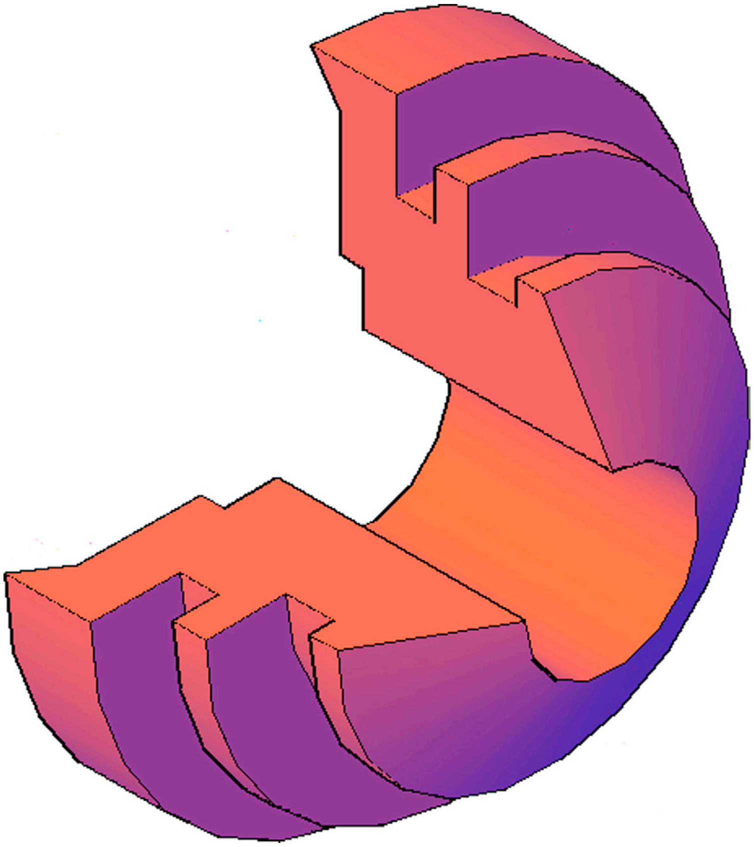
Chapter 24

Solid Modeling



Learning Objectives

In this chapter, we introduce a completely new set of tools in solid modeling, which are used for "curved design." These dramatically expand the range of design work that you can do. We specifically look at

- Revolve
- Shell
- Taper
- Loft'
- Path extrusion
- Sweep

By the end of this chapter, you will be able to create designs that were not possible with previous tools, with much of this new knowledge applicable to engineering design.

Estimated time for completion of this chapter: 2–3 hours.

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24.1 INTRODUCTION TO SOLID MODELING

With this chapter, we head in a new direction and begin talking about some of the advanced solid modeling tools that make up what can be referred to as *curved design*. While these tools are not necessarily more difficult to master, they do open up a new range of modeling possibilities, including some rather complex 3D designs. We introduce revolve, shell, taper, loft, path extrusion, and sweep. Although some of these commands, such as taper and shell, do not involve curves, they are included with this advanced tools group.

Revolve

As essential as extrude was to flat design, such is the importance of revolve to curved design, and it is the first command we cover. Revolve easily creates complex revolved shapes based on the concept of solid or surface of revolution, and this command has no other equivalent in AutoCAD.

The basic idea is the following. We would like to draw a 2D outline of an object and revolve this outline or "profile" around an axis. If the profile is closed, the result is a solid of revolution; if the profile is open (has gaps) or lines are used instead of polylines, the result is a surface of revolution. This process can create some unique shapes that are impossible to make any other way, but the entire concept rests on the designer's ability to visualize the profile on which the model is built.

In past releases of AutoCAD, the revolve command was a bit picky, and you had to use a closed polyline for the revolution to work, no exceptions. These restrictions have been lifted, and you can create a profile using the regular line command and even leave gaps in it; although, once again, just be aware that this creates surfaces, not solids, as is shown later. Your axis of revolution can be anywhere on or near the object, and you need not revolve all the way around but can use any degree value in between if necessary, such as when you want to show a cross section.

Let us put all this together with some examples. In 2D, create a profile approximately similar to the one shown in Fig. 24.1, using a polyline to ensure a solid. We try it with regular lines later on.

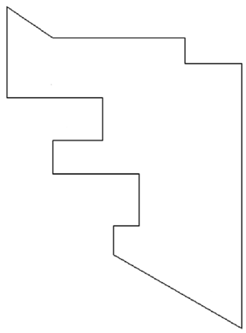
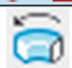


FIGURE 24.1 Polyline profile for revolve.

