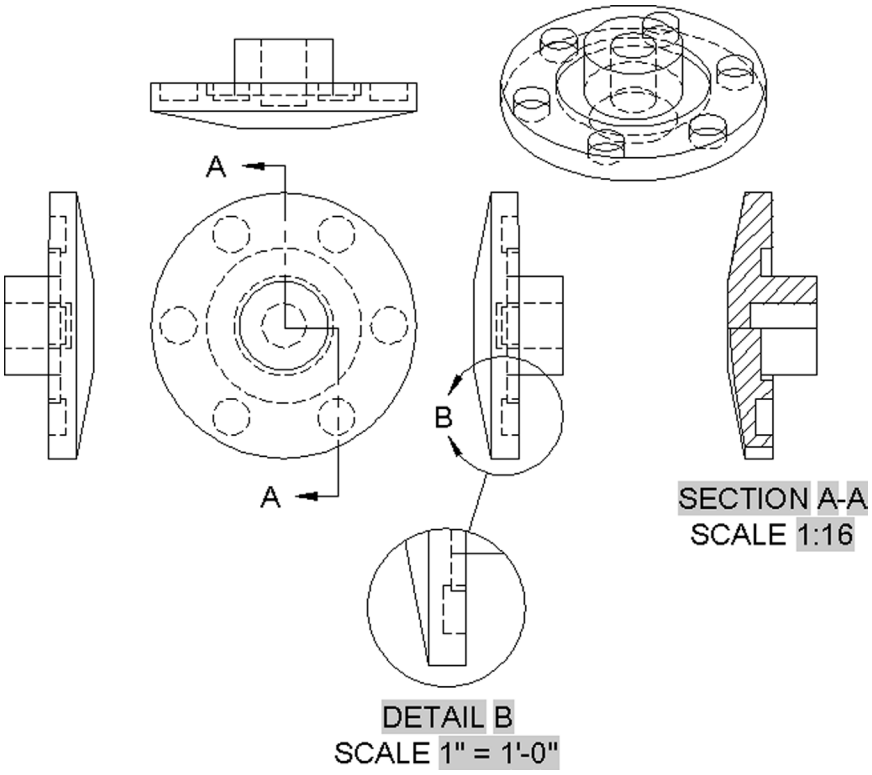
Chapter 27

Slicing, Sectioning, Layouts, and Vports



Learning Objectives

In this chapter, we introduce and thoroughly cover the concepts of sectioning and presenting views. We specifically introduce the following topics:

- Slice
- Section
- Base view
- Projected view
- Section view
- Detail view
- Additional layout options
- Flatten and flatshot
- Vports

By the end of this chapter, you will be able to slice and section 3D models, create complete 2D layouts of the 3D models, and present your model in multiple viewports.

Estimated time for completion of this chapter: 2 hours.

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27.1 INTRODUCTION TO SLICING, SECTIONING, LAYOUTS, AND VPORTS

This is the first of three chapters that deals with the various ways you can detail, present, view, clarify, and interact with your 3D design. This is a broad topic, which covers slicing, sectioning, layouts, and viewports in this chapter, with more discussion on dynamic views, path animation, and many other tools in Chapter 28, Advanced UCS, Views, Text, and Dimensions in 3D and Chapter 29, Dview, Camera, Walk and Fly, Path Animation. Four releases ago, AutoCAD 2013 introduced significant enhancements to create 2D layouts from 3D models (similar to advanced 3D parametric design software), and we cover this in detail after the basics of slicing and sectioning.

27.2 SLICING AND SECTIONING

You already know how to rotate and view your design from various angles, but what if you wanted to get inside of it? The only approach thus far was to revolve a shape less than 360°, revealing the inside. But, what about a nonrevolved design or being able to look inside a fully revolved view? Here, sectioning comes in handy. Section views are used in both engineering and architecture to show the inner workings of mechanical parts and architectural elevations. We focus here on simple mechanical devices, but you can apply the techniques to any design.

To illustrate slicing and sectioning, we need to create a basic shape to use as an example. Create the profile in Fig. 27.1 using a pline, and revolve it 360°. Color and rotate it bottom side up, as seen in the figure. Then, add a circle of 6″ diameter and extrude it—3″ as shown, polar arraying it six times. Explode the array and subtract the pieces out to create the cavities, as seen in Fig. 27.2.

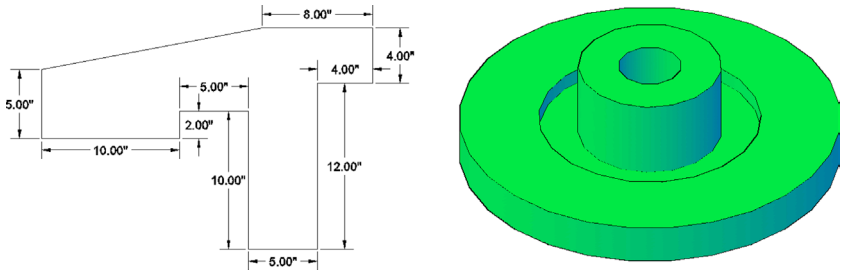

FIGURE 27.1 Profile and revolution.

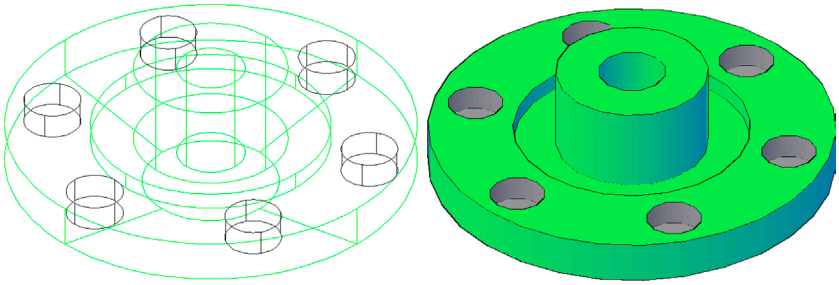

FIGURE 27.2 Add cavity, array, and subtract

Slice

Slice is more basic than sectioning (though technically the same idea), so we start here. The slice command chops off a piece of the part so you can see inside. You can either keep the chopped-off side or get rid of it. We demonstrate both approaches. The most common way to indicate where to slice is via planes (XY/YZ/ZX), but you can also indicate via three points or even use an object as the "slicer."

Keyboard: Type in **slice** and press Enter
Cascading menus: **Modify→3D Operations→Slice**
Toolbar icon: none
Ribbon: **Home tab→Solid Editing**

Step 1. Begin the slice command via any of the preceding methods.
- AutoCAD says: `Select objects to slice:`

Step 2. Select the revolved shape and press Enter.
- AutoCAD says: `Specify start point of slicing plane or [planar Object/Surface/Zaxis/View/XY/YZ/ZX/3points] <3points>:`

Step 3. We use the planes as a slicer. If your UCS is in the standard (World) configuration (as seen in Fig. 27.3), then select `ZX` as the plane.
- AutoCAD says: `Specify a point on the ZX-plane <0,0,0>:`

Step 4. Here, you just need to pick a point where you want the plane to start. It is infinite in both planar directions, so you are just locating an anchor point. Using OSNAPs, select the lower left quadrant (also seen in Fig. 27.3).
- AutoCAD says: `Specify a point on desired side or [keep Both sides] <Both>:`

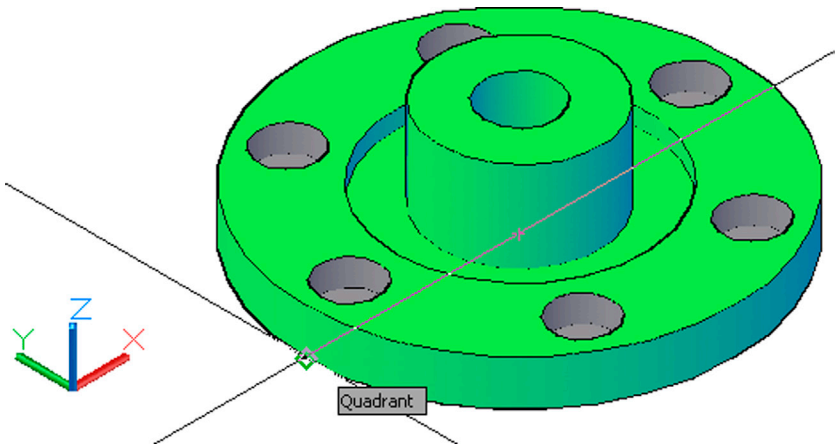

FIGURE 27.3 Select anchor point for plane slice.

Step 5. Here AutoCAD is asking which side of the slice you want to keep. Pick the one at the top of the screen by clicking on it, or in its general area, and you will see one half of the part disappear (Fig. 27.4).

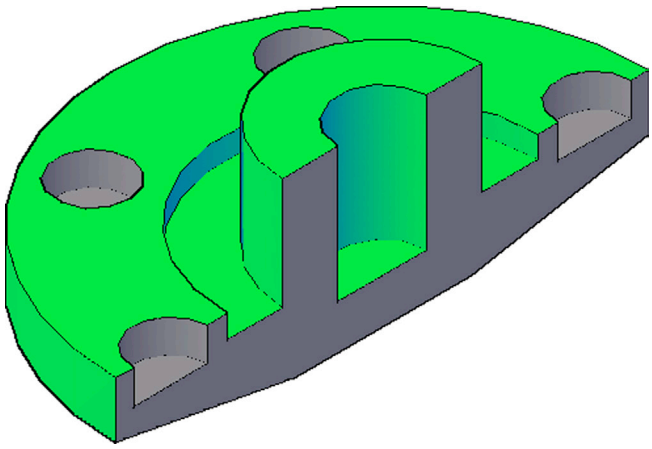

FIGURE 27.4 Completed slice.

