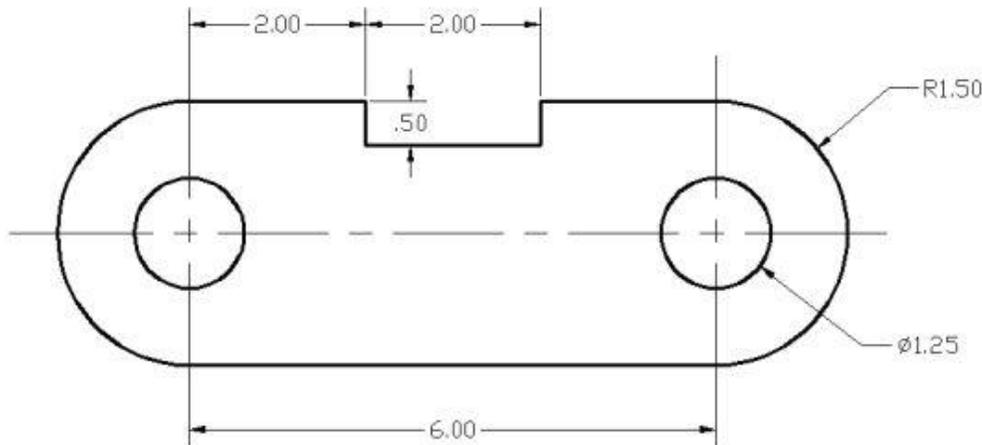


The Spacer Design

We will next create the spacer design using more of AutoCAD's drawing tools.



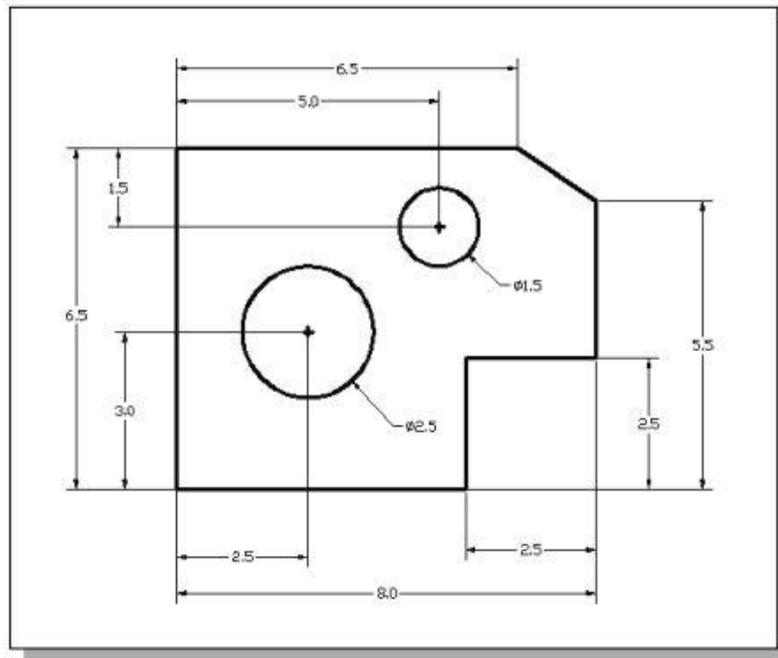
1. In the Application Menu, select [New] to start a new drawing.
2. The Select Template dialog box appears on the screen. Click Open to accept the default acad.dwt as the template to open. The dwt file type is the AutoCAD template file format. An AutoCAD template file contains pre-defined settings to reduce the amount of tedious repetitions.
3. In the Menu Bar select: [Format] \diamond [Units]- The AutoCAD Menu Bar contains multiple pull-down menus where all of the AutoCAD commands can be accessed. Note that many of the menu items listed in the pull-down menus can also be accessed through the Quick Access toolbar and/or Ribbon panels
4. Click on the Length Type option to display the different types of length units available. Confirm the Length Type is set to Decimal.
5. On your own, examine the other settings that are available.
6. In the Drawing Units dialog box, set the Length Type to Decimal. This will set the measurement to the default English units, inches.
7. Set the Precision to two digits after the decimal point as shown in the above figure.
8. Pick OK to exit the Drawing Units dialog box.
9. In the Menu Bar select: [Format] \diamond [Drawing Limits]
10. In the command prompt area, the message "Reset Model Space Limits: Specify lower left corner or [On/Off] :'" is displayed. Press the ENTER key once to accept the default coordinates
11. Inside the Menu Bar area select: [View] \diamond [Zoom] \diamond [All] The Zoom All command will adjust the display so that all objects in the drawing are displayed to be as large as possible. If no objects are constructed, the Drawing Limits are used to adjust the current viewport.