Keyboard: Type in **revolve** and press Enter
Cascading menus: Draw→Modeling→Revolve
Toolbar icon: **Modeling** toolbar→ 
Ribbon: **Solid** tab→**Solid** 

Step 1. Start revolve via any of the preceding methods.

- AutoCAD says: `Current wire frame density: ISOLINES = 4,`
  `Closed profiles creation mode = Solid`
  `Select objects to revolve or [MOde]:`

Step 2. Select the profile and press Enter.

- AutoCAD says: `Specify axis start point or define axis by [Object/X/Y/Z]`

Step 3. Here, AutoCAD is asking for an axis of revolution for the profile. This can be addressed in many ways, and the axis can be on the object or some distance away from it (causing the revolved solid to have a hole about the axis). For this example, we pick two points at the top right and bottom right of the profile, as seen in Fig. 24.2. Using the endpoint OSNAP, click on these two locations.

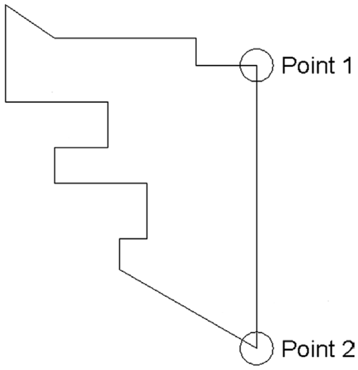


FIGURE 24.2 Selecting rotation axis.

Step 4. After the first click,

- AutoCAD says: `Specify axis endpoint:`

  After the second click,
- AutoCAD says: `Specify angle of revolution or [STart angle/Reverse/EXpression] <360>:`

Step 5. Here, you can ask for a full 360° revolution by just pressing Enter, but instead type in `270`, so we can see a cutout of the part. If you are in 3D, you can also just trace out the revolution angle with movements of your mouse. For now though, you see a 2D revolved surface, as in Fig. 24.3.

To see the full effect of what you have done, switch to the familiar SW Isometric view and also shade and color the solid. The far more dramatic result is seen in Fig. 24.4.

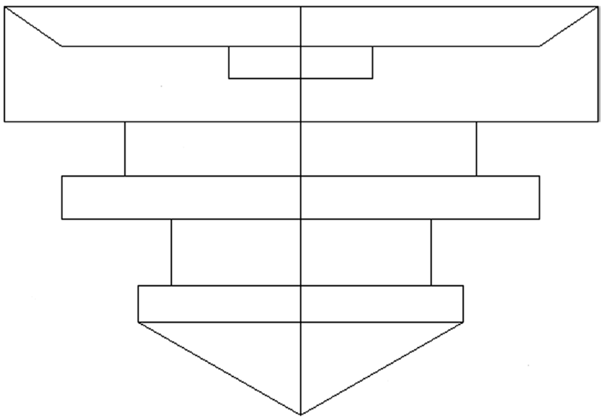


FIGURE 24.3 Revolved profile in 2D wireframe.

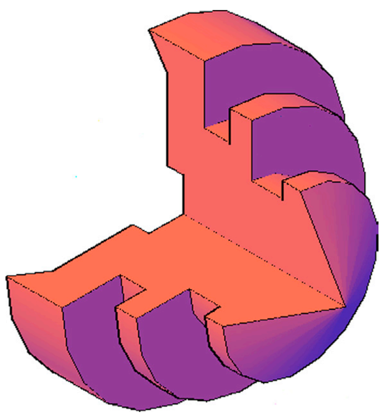


FIGURE 24.4 Completed revolution, shaded and colored.

For some other ideas, undo your last step, and try locating the axis of revolution some distance away from the object (Fig. 24.5). This creates a hole in the center of the object on execution of revolve, as seen in Fig. 24.6.

The axis of revolution can actually be anywhere. In Fig. 24.7, it is above the profile and at an angle, creating a completely different solid of revolution, as seen in Fig. 24.8.

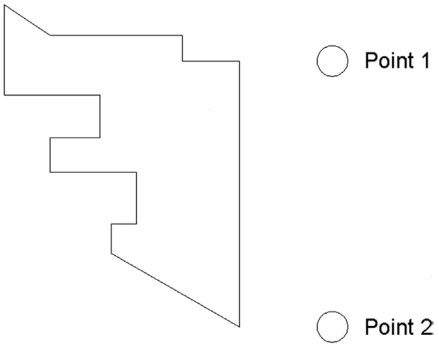


FIGURE 24.5 Offset axis of revolution, Example 1.