If you want to keep both sides and have the slice act literally like a knife and no more, then select `B` for `Both` sides in Step 4. You can then just move one of the pieces off to the side and reveal the inside of both, as seen in Fig. 27.5. Go ahead and undo what you just did and run through the slice command again, keeping both sides this time.

Finally, you can also use an object as a slicer, as seen here with a randomly sized rectangle (turned into a region for clarity) standing in for a plane (Fig. 27.6). Select the `O` for `Object` option in Step 2, and delete one of the sliced sides.

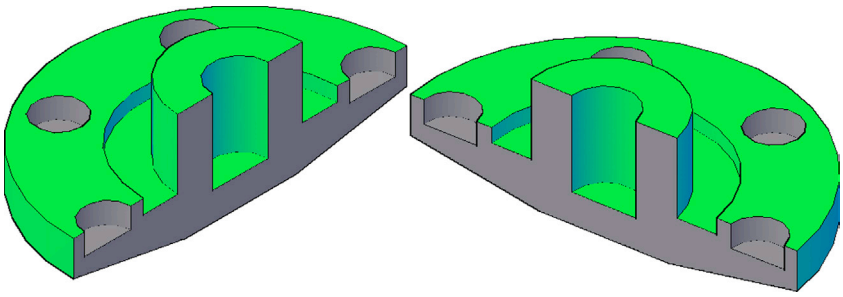

FIGURE 27.5 Completed slice—Both sides option.

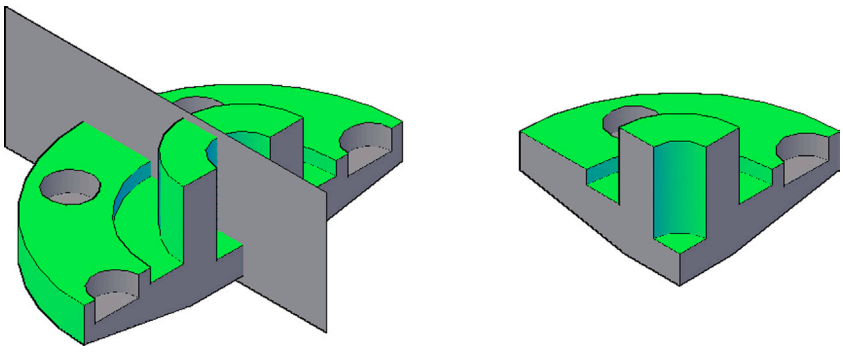

FIGURE 27.6 Completed slice—Object option.

Section

A section is related to the slice. Now, instead of cutting off a piece of the 3D model, we are just creating a cross-section plane. This plane can then be moved to another view and worked on as if it is a model all its own. So the critical difference here is that we are not directly modifying the original part but using it as the basis for something new.

Let us go back to the original unmodified model of Fig. 27.2. We want to create a section in the exact same plane (and using the same basic theory) as with the slice example. We then move the newly created section out of the model and place it somewhere else.

Step 1. This command is available only via typing, so go ahead type in `section` and press Enter.
- AutoCAD says: `Select objects:`

Step 2. Select the 3D model.
- AutoCAD says: `Specify first point on Section plane by [Object/Zaxis/View/XY/YZ/ZX/3points] <3points>:`

Step 3. Select the `ZX` plane as in the previous example with slice.
- AutoCAD says: `Specify a point on the ZX-plane <0,0,0>:`

Step 4. Select the lower quadrant, also similar to the slice example. The section is created. You may have to switch to wireframe to see it, as shown on the left-hand side of Fig. 27.7. On the right-hand side the section is moved outside the 3D part, and both are shaded.

The section itself can now be extruded if needed and of course labeled with dimensions, which is usually the reason behind creating it in the first place: to annotate it for clarity.

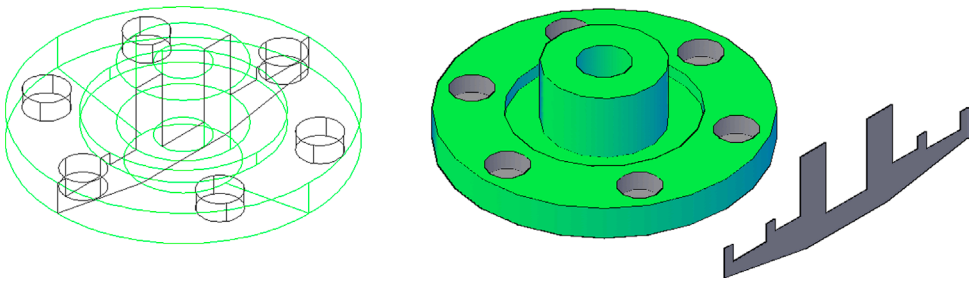

FIGURE 27.7 Completed section.

27.3 LAYOUTS AND DETAILS

Although you have worked with layouts and Paper Space before, this was limited to 2D views of 2D designs. But, what if you have a 3D model and want to quickly create 2D views of it automatically, including sections and enlarged details? This is something that was available for years in high-end design software like CATIA, NX, SolidWorks, and others. AutoCAD finally incorporated this feature in the 2013 release. You can now create instant views, sections, and details after just a few clicks.

This process is relatively simple, and no viewports are involved, but it does follow strict drafting standards and depends on you knowing what these standards and conventions are in order to create a meaningful layout. Use of parent views, projections, sections, and details flows directly from hand drafting. Here is a quick review.

- *Parent View*: This is the view of the model that generates all other views. If it changes in any way, you have to update all the other associated views. AutoCAD, of course, does this for you.
- *Projections*: These can be front, back, top, bottom, side, and isometric in nature. Be aware of how projected views are oriented from the parent view and where they are located. In the commonly used Third Angle projection, a head-on view turns to the right and faces the parent view (relative to you, the observer). Fig. 27.8 demonstrates the convention used. There is also a First Angle projection, where the side view is the main one, not the front.

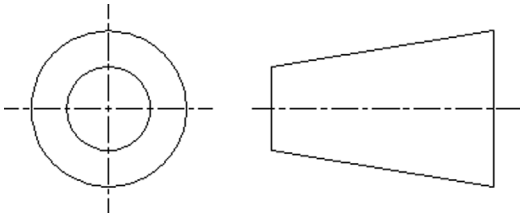

FIGURE 27.8 Third Angle view projection.

- *Sections*: These can be generally classified as straight sections, which run from one end of the part to another, or jogged sections that turn at various angles as they make their way through the part.
- *Details*: These are blowups or enlargements of a part of the model for clarification purposes. A detail can be attached to the view that generated it, or it can be free standing.

With these in mind, we explore the available options and create a full layout of the mechanical part we have been using.

Creating these layouts is done primarily via the Ribbon. There are no toolbars or cascading menus to use. Typed equivalents exist, and they will be shown, but the Ribbon is the simplest way to proceed. Go ahead and select a shaded top view of the same 3D model we have been using. Be sure you are in the 3D Modeling workspace and select the Ribbon's Home tab. You should have what is shown in Fig. 27.9, with the Base function circled at the very upper right of the screen. Be aware that, as of AutoCAD 2014, a Layout tab in the Ribbon has the same function, but that tab is no longer there in the beginning. We encounter it again shortly.

Note: *In AutoCAD 2017, the following commands (VIEWBASE, VIEWPROJ, VIEWSECTION, and VIEWDETAIL) are only available on 64-bit computer systems and will not appear on 32-bit machines. AutoCAD 2016 allowed the commands in both 64 and 32 bit. The commands themselves are essentially the same in both releases and* Fig. 27.9 *is retained from the previous release.*

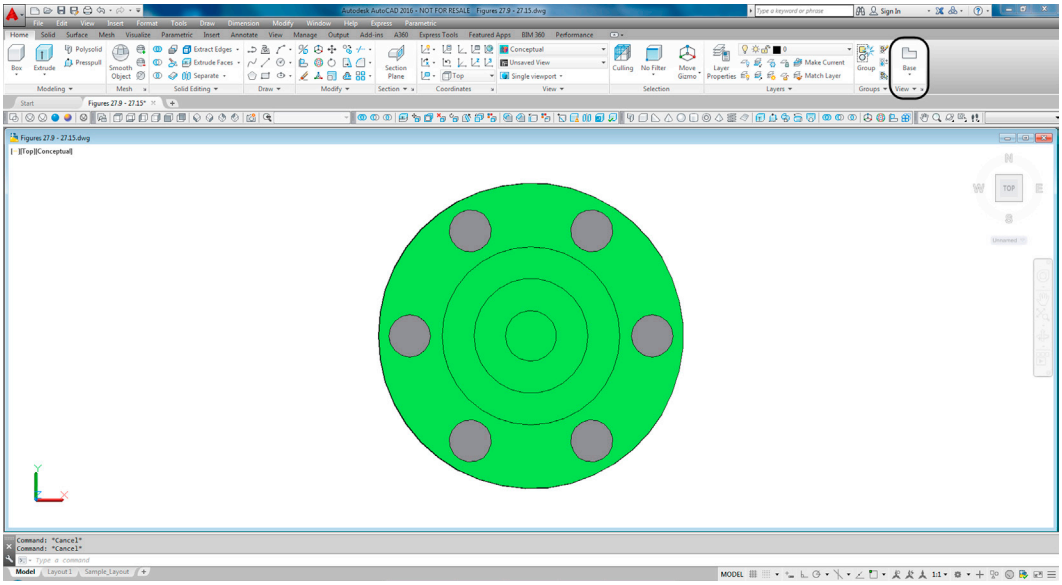

FIGURE 27.9 Setup for Layouts.

VIEWBASE Establishes the Base (Parent) View

Step 1. In the Ribbon's Home tab, View panel, select the Base option, Base View From Model Space.