12. Move the graphics cursor near the upper-right corner inside the drawing area, and note that the display area is updated.
13. In the Status Bar area, right-mouse-click on SnapMode and choose [Snap Settings].
14. In the Drafting Settings dialog box, switch on the Snap and Grid options as shown. On your own, exit the Drafting Settings dialog box and reset the status buttons so that only GRID DISPLAY and SNAP MODE are turned ON as shown.
15. Select the Line command icon in the Draw toolbar. In the command prompt area, near the bottom of the AutoCAD graphics window, the message “_line Specify first point:” is displayed. AutoCAD expects us to identify the starting location of a straight line.
16. To further illustrate the usage of the different input methods and tools available in AutoCAD, we will start the line segments at an arbitrary location. Start at a location that is somewhere in the lower left side of the graphics window.
17. We will create a horizontal line by using the relative rectangular coordinates entry method, relative to the last point we specified: @6,0 [ENTER]
18. Next, create a vertical line by using the relative polar coordinates entry method, relative to the last point we specified: @3<90 [ENTER]
19. Next, we will use the direct input method. First, move the cursor directly to the left of the last endpoint of the line segments
20. On your own, turn the mouse wheel to zoom in and drag with the middle mouse to reposition the display.
21. Use the direct distance entry technique by entering 2 [ENTER].
22. On your own, repeat the above steps and create the four additional line segments, using the dimensions as shown.
23. To end the line command, we can either hit the [Enter] key on the keyboard or use the Enter option, right mouse-click and a pop-up menu appears on the screen.
24. Select Enter with the left-mouse-button to end the Line command
25. Select Enter with the left-mouse-button to end the Line command
26. Pick Erase in the Modify toolbar. The message “Select objects” is displayed in the command prompt area and AutoCAD awaits us to select the objects to erase.
27. Select the vertical line as shown.
28. Click once with the right mouse-button to accept the selection and delete the line
29. Click the down-arrow icon of the Arc command in the Draw toolbar to display the different Arc construction options.
30. AutoCAD provides eleven different ways to create arcs. Note that the different options are used based on the geometry conditions of the design. The more commonly used options are the 3-Points option and the Center-Start-End option
31. Select the Center-Start-End option as shown. This option requires the selection of the center point, start point and end point location, in that order, of the arc.
32. Move the cursor to the middle of the two horizontal lines and align the cursor to the two endpoints as shown. Click once with the right mouse-button to select the location as the center point of the new arc.

33. Move the cursor downward and select the right endpoint of the bottom horizontal line as the start point of the arc.
34. Move the cursor to the right endpoint of the top horizontal line as shown. Pick this point as the endpoint of the new arc.
35. On your own, repeat the above steps and create the other arc as shown. Note that in most CAD packages, positive angles are defined as going counterclockwise; therefore, the starting point of the second arc should be at the endpoint on top.
36. Select the [Circle] \diamond [Center, Diameter] option as shown.
37. Select the same location for the arc center as the center point for the new circle.
38. In the command prompt area, the message "Specify diameter of circle:" is displayed. Specify diameter of circle: 1.25 [ENTER]
39. On your own, create the other circle and complete the drawing as shown.
40. In the Quick Access Toolbar, select [Save].
41. Note the command can also be activated with the quick-key combination of [Ctrl]+[S].
42. In the Save Drawing As dialog box, select the folder in which you want to store the CAD file and enter Spacer in the File name box
43. Click Save in the Save Drawing As dialog box to accept the selections and save the file. Note the default file type is DWG, which is the standard AutoCAD drawing format.
44. To exit AutoCAD 2021, select Exit AutoCAD in the Menu Bar or type QUIT at the command prompt. Note the command can also be activated with the quick-key combination of [Ctrl]+[Q].

The Guide Plate

We will next create a mechanical design using the different coordinate entry methods.