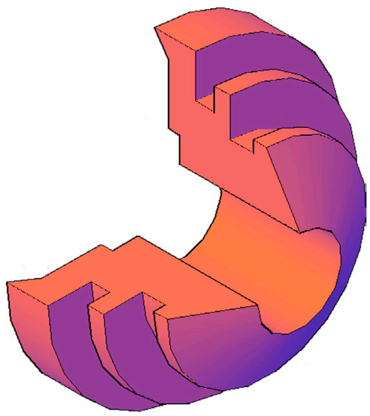


FIGURE 24.6 Completed offset revolution, Example 1.

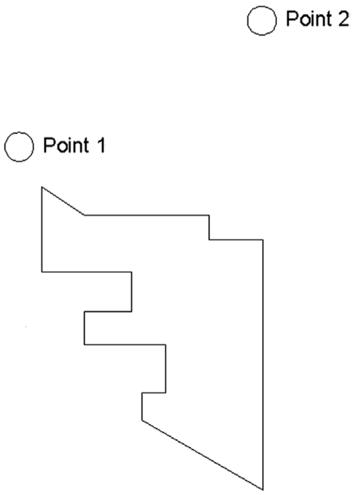


FIGURE 24.7 Offset axis of revolution, Example 2.

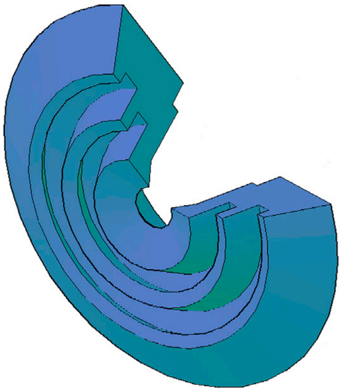


FIGURE 24.8 Completed offset revolution, Example 2.

Next, let us try the same profile but with a piece missing (Fig. 24.9). The profile no longer is a closed polyline, and in this case, the revolve command creates a surface, not a solid, a difference you can easily see. The same effect is achieved using plain old lines. Even if they are connected in a loop, AutoCAD still defaults to the surface revolution. Fig. 24.10 shows the result.

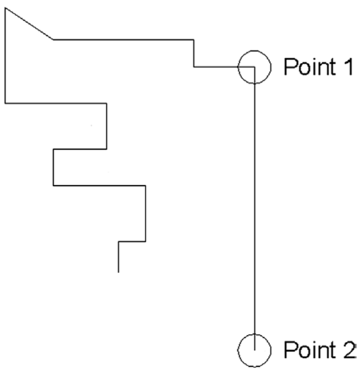


FIGURE 24.9 Surface of revolution profile.

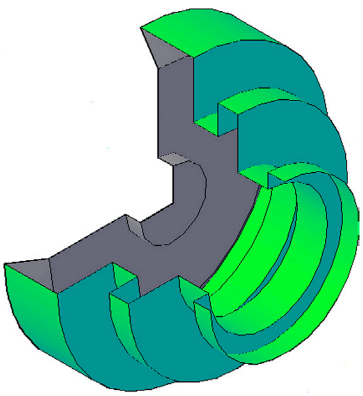


FIGURE 24.10 Completed surface revolution.

Try the following example of revolution, a 3D cup. Draw the profile in Fig. 24.11, pedit, and revolve 360°. 0054hen, switch to 3D, and shade, color, and rotate if needed. You now have your first piece of glassware for a future 3D house, as seen in Fig. 24.12.

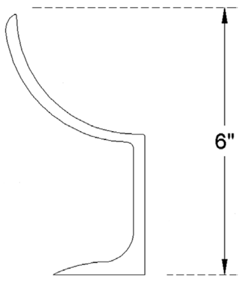


FIGURE 24.11 Cup profile.

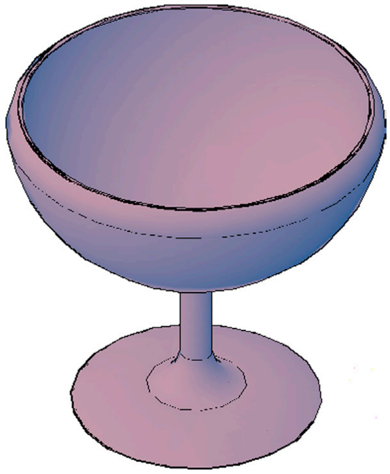


FIGURE 24.12 Completed cup.

As you can see, revolve is a very versatile, powerful command and is really limited only by your imagination and visualization skills. Be sure to go through each example presented thus far. None of the profiles or solids has to look exactly like the ones shown, and indeed, no dimensions are provided (except for the cup height), but be sure to understand the underlying principles and what is being done to produce what you see.