- AutoCAD says: `Specify model source [Model space/File] <Model space>:_M`
  `Select objects or [Entire model] <Entire model>:`

Step 2. Select the entire model.

- AutoCAD says: `Enter new or existing layout name to make current or [?] <Layout 1>:`

Step 3. Type in `Sample_Layout` or any other name you wish. Press Enter. AutoCAD switches to Paper Space and a new tab with the name you gave it appears. A view of the part is attached to the movements of your mouse.

- AutoCAD says: `Restoring cached viewports - Regenerating layout.`
  `Type = Base and Projected Hidden Lines = Visible and hidden lines Scale = 1:16`
  `Specify location of base view or [Type/sElect/Orientation/Hidden lines/Scale/Visibility] <Type>:`

Step 4. Click to place the view anywhere on the screen.

- AutoCAD says: `Select option [sElect/Orientation/Hidden lines/Scale/Visibility/Move/eXit] <eXit>:`

Step 5. Press Enter, then click again to place another view below the first one

- AutoCAD says: `Specify location of projected view or <eXit>:`
  Press Enter one final time to get two views as seen in Fig. 27.10.
- AutoCAD says: `Base and 1 projected view(s) created successfully.`

Note that you need not stop at just two views; you could continue to add additional views to the right, left, top, bottom, and isometric, as desired. As practice, undo what you just did and repeat all the steps, this time adding an additional isometric view, as seen in Fig. 27.11. Adding additional projections (or removing them) can easily be done at the next step, so just two is adequate in the initial setup, but we keep the isometric projection.

Next, we would like to add (or delete) additional projections. How many and what type need to be shown on a drawing is really up to the engineer or designer and company standards and needs. We establish just a few more projections, leaving enough room for sections and details. These new views inherit the same scale and other parameters of the parent view that you choose.

VIEWPROJ Establishes Additional Projections

Step 1. Select, by clicking once, whichever of the existing projections you would like to use as the "parent" projection in order to generate some new ones. In this example, the front view in the middle is chosen. Once it is clicked on, the Ribbon changes to the Drawing View tab. Select the Projected icon, as seen in Fig. 27.12.

- AutoCAD says: `Specify location of projected view or<eXit>:`

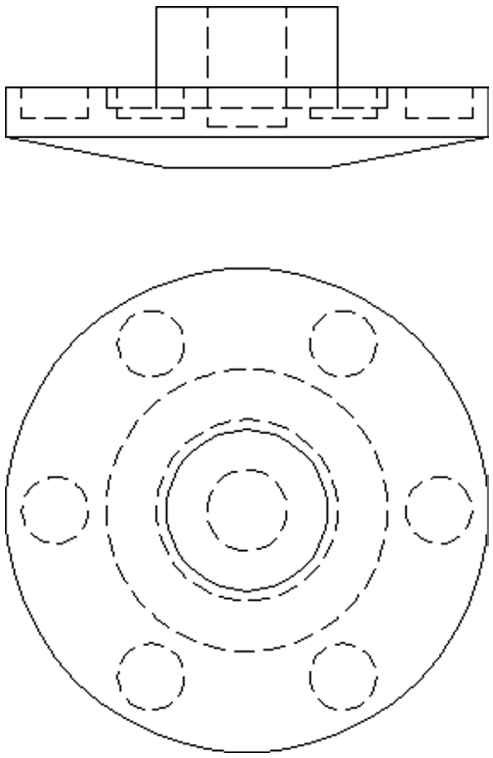

FIGURE 27.10 Base and projected views.

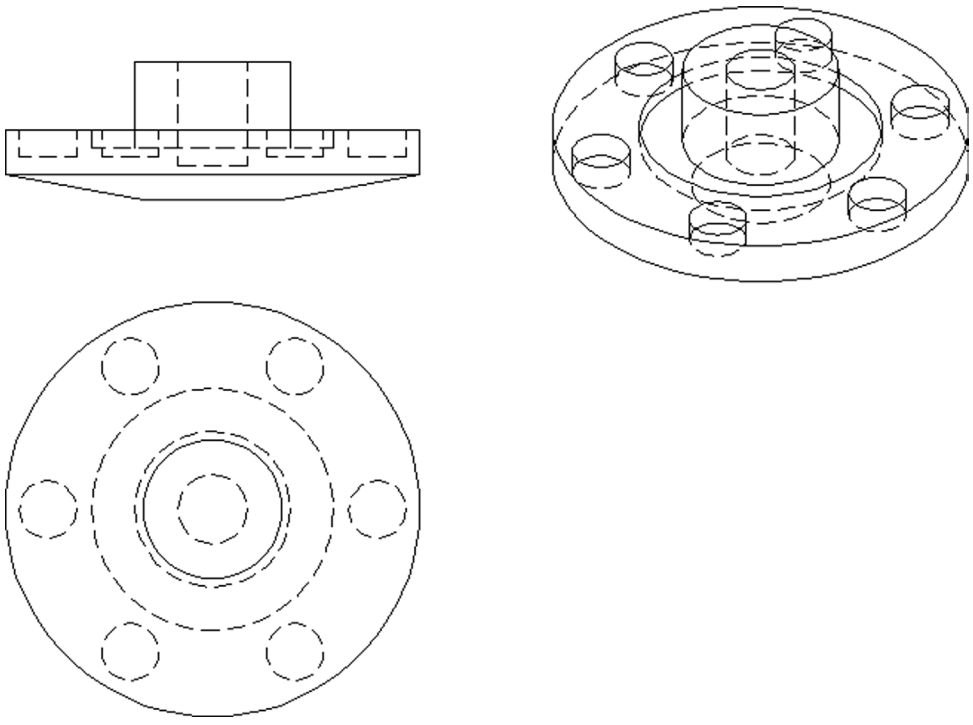

FIGURE 27.11 Isometric view added.

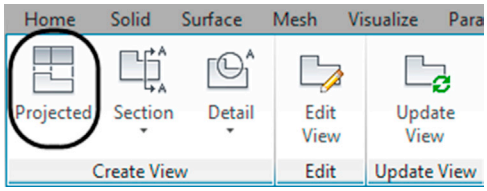

FIGURE 27.12 Drawing View tab, Projected.

Step 2. The new views are attached to the movements of your mouse. Note how it can be located only in certain planes; left right, top, and bottom. Click to place the projections one time to the left and one time to the right of the main view, pressing Enter when done. The result is shown in Fig. 27.13.

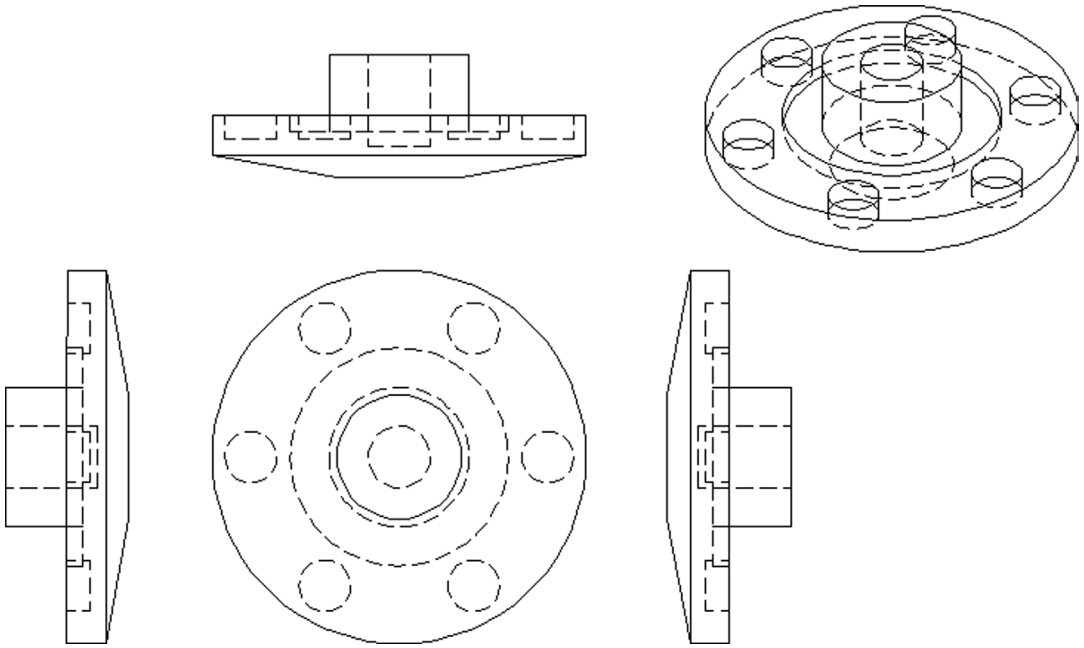

FIGURE 27.13 Additional projections.

Note that, within the proper plane, you can delete or move any of these projected views. The scale of each view can also be adjusted, if needed, by clicking on the view once to reveal a set of grips. The grip that looks like an arrow shows a set of scale values if you click on it again. We need not do this at this time, but it is necessary to know how to, just in case.

Now that you have all the views set up, you may need to add sections and details. The process for adding section views is described next.

VIEWSECTION Creates Section Views

Step 1. Select, by clicking once, whichever view you want to section—the main view in the center in this example. The Ribbon once again changes to the Drawing View tab seen in Fig. 27.12. Select the Section icon, just to the right of Projected. As you drop down the menu, take note of the five types of sections available: Full, Half, Offset, Aligned, and From Object. We try only the Offset one, leaving the rest for you to practice. They all work on the same principle. Click the Offset option and the Ribbon changes to the Section View Creation tab.