1. Select the Line command icon in the Draw toolbar. In the command prompt area, near the bottom of the AutoCAD graphics window, the message “_line Specify first point:” is displayed. AutoCAD expects us to identify the starting location of a straight line.
2. We will locate the starting point of our design at the origin of the world coordinate system. Command: _line Specify first point: 0,0 (Type 0,0 and press the [ENTER] key once.)
3. We will create a horizontal line by entering the absolute coordinates of the second point. Specify next point or [Undo]: 5.5,0 [ENTER]
4. Note that the line we created is aligned to the bottom edge of the drawing window. Let us adjust the view of the line by using the Pan Real time command.
5. In the Menu Bar area select: [View] \diamond [Pan] \diamond [Real time]. The available Pan commands enable us to move the view to a different position. The Pan-Real time function acts as if you are using a video camera.
6. Move the cursor, which appears as a hand inside the graphics window, near the center of the drawing window, then push down the left-mouse-button and drag the display toward the right and top side until we can see the sketched line. (Notice the scroll bars can also be used to adjust viewing of the display.)
7. Press the [Esc] key to exit the Pan-Real time command. Notice that AutoCAD goes back to the Line command.
8. We will create a vertical line by using the relative rectangular coordinates entry method, relative to the last point we specified: Specify next point or [Close/Undo]: @0,2.5 [ENTER]

9. Left-click once on the coordinates displays area to switch to a different coordinate display option. Click again to see the other option. Note the coordinates display has changed to show the length of the new line and its angle. Each click will change the display format of the cursor coordinates.
10. On your own, left-click on the coordinates display area to observe the switching of the coordinate display; set the display back to using the world coordinate system.
11. We can mix any of the entry methods in positioning the locations of the endpoints. Move the cursor to the Status Bar area, and turn ON the SNAP MODE option Note that the Line command is resumed as the settings are adjusted.
12. Create the next line by picking the location, world coordinates (8, 2.5), on the screen.
13. We will next use the relative polar coordinates entry method; distance is 3 inches with an angle of 90 degrees, relative to the last point we specified: Specify next point or [Close/Undo]: @3<90 ENTER
14. Using the relative rectangular coordinates entry method to create the next line, we can imagine a reference coordinate system aligned at the previous point. Coordinates are measured along the two reference axes. Specify next point or [Close/Undo]: @-1.5,1 [ENTER] -1.5 and 1 inches are measured relative to the reference point.)
15. Move the cursor directly to the left of the last point and use the direct distance entry technique by entering 6.5 [ENTER].
16. For the last segment of the sketch, we can use the Close option to connect back to the starting point. Inside the graphics window, right-mouse-click and a pop-up menu appears on the screen.
17. Select Close with the left-mouse-button to connect back to the starting point and end the Line command.
18. In the Draw toolbar, click on the little triangle below the circle icon. Note that the little triangle indicates additional options are available.
19. In the option list, select: [Center, Diameter]
20. Notice the different options available under the circle submenu: • Center, Radius: Draws a circle based on a center point and a radius. • Center, Diameter: Draws a circle based on a center point and a diameter. • 2 Points: Draws a circle based on two endpoints of the diameter. • 3 Points: Draws a circle based on three points on the circumference. • TTR–Tangent, Tangent, Radius: Draws a circle with a specified radius tangent to two objects. • TTT–Tangent, Tangent, Tangent: Draws a circle tangent to three objects.
21. In the command prompt area, the message “Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:” is displayed. AutoCAD expects us to identify the location of a point or enter an option. We can use any of the four coordinate entry methods to identify the desired location. We will enter the world coordinates (2.5,3) as the center point for the first circle. Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: 2.5,3 [ENTER]
22. In the command prompt area, the message “Specify diameter of circle:” is displayed. Specify diameter of circle: 2.5 [ENTER]
23. Inside the graphics window, right-mouse click to bring up the pop-up option menu.
24. Pick Repeat CIRCLE with the left-mouse button in the pop-up menu to repeat the last command.

25. Using the relative rectangular coordinates entry method, relative to the center-point coordinates of the first circle, we specify the relative location as (@2.5,2).
26. Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: @2.5,2 [ENTER]
27. In the command prompt area, the message "Specify Radius of circle: " is displayed. The default option for the Circle command in AutoCAD is to specify the radius and the last radius used is also displayed in brackets.
28. Inside the graphics window, right-mouse-click to bring up the pop-up option menu and select Diameter as shown.
29. . In the command prompt area, enter 1.5 as the diameter. Specify Diameter of circle: 1.5 [ENTER]
30. In the Application Menu, select: [Application] ◇ [Save]
31. Note the command can also be activated with the quick-key combination of [Ctrl]+[S].