Shell

Shell is one of several AutoCAD 3D commands "borrowed" from engineering solid modeling software due to their usefulness. The command creates a "shell" of a predetermined thickness out of a solid object. The shell is of an even thickness all around and can have as many open surfaces as needed.

Let us demonstrate by example. Draw a 10″ × 8″ rectangle and extrude it out to 4″, as seen in Fig. 24.13.

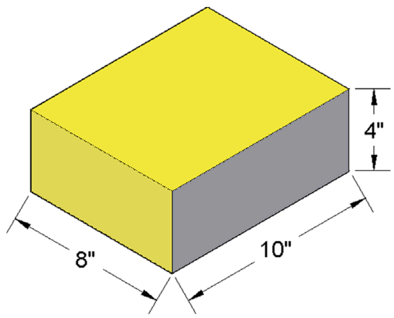


FIGURE 24.13 Basic block for shell.

To activate the command, you cannot type in `shell` (that is an OS command), but you can use other methods, as seen next in the command matrix.

Keyboard: **Solidedit**, Enter. **Body** option, then **Shell**
Cascading menus: **Modify→Solid Editing→Shell**
Toolbar icon: **Solid Editing** toolbar→ 
Ribbon: **Solid** tab→**Solid Editing** 

Step 1. Start the shell command via any of the preceding methods.

- AutoCAD gives you a lengthy response:

```
Solids editing automatic checking: SOLIDCHECK = 1
Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>:_body
Enter a body editing option
[Imprint/seParate solids/Shell/cLean/Check/Undo/eXit] <eXit>:_shell
Select a 3D solid:
```

Step 2. Select the extruded block.

- AutoCAD says: `Remove faces or [Undo/Add/ALL]:`

Step 3. Carefully pick the upper surface of the extruded block. Be careful not to select an edge in this case but an actual flat surface. You may select more than one surface, and it is something you should try the next time you practice the command. For now, just press Enter after selecting the top surface.

- AutoCAD says: `Enter the shell offset distance:`

Step 4. Here, you need to enter a valid shell size (thickness). Some complications can result at this point if your shape has other modifications to it, such as fillets. Then, if the thickness of the walls is too thin, you can "eat through" the material, causing errors or actual holes in the shape. This particular example should proceed smoothly. Enter `.5'` for the thickness.

- AutoCAD says: `Solid validation started.`

```
Solid validation completed.
Enter a body editing option
```

The shell command executes, and the result is as seen in Fig. 24.14.

As suggested earlier, try removing more than one face. The result is an "open" box, as seen in Fig. 24.15. You can remove all four sides if you wish and end up with a flat plate, although that somewhat defeats the whole point of shell.

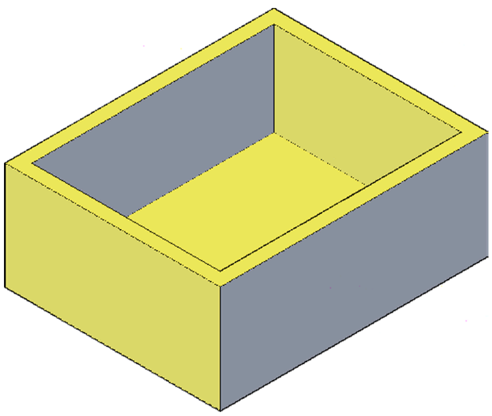


FIGURE 24.14 Completed shell.

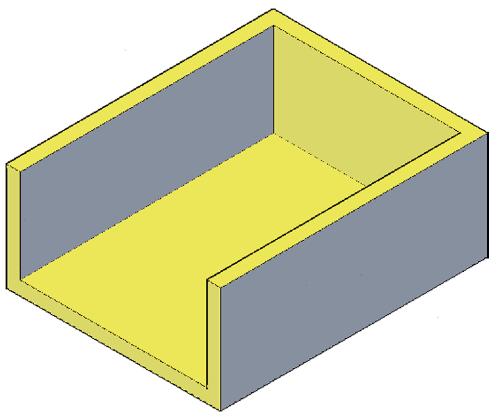


FIGURE 24.15 Completed shell (two faces removed).

Taper

Taper is a useful command that does exactly what it advertises: It tapers (or creates a gradient) on one or more sides of an object. A simple example, and one we actually model soon, is a plastic wastebasket commonly found in any office. If you have one nearby, take a look at it. Its base is a slightly smaller rectangle than the top rim. The reason for tapering such items is because of the nature of plastic molds and the manufacturing process. Another useful result is that now these wastebaskets are easily stackable one on top of another.

Let us practice this command on a tall rectangular box. Draw a 20″ × 15″ rectangle and extrude it to 30″, as seen in Fig. 24.16.