- AutoCAD says:
```
Hidden Lines = Visible lines Scale = 1:16 (From parent)
Specify start point or [Type/Hidden lines/Scale/Visibility/Annotation/hatCh]
<Type>:_t
Select type [Full/Half/OFfset/Aligned/OBject/eXit] <eXit>:_of
Specify start point:
```

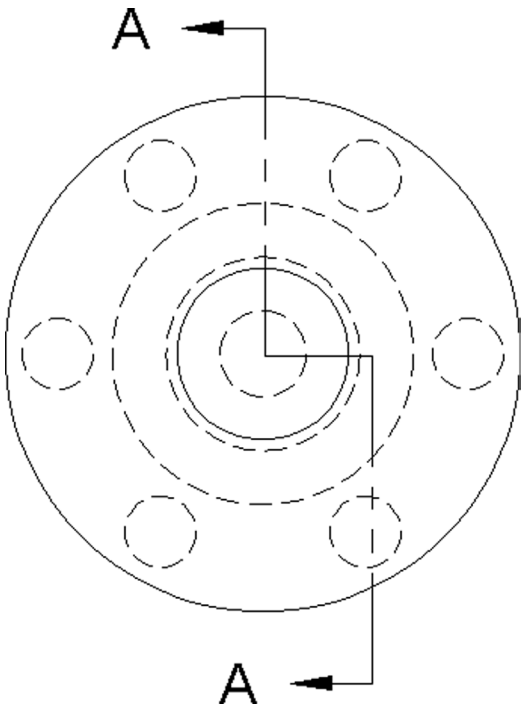

FIGURE 27.14 Offset section.

Step 2. An `A` and an arrow, the start of a section designator, is attached to your mouse movements. Click your way down the main top view, as seen in Fig. 27.14. Note how the section lines pass through several of the cutouts in the model. Seeing those cutouts in the section is usually of interest to the designer, so be sure to trace a path that reveals useful information, not just a random cut.

Step 3. When done press Enter and a section view appears, attached to the motions of your mouse, in plane.

- AutoCAD says: `Specify location of section view or:`

Step 4. Move your mouse off to the right, some distance from the main view, and click to place the section view.

- AutoCAD says: `Select option [Hidden lines/Scale/Visibility/Projection/Depth/Annotation/hatCh/Move/eXit] <eXit>:`

These are additional options, some of which we discuss shortly, but for now, just press Enter to complete the command. Your view loses its shading and retains hatching, as is proper for a section view. Below it, editable text appears, indicating the name of the section and its scale (same as the parent view). The final result is shown in Fig. 27.15.

The final major step involves creating details. As mentioned, these are nothing more than larger views of existing geometry and are primarily for clarification purposes of hard-to-see parts of the design.

VIEWDETAIL Creates Detail Views

Step 1. Select, by clicking once, whichever view you want to section—the bottom of one of the side projections on the right is chosen in this example. The Ribbon once again changes to the Drawing View tab seen in Fig. 27.12. Select the Detail icon, just to the right of Section. As you drop down the menu, take note of the two types of details available: Circular and Rectangular. We try only the Circular one, leaving the Rectangular one for you to practice. They all work on the same principle. Click on the Circular option and the Ribbon changes to the Detail View Creation tab.

