the screen change to a menu box containing various submenus. In the submenu labeled “Direction 1” Select *blind* from the drop down menu, and then enter the depth value of **2.00 in.** in the “D1” depth box. A preview of the extrusion should appear in the model space, and can then be applied by clicking the green checkmark in the top corner of the Extrude menu. If done correctly the result should look somewhat similar to the following:

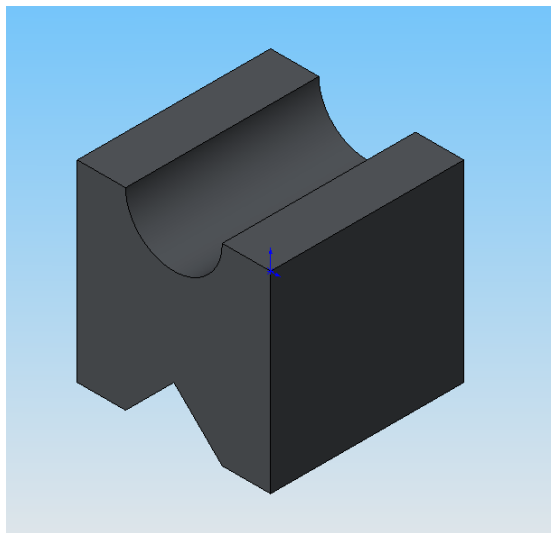


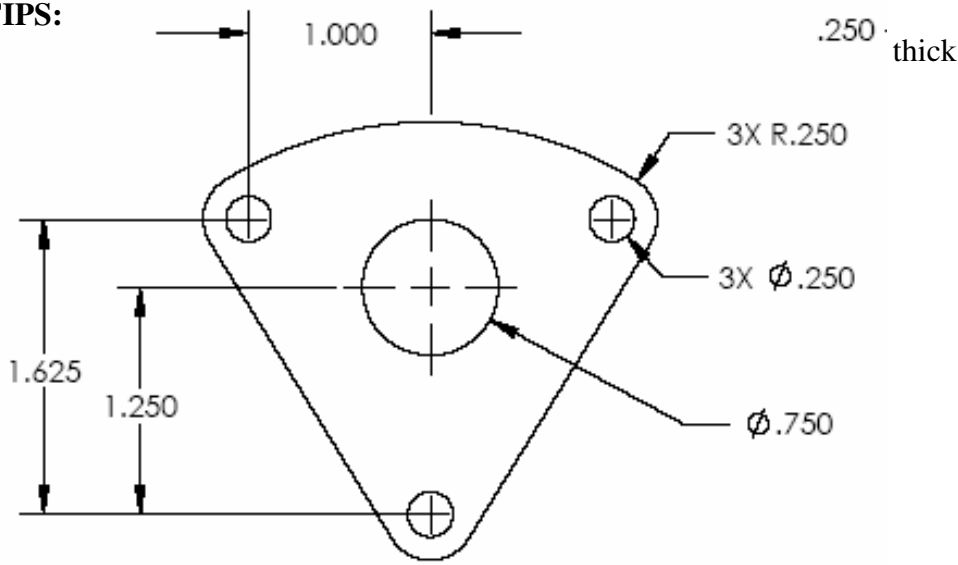
Figure 2

6. **Adding your Name and printing:** To add your name select *Insert/Annotations/Note* in the Windows toolbar. A text box will appear in which you can type your name (use the mouse and click in the drawing to ‘finish’ typing, then Esc to leave this command). To print, first select *File/Print Preview* to make sure it is OK and then *Print* (or back to *File/Print*).

Fully Defined Geometries

Please construct the following part in SolidWorks making sure that the geometry is fully defined. The part is symmetric about the vertical centerline through the 0.75 diameter hole and is 0.25 inches thick. Dimensions are in inches as in the previous two lab examples and should be displayed to three decimal places. Hand in a 3D isometric representation labeled with your name by the end of the class period.

TIPS:



- Under-defined geometric features will appear in blue in the model space and require more relationship/dimensional constraints; fully defined geometry will appear as black. There are two key ways in which to properly define geometric features in SolidWorks.
 - Adding dimensions using the smart dimensioning tool.
 - Adding relations between various features (e.g. parallel, perpendicular, collinear, midpoint, concentric, equal, etc.) which can be accomplished by selecting the **add relations** button in the sketching toolbar. Then simply select the two entities you wish to relate and then the relation you wish to apply in the menu on the left-hand side of the model space.
- Sketching tools you may find useful include **Trim**, **Offset**, and **Add Relations** which can all be found on the sketching toolbar. (The sketching toolbar is often too big to fit on the screen in which case click the >> button on the right-hand side of the toolbar to view the cut off options.)

Please submit a printout of your (final) result - with your name - at the end of the laboratory session. If you do not finish in time, please still submit a printout of your progress by the end of lab - it MUST include your name.