The taper command is another one that cannot be typed in, and the other input methods must be used.

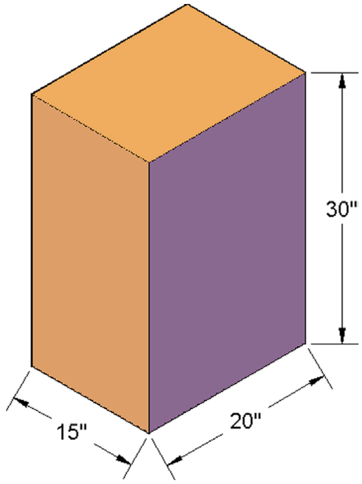
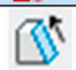


FIGURE 24.16 Box for tapering.

Keyboard: **Solidedit**, Enter. **Face** option, then **Taper**
Cascading menus: **Modify→Solid Editing→Taper faces**
Toolbar icon: **Solid Editing** toolbar→ 
Ribbon: **Solid** tab→**Solid Editing** 

Step 1. Start the taper command via any of the preceding methods.

- AutoCAD again gives you a lengthy response:

```
_solidedit
Solids editing automatic checking: SOLIDCHECK = 1
Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>:_face
Enter a face editing option
[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/mAterial/Undo/eXit] <eXit>:
_taper
```

Step 2. Here, the first task is to select all the faces you want to taper. This can be as few as one face or as many as all of them. Be careful though; by *face*, we mean the ones that will taper. Therefore, the top and bottom of the box are not valid choices, only the sides. Also, note that it is easier to select faces when you are in wireframe view. In shaded view, you need to rotate around the object to get all the faces; in wireframe, you can (carefully) pick "through" the shape. When you pick a face it becomes dashed.

- AutoCAD says: `Select faces or [Undo/Remove/ALL]: 1 face found.`

Step 3. When done press Enter.

- AutoCAD says: `Specify the base point:`

Step 4. Here, you need to consider what is meant by *base point*. It is the start of the taper. You would need to pick either the top or the bottom of the block. Since this is a wastebasket, which tapers in as you go down, we pick the midpoint of one of the lines representing the *top face or surface*. If you choose a similar point on the bottom face or surface, then the block tapers up, like a pyramid. Go ahead and select the base point, as discussed.

- AutoCAD says: `Specify another point along the axis of tapering:`

Step 5. This, of course, is the opposite of where the first point was, so pick the midpoint of the *bottom face* or *surface*. In summary, you are giving AutoCAD what amounts to a straight line that tells it in what direction the taper is going.

- AutoCAD says: `Specify the taper angle:`

Step 6. This final request asks for the amount of taper in degrees. Enter a small value, perhaps `4` or `5`, and press Enter. The block tapers evenly on all sides. The result in shade mode is shown in Fig. 24.17.

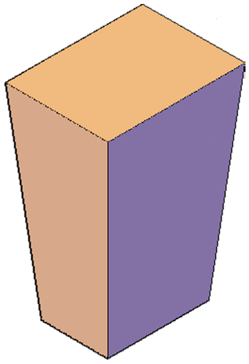


FIGURE 24.17 Completed taper.

3D Modeling Exercise: Wastebasket

Let us put extrude, taper, fillet, and shell together into one exercise.

Step 1. Clear your screen of previous work, and make sure you are in the SW Isometric view. Starting from the beginning, draw a 22″ × 18″ rectangle and extrude it to 34″. The result is a block of material similar to what we just created when we practiced the taper command. It is shown shaded, we use a different color, in Fig. 24.18.
Step 2. Taper the block on all four sides to 4°, as seen in Fig. 24.19.
Step 3. Fillet the side edges with a 3″ radius, and the bottom with a 1.5″ radius, as seen in Fig. 24.20.
Step 4. Shell the wastebasket to a 0.35″ thickness, as seen in Fig. 24.21.

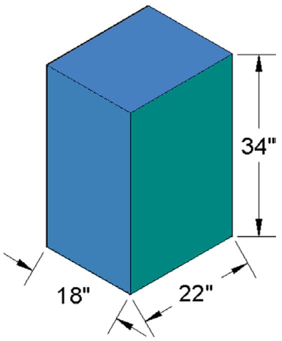


FIGURE 24.18 Wastebasket, Step 1.

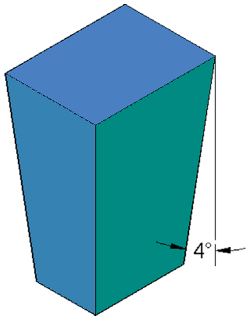


FIGURE 24.19 Wastebasket, Step 2.

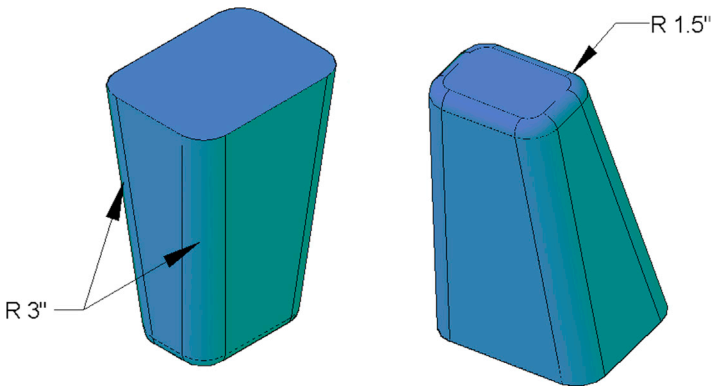


FIGURE 24.20 Wastebasket, Step 3.

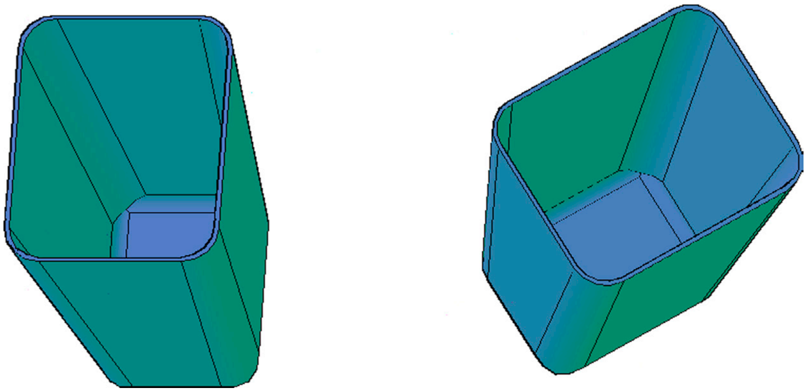


FIGURE 24.21 Wastebasket, Step 4.

Loft

Loft is a relatively recent addition to the AutoCAD toolbox, added along with many other 3D enhancements in AutoCAD 2007. It is another command "borrowed" from the engineering solid modeling world and is one of the more useful and versatile.

Despite some rather complex surface and part modeling programming "under the hood" with loft, as far as the AutoCAD end user is concerned, the command works on a simple principle. It connects together open or closed profiles. All you have to do is pick those profiles in the order you want them lofted and AutoCAD "connects the dots," so to speak, and creates either a surface (if the profile is open) or a solid (if the profile is closed). Take the very simple example of a classic wooden barrel, as shown in Fig. 24.22.

While, in this particular case, you can use revolve to model the object, this is only because it is symmetric about the center axis. We often do not have such a convenience, so we need to default to loft instead. The barrel's lengthwise profile can be modeled as three circles, one after another, with the first and third representing the top and bottom, and the second (slightly larger one) representing the fatter middle part, as seen in Fig. 24.23.

These rings are drawn to a random size, but both outer rings are an equal distance from the larger center one, ensuring our barrel is not lopsided. You can easily set up these 3D profiles by drawing two sizes of circles in 2D Top View, with each having the same center point. Then, switch to a side view and "lift" the smaller circle upward (or the larger one downward). Finally, a quick mirroring completes the profile. You should be able to do this type of 3D work with little effort at this point in the course.

Let us now run the loft command and get a barrel.

FIGURE 24.22 Barrel.

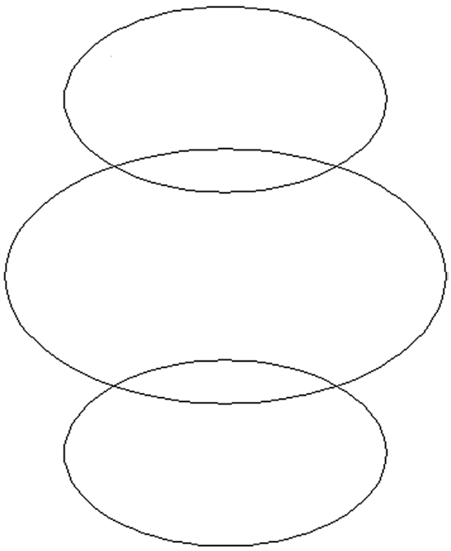


FIGURE 24.23 Three-ring profile of a barrel.

Keyboard: Type in **loft** and press Enter
Cascading menus: Draw→Modeling→Loft
Toolbar icon: Modeling toolbar→
Ribbon: Solid tab→Solid

Step 1. Start the command via any of the preceding methods.

- AutoCAD says:

```
Current wire frame density: ISOLINES = 4,
Closed profiles creation mode = Solid
Select cross sections in lofting order or [POint/Join multiple edges/MOde]:_MO
Closed profiles creation mode [SOlid/SUrface] <Solid>:_SO
Select cross sections in lofting order or [POint/Join multiple edges/MOde]:
```

Step 2. Select the hoops, one after another, from top to bottom (or vice versa). Each one becomes dashed as you select it. Selecting the hoops out of order does not necessarily stop the command but gives you unexpected results.

- AutoCAD says: `1 found, 3 total`

Step 3. Press Enter when all three are selected.

- AutoCAD says: `Enter an option [Guides/Path/Cross sections only/Settings]<Cross sections only>:`

Step 4. Press Enter. The loft is performed and the barrel is visible if you are in the shaded mode. You also see a prominent down arrow, which when pressed, reveals a menu to fine-tune the loft (Fig. 24.24). Experiment with the various choices to see the effect on the overall shape. In almost all cases, we want a smooth fit (the default), so after experimenting, restore that choice.

The final result is shown again in Fig. 24.25. The down arrow is gone but can be brought back by a single mouse-click on the shape.

While this was a very simple example, the essentials of loft are all there. Because the circles by definition are closed profiles, we get a solid out of this. If, for example, we had three arcs instead (in the same positions), loft would yield a surface, as seen in Fig. 24.26. Go ahead and try this by trimming the circles. We use this surfacing ability later on to generate some rather complex surfaces quickly and easily.

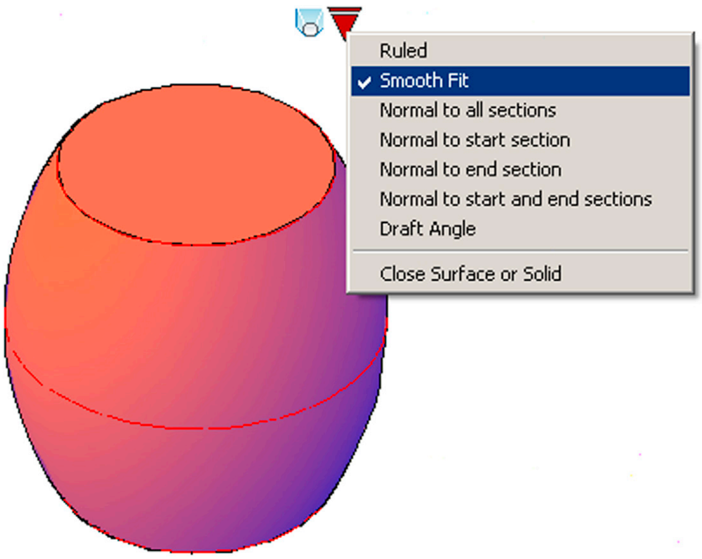


FIGURE 24.24 Loft settings menu.

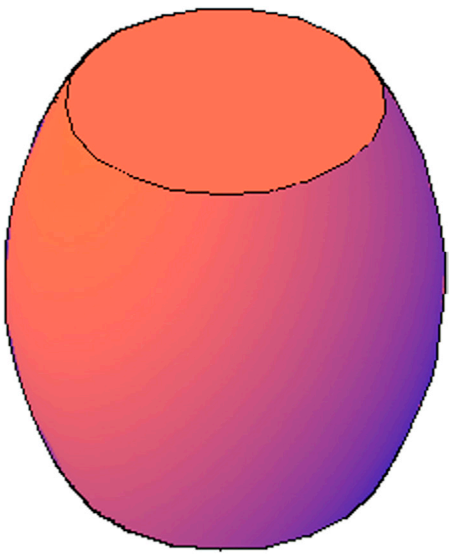


FIGURE 24.25 Loft completed.

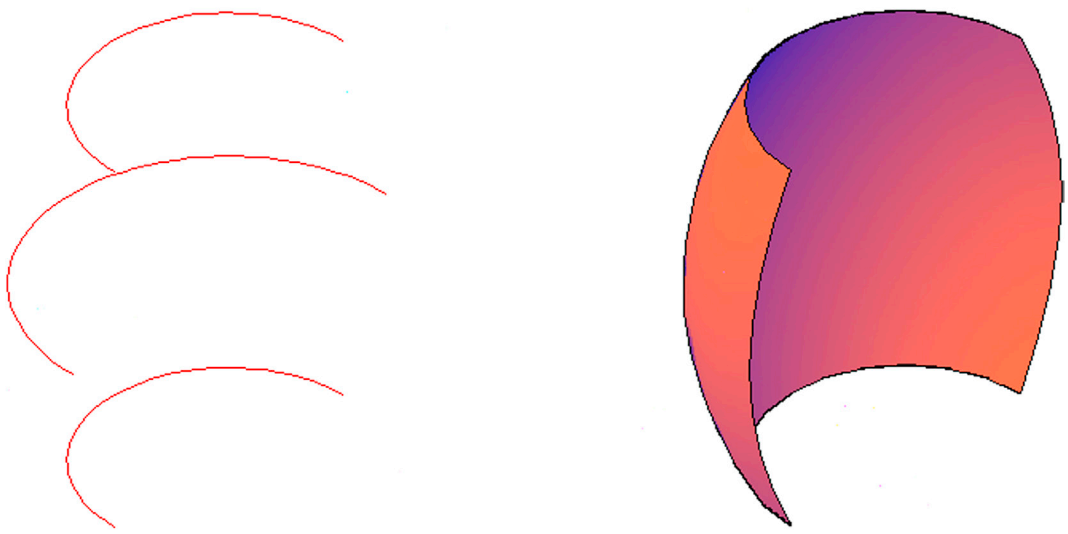


FIGURE 24.26 Loft using arcs.

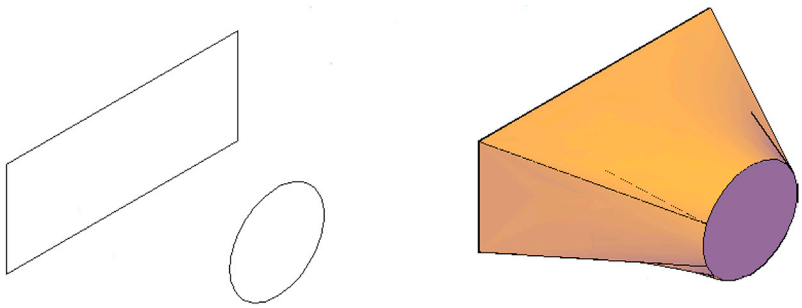


FIGURE 24.27 Loft using a rectangle and a circle.

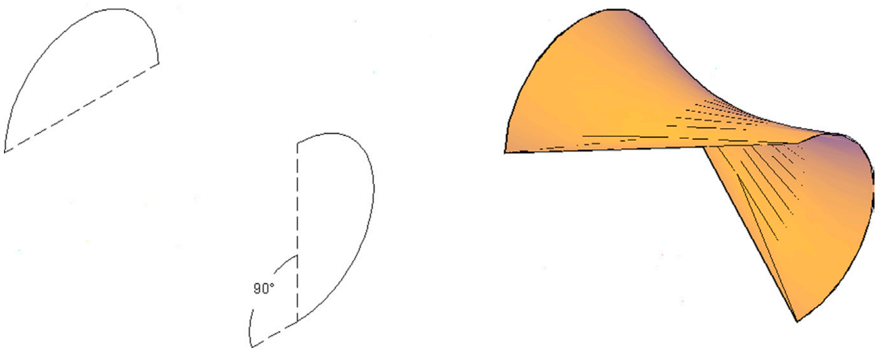


FIGURE 24.28 Loft using two arcs.

The true power of loft shows when you need to smoothly merge profiles of varying shapes, sizes, and locations. AutoCAD's Help files have an extensive list of items that qualify as cross sections, so be sure to take a look. Sometimes, you may hit a snag in performing loft due to an unusual combination of elements, but as long as they are not on the same plane (two plates on a dinner table is an example of objects on the same plane), you should be able to complete the operation. Some examples shown next include lofting a solid between a rectangle and a circle some distance away (Fig. 24.27) and lofting a surface between two arcs some distance apart (Fig. 24.28).

3D Modeling Exercise: Mechanical Drill Bit

This is a good example of a loft application. We create three profiles, position them, and loft between them, creating a complex shape.

Step 1. Draw a circle and one tooth, similar to what is found on a gear. Then, array it 12 times around the circle. Finally, do a little bit of cleanup trimming and pedit the entire structure. This is your basic profile, as seen in Fig. 24.29. You need not draw the exact same thing; something close is just fine.

Step 2. Now copy the profile for a total of three items. Make each new copy 1.5 times larger than the previous one using scale, as seen in Fig. 24.30.

Step 3. Carefully place the three profiles one on top of another using their center points and rotate profiles 2 and 3 (the medium and the largest ones) 10° and 20°, respectively, as seen in Fig. 24.31.

Step