- AutoCAD says:

```
Boundary = Circular Model edge = Smooth Scale = 1″= 1′-0″
Specify center point or [Hidden lines/Scale/Visibility/Boundary/model Edge/Annotation] <Boundary>:_b
Select boundary type [Circular/Rectangular/eXit] <Circular>:_c
Specify center point or [Hidden lines/Scale/Visibility/Boundary/model Edge/Annotation]
<Boundary>:
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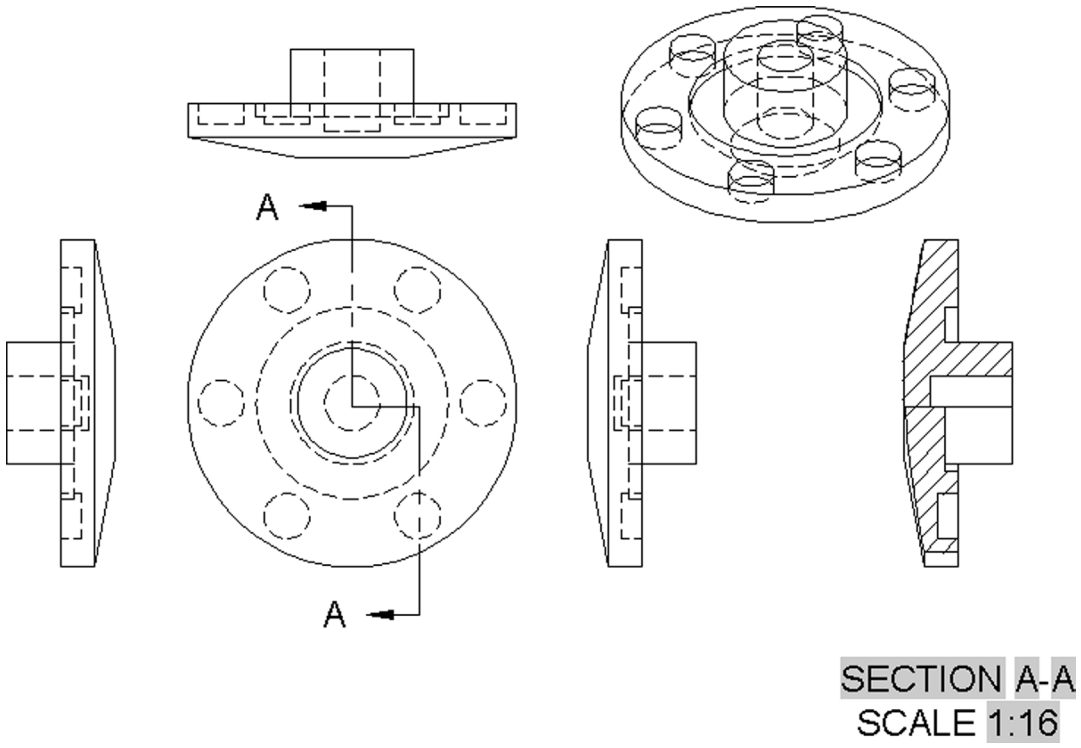


FIGURE 27.15 Offset section.

Step 2. Click somewhere in the lower region of the projection. You see a circular detail marker with the letter `B`.
- AutoCAD says: `Specify size of boundary or [Rectangular/Undo]:`

Step 3. Move the mouse out slowly until the circle is big enough to cover the area you want to detail.
- AutoCAD says: `Specify location of detail view:`

Step 4. As you are positioning the detail, be sure to select the Smooth with connection line under Model Edge of the Ribbon's Detail View Creation tab. Click somewhere below the view—or anywhere there is room.
- AutoCAD says: `Select option [Hidden lines/Scale/Visibility/Boundary/model Edge/Annotation/Move/eXit] <eXit>:`

Step 5. Press Enter, and the detail loses its shading, appearing with some editable text.
- AutoCAD says: `Detail view created successfully.`

The final result is shown in Fig. 27.16.

Additional Layout Options

What we just covered are the essentials of creating 2D layouts from a 3D model. Note, of course, that this should all take place within the confines of a title block; something that was omitted for clarity in the preceding discussion. You first want to create or import the title block into Paper Space, then use the available room to properly position all the views, sections, and details. This can sometimes take considerable effort to position well for maximum clarity.

Of course, we glossed over a few other options. These mainly have to do with appearances of the views. So far, they have appeared as basic linework, with hidden and visible lines. You can easily change the appearance to include shading, hidden lines, and visible lines. This is done by selecting, by clicking once, any of the views. The Ribbon then changes to the Drawing View tab. Select the Edit View icon under the Edit panel. The Ribbon then changes to the Drawing View Editor, and you can select from several hidden, visible, and shaded choices, such as seen in Fig. 27.17. When done press OK, or Cancel if it is not what you wanted.

The result is shown in Fig. 27.18. Take some time to look over all the options, including ones for modifying the text that appears with the new sections and details.

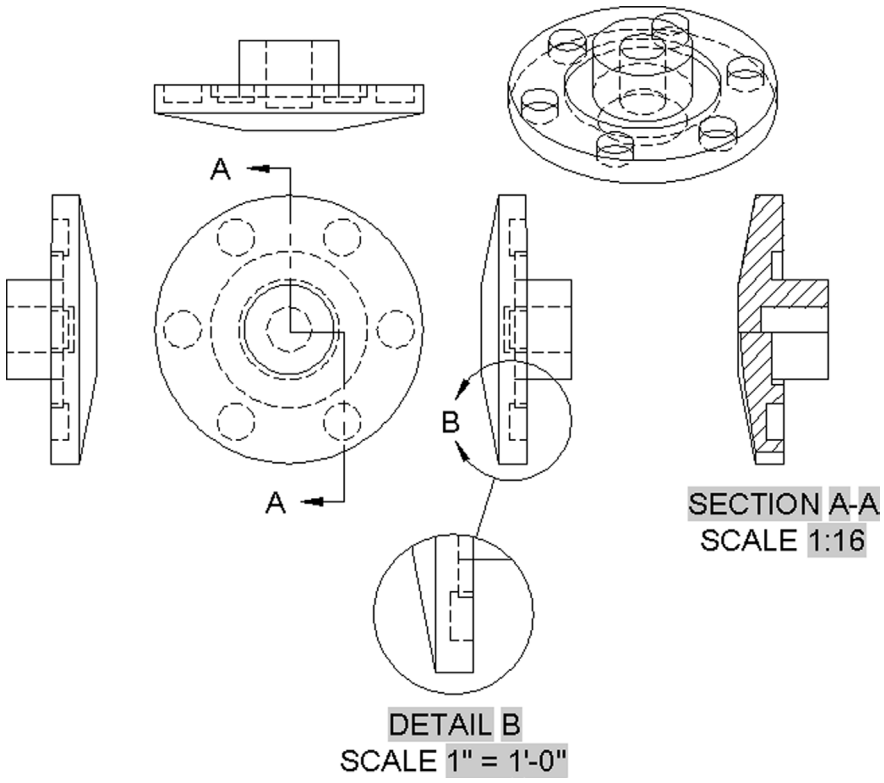


FIGURE 27.16 Full set of projections, sections, and details.

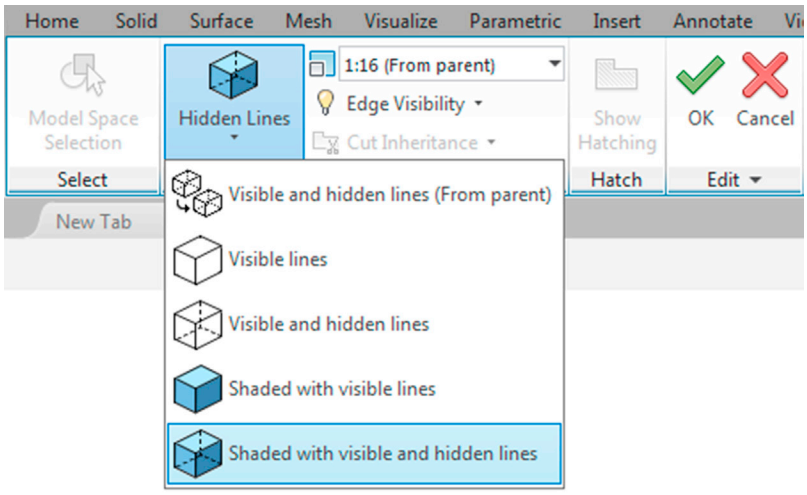


FIGURE 27.17 Shaded with visible and hidden lines.

27.4 FLATTEN AND FLATSHOT

The ability to create 2D flat views from 3D models was the central theme of the previous discussion, and we now add another set of tools to accomplish this. Flatten and Flatshot are two sides of the same coin. The commands exist to create 2D views, for illustrative purposes, out of 3D models. Note that these two commands do not create intelligent layouts, as

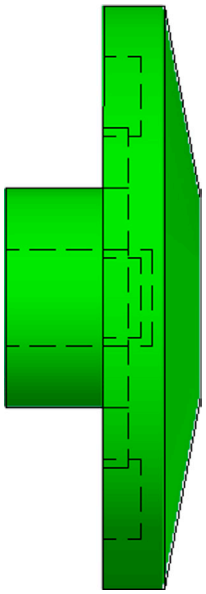


FIGURE 27.18 Result of modifying the view.

just discussed, rather they are used for a simple 2D "snapshots" of the model. What you see is what you get with these. The differences between Flatten and Flatshot are as follows:

- *Flatten*: The simpler of the two commands, it creates a 2D snapshot by just converting the current 3D view and model to 2D. You have to be careful using this one, as the original model is destroyed on conversion to 2D.
- *Flatshot*: A slightly more involved command, it gives you additional options in regard to scaling, positioning, and exporting the new 2D view. Most important, it does not destroy the original model but makes a 2D copy of it.

Both Flatten and Flatshot allow you to select whether or not to show hidden lines. Let us give them both a try with one of the side views we generated in the previous exercise.

Flatten

Step 1. There is no toolbar, cascading menu or Ribbon alternative, so type in `flatten` and press Enter.
- AutoCAD says: `Select objects to convert to 2d...`
  `Select objects:`

Step 2. Select the side view of the 3D model and press Enter.
- AutoCAD says: `Remove hidden lines? <No>:`

Step 3. You can pick `yes` or `no`. In this example, we keep the hidden lines. Press Enter. The flattened 2D model appears as seen in Fig. 27.19.

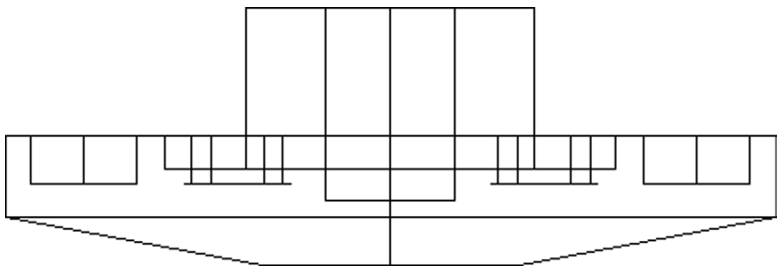


FIGURE 27.19 Flatten results.

Flatshot

Step 1. There is no toolbar or cascading menu alternative, but you can use the Ribbon's Home tab, Section panel, or type in `Flatshot` and press Enter. The dialog box shown in Fig. 27.20 appears.

Step 2. Press the Create button on the dialog box, and the 2D model appears on the screen attached to the movements of your mouse.

- AutoCAD says: `Units: Inches Conversion: 1.0000`
  `Specify insertion point or [Basepoint/Scale/X/Y/Z/Rotate]:`

Step 3. Click anywhere on the screen to place the model.

- AutoCAD says: `Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>:`

Step 4. Press Enter to accept the 1 to 1 X scale factor (unless you want to change it).

- AutoCAD says: `Enter Y scale factor <use X scale factor>:`

Step 5. Press Enter to accept the 1 to 1 Y scale factor (unless you want to change it).

- AutoCAD says: `Specify rotation angle <0>:`

Step 6. Press Enter to accept the 0 angle of rotation (unless you want to rotate the view). You are done; the 3D model looks like what is shown in Fig. 27.21.

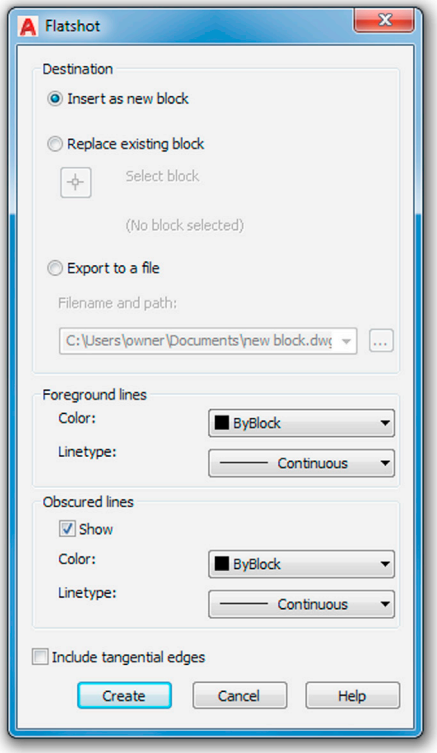


FIGURE 27.20 Flatshot dialog box.

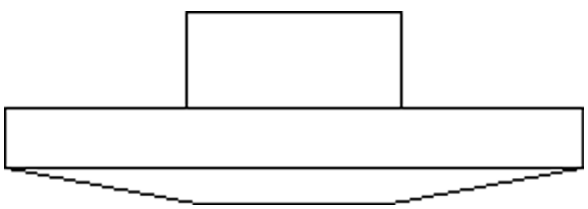


FIGURE 27.21 Flatshot results.

Make a note of some of the options in the Flatshot dialog box, such as exporting the file or changing the colors of the hidden lines, if you wish to do so.

27.5 VPORTS

Vports, which is short for *viewports*, is a unique tool in AutoCAD that allows you to view the same model from multiple angles all at once. The screen can be split in one of the several available ways to best fit the situation. Your design then appears in all screens, and you can assign each screen its own view and properties, such as shade or wireframe mode. A few releases ago, in AutoCAD 2015, some enhancements were added to this feature. To access vports, let us first bring up a simple 3D model, such as the wastebasket created in Chapter 24, Solid Modeling, shown again in Fig. 27.22.

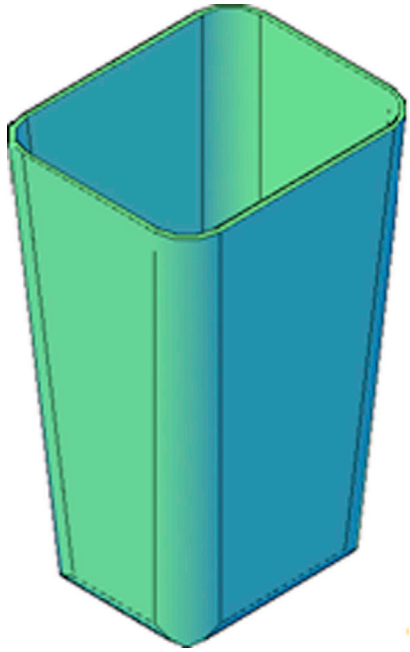


FIGURE 27.22 Wastebasket model.

Once you have this file opened or created the basket again, you can access vports via any of the following methods.

Keyboard: Type in **vports** and press Enter
Cascading menus: **View→Viewports→New Viewports...**
Toolbar icon: **Layouts** toolbar→ 
Ribbon: **Vizualize** tab→**Model viewports**

This brings up the dialog box seen in Fig. 27.23. Note that, if you use the Ribbon, you just choose one of these configurations right away, and the dialog box is bypassed.

As you can see from the configuration choices on the left, you have a few ways to arrange the screen. The default view is always Single, and that is the choice you will select later on, when you want to get back to the familiar single screen. For now, to try this out, choose the Four: Equal configuration and press OK. You should see what is shown in Fig. 27.24.

Notice that the view of the wastepaper basket has been duplicated four times and has not undergone any further changes. Keep in mind that only the view has been duplicated, not the actual 3D model. You may have to zoom to extents in each viewport to get the model to take up all the available area.

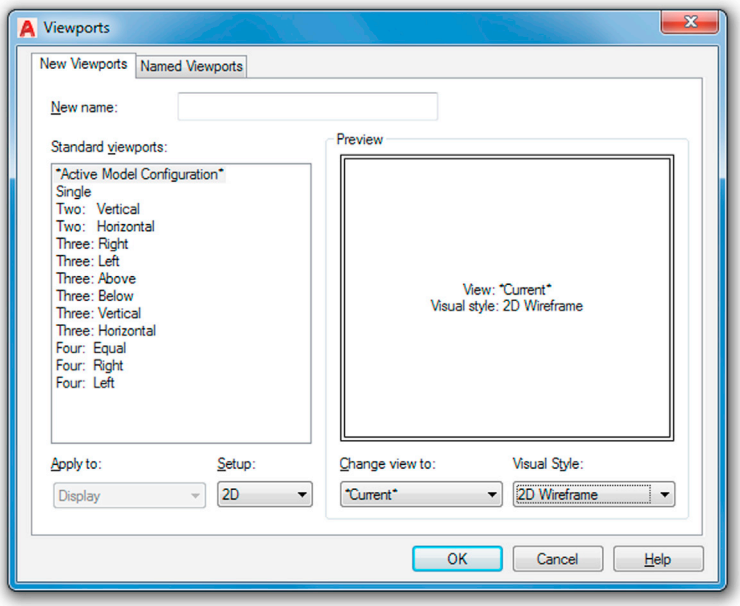


FIGURE 27.23 Viewports.

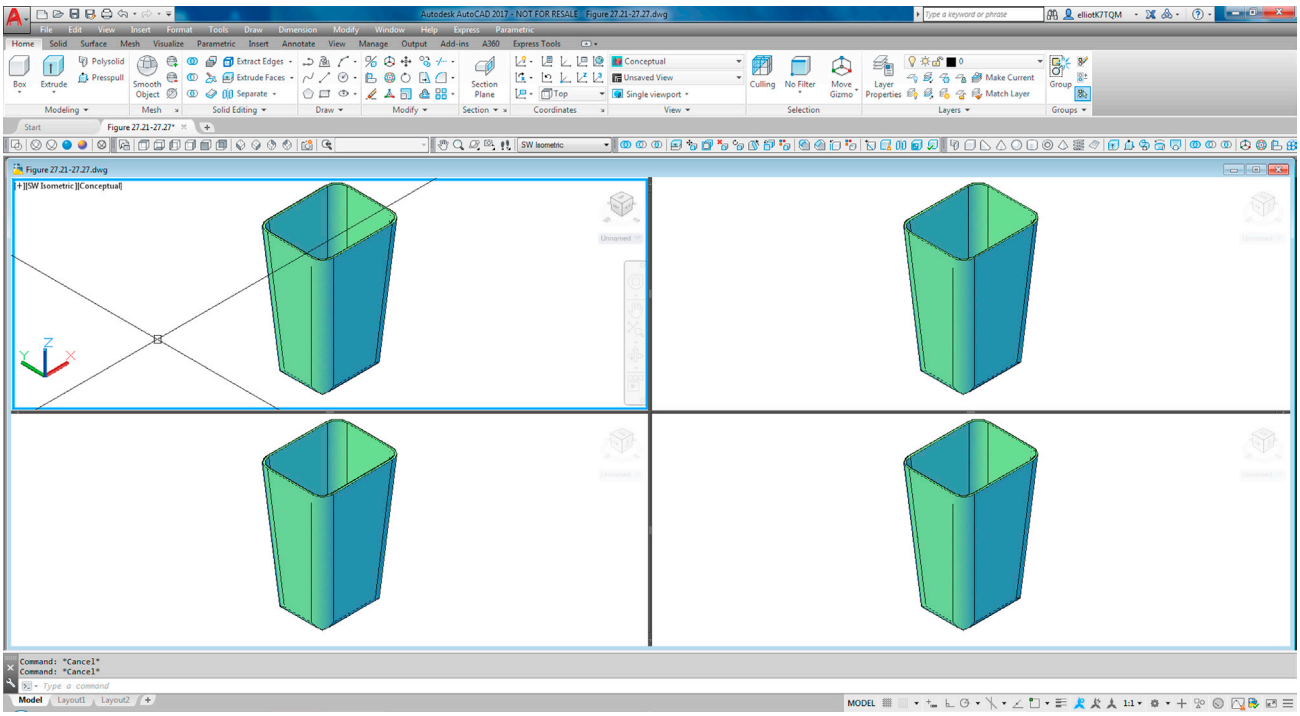


FIGURE 27.24 Four: Equal split.

The current viewport is highlighted with a blue frame, and you can also see the crosshairs and the ViewCube/Navigation Bar combination in the upper right corner. You can now easily jump from screen to screen by just clicking once in it. Go ahead and visit each one and modify the views to a combination of top, side, and isometric, as seen in Fig. 27.25.

Now, try working in one viewport. Notice that anything you do in one is reflected in the others, as it should be, since it is just one model with which you are working. Another popular configuration, Four: Left, is shown in Fig. 27.26. Here, you have one primary large view and three supplemental views to see other parts of the model.

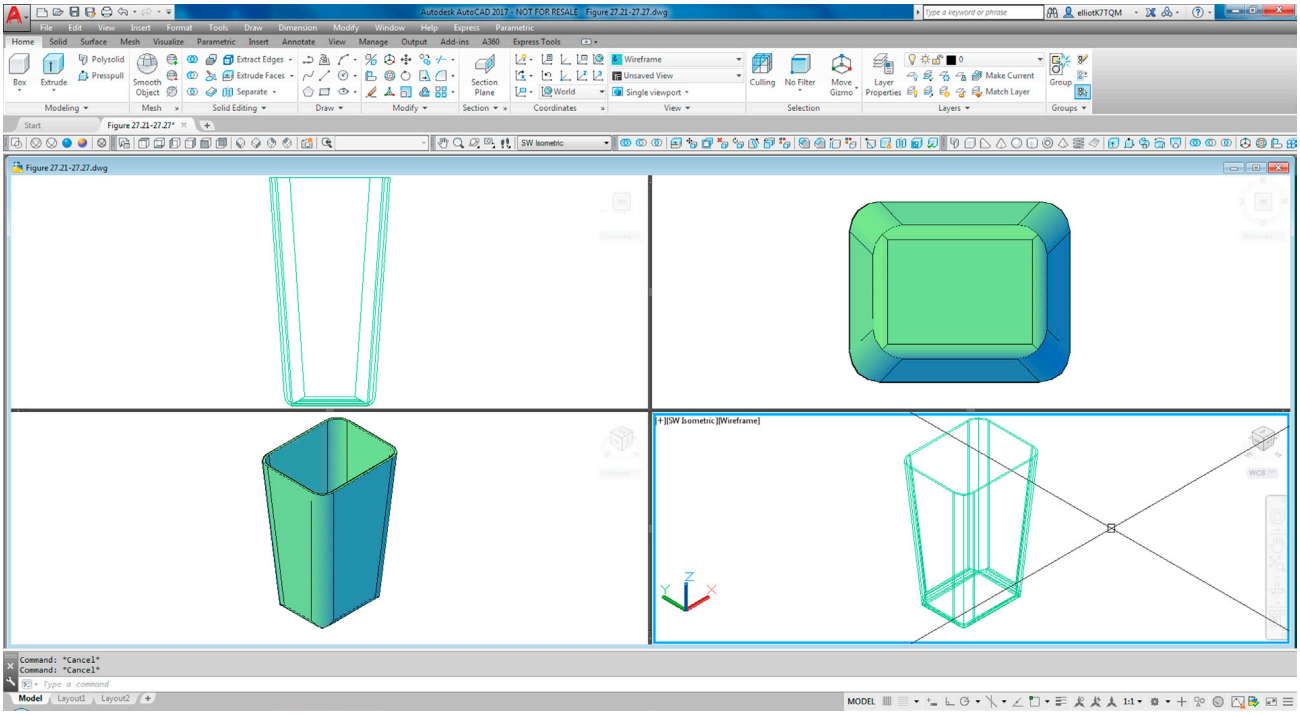


FIGURE 27.25 Four: Equal split with view modifications.

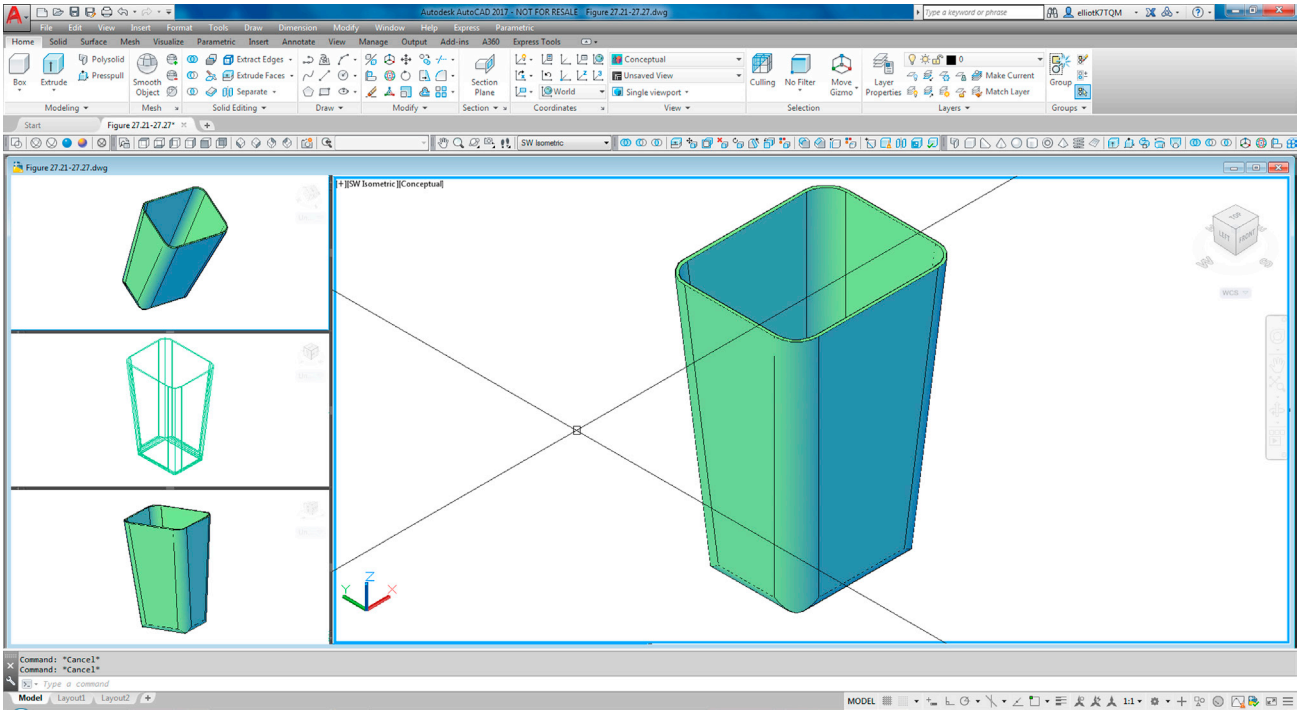


FIGURE 27.26 Four: Left split.

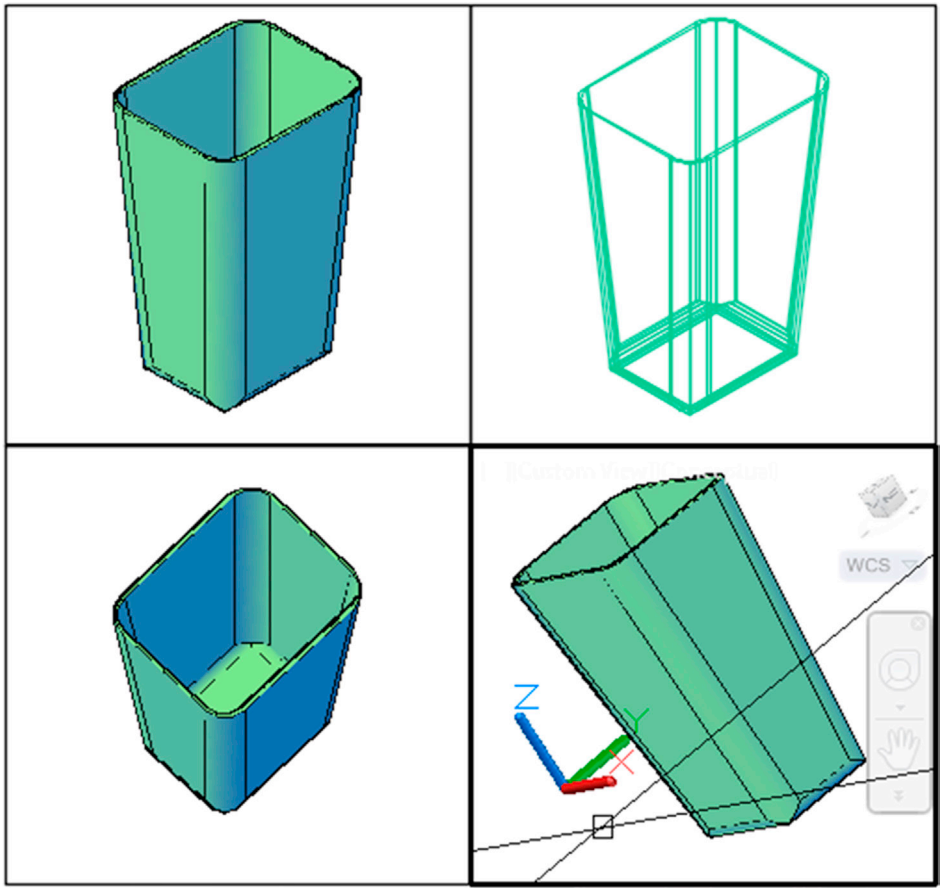


FIGURE 27.27 Four: Equal in Paper Space.

How you use these vports (or whether you use them at all) is up to you and your style of working in 3D. In some cases, they are very useful; in others, it is best to just rotate the model as you need and not maintain a constant view.

Another use for these vports is in plotting out the design for presentations, and it is here that this command gets a chance to really shine. While printing all the vports in Model Space is not possible, if you switch to Paper Space and repeat the vports Four: Equal command, you get what is shown in Fig. 27.27. Note that you have to actually draw the area where you want the vports to go before they appear and that the Ribbon's vports commands do not work in Paper Space. Zoom to extents in each one, and add some other modifications to arrive at what is shown in the figure.

Each view is still active and can be modified, the same as in Model Space, but now also the overall scene with all the viewports can be plotted or printed for an impressive display.

A new feature added to viewports in AutoCAD 2015 and continuing with AutoCAD 2017, is the ability to dynamically resize them and add/subtract new ones. This essentially frees you from the premade configurations available and allows you to position and size the viewports in virtually any way desired. To try this out, first start with the Four: Equal Split configuration seen in Fig. 27.24, then look for a tiny + and - seen in the frame of each viewport (to the right of the arrow in Fig. 27.28).

FIGURE 27.28 Viewport controls.

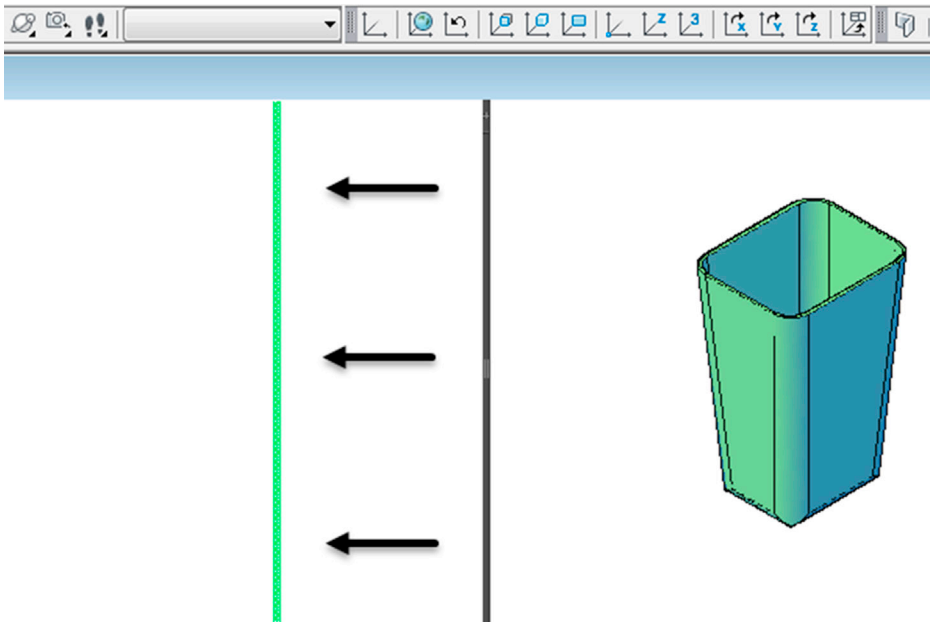


FIGURE 27.29 Creating new viewport.

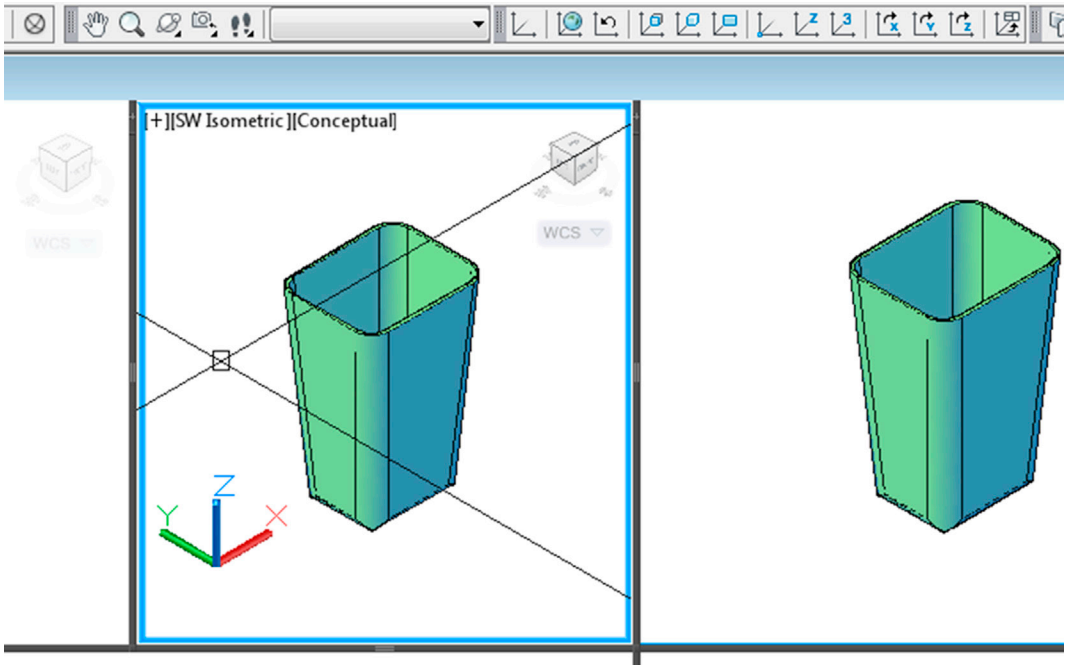


FIGURE 27.30 New viewport completed.

Press the plus or minus sign and drag a new viewport frame to either side until you have it where you want it. The frame is green while you are dragging it (in this case to the left), as seen in Fig. 27.29 by the black arrows. When you stop, a new viewport is created, as seen in Fig. 27.30. Existing viewports can also be adjusted dynamically by clicking and dragging the fames in either the vertical or horizontal direction.

Finally (and this is purely for entertainment purposes), you can split the screen into a maximum of 64 vports, as seen in Fig. 27.31. The standard vports dialog box does not let you do this, but type in `-vports` (a hyphen means it is "command line" only) and press Enter.

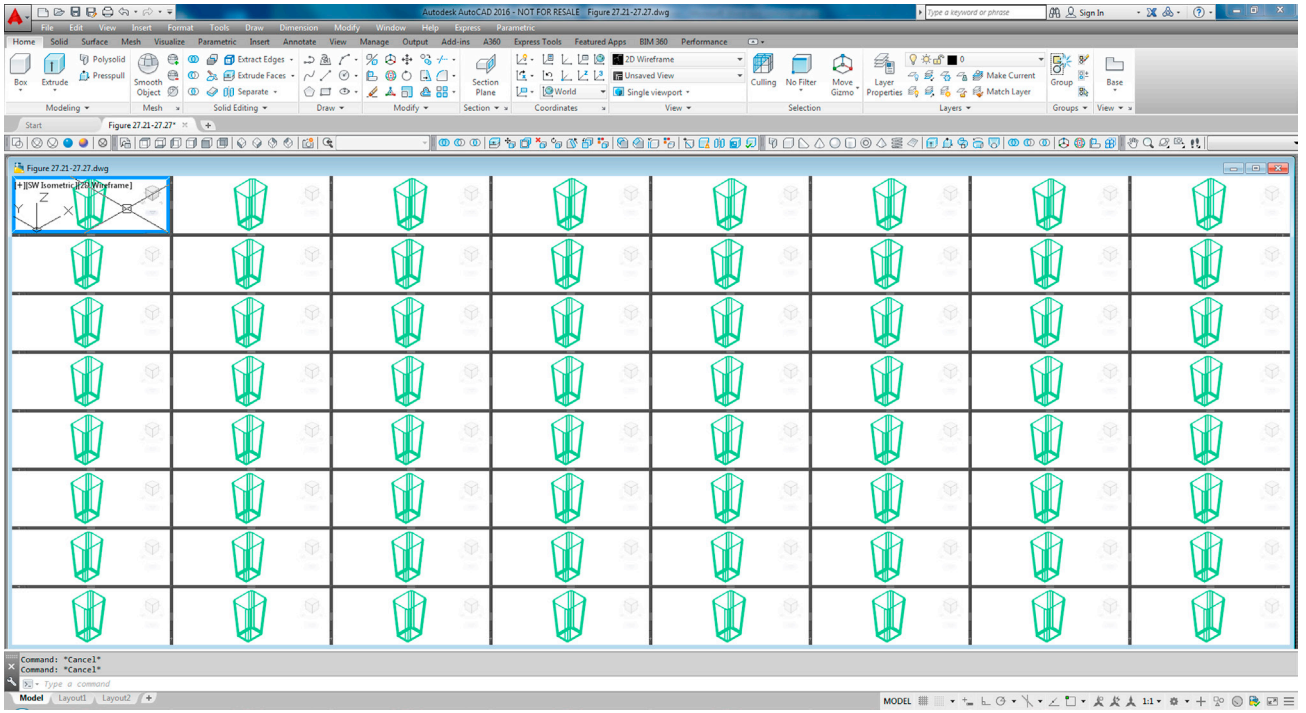


FIGURE 27.31 64 vports.

- AutoCAD says: `Enter an option [Save/Restore/Delete/Join/SIngle/?/2/3/4]<3>:`

  Enter the value `4` to split the screen into four views. Then, click into each new view and split it again in the exact same manner. In case you are wondering, no, this does not serve any real purpose.

27.6 SUMMARY

You should understand and know how to use the following concepts and commands before moving on to Chapter 28, Advanced UCS, Views, Text, and Dimensions in 3D:

- Slice
- Section
- Base view
- Projected view
- Section view
- Detail view
- Additional layout options
- Flatten
- Flatshot
- Vports

Review Questions

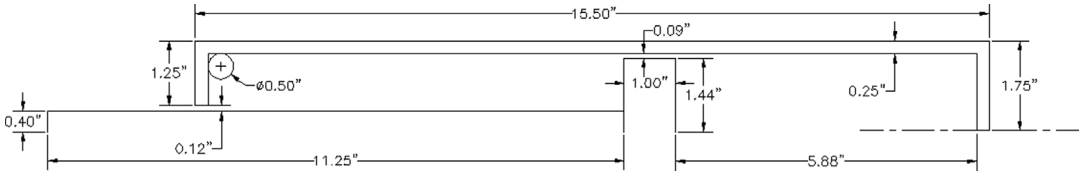
Answer the following based on what you learned in this chapter:

1. Describe the slice command.
2. Describe the section command.
3. What is the major difference between slice and section?
4. What does the VIEWBASE command accomplish?
5. What does the VIEWPROJ command accomplish?
6. What does the VIEWSECTION command accomplish?
7. What does the VIEWDETAIL command accomplish?
8. What is the critical difference between flatten and flatshot?
9. What are vports useful for? List two functions.

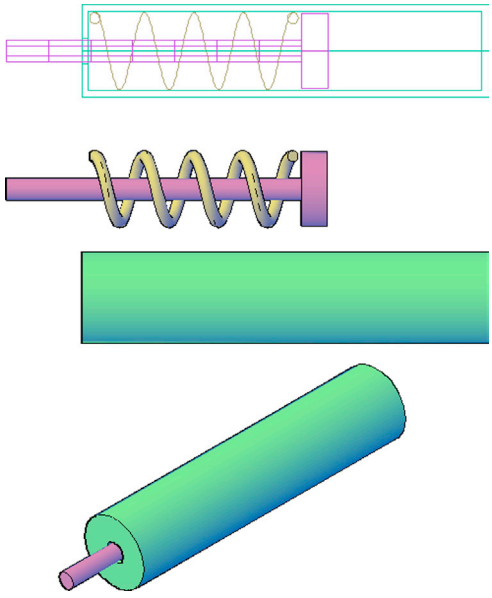
Exercise

1. To practice the concepts you learned in this chapter, let us create another shape (somewhat similar to one of the Level 1 projects) and use the slice, section, and layout tools to present it. You then create vports to view it from multiple angles. Although this is one exercise, it is broken down into multiple parts. (Difficulty level: Advanced; Time to completion: 60–90 minutes.)

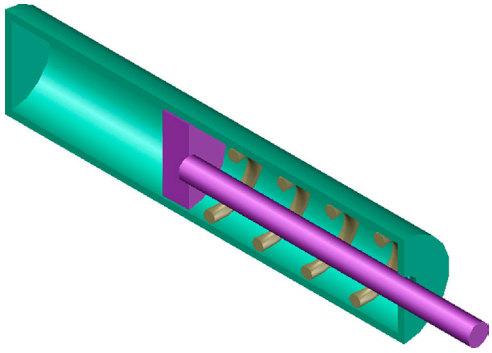
   Step 1. In 2D, create the following shape. It is a half-cylinder with a piston inside and a spring. The helix for the spring is based on the circle created via the TTR option.



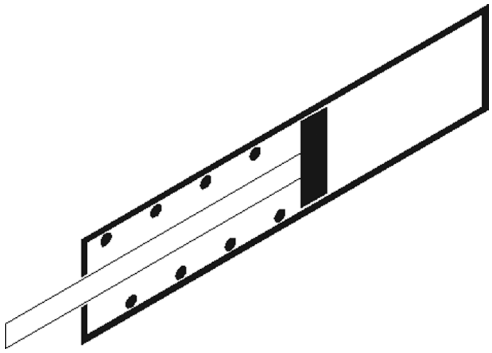
Step 2. Revolve the inner parts around the centerline, seen in Step 1, to create the piston and use the helix command to create the spring. Adjust the diameter and number of turns of the helix to fit the appropriate area, as shown. Revolve the outer shell to create the cylinder. Shown next are the individual parts, unshaded and shaded, as well as the full assembly. Of course, you do not see the inside structure once you complete Step 2.



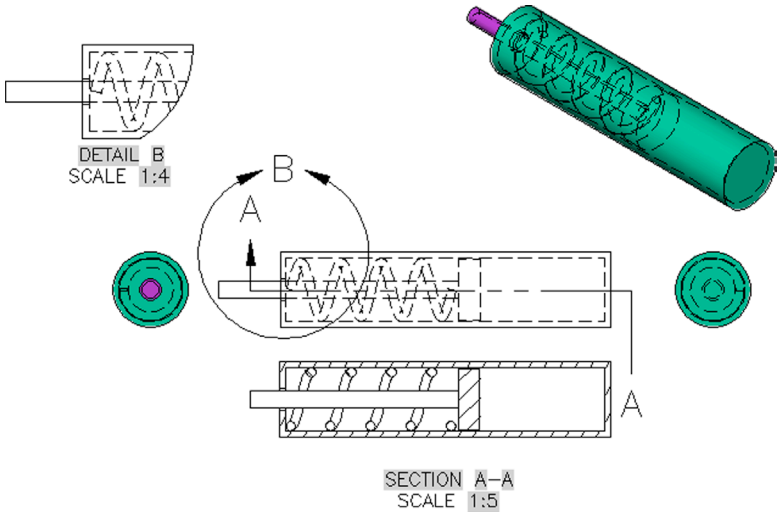
Step 3. Slice the view lengthwise as seen next. Use the YZ axis as the cutting plane.



Step 4. Section the view lengthwise, as seen next. Use the YZ axis as the cutting plane.



Step 5. Create a layout according to the steps of Section 27.3. A suggested layout is shown next, but you can improvise.



Step 6. As a final step, create multiple view of the design using the vports command, as seen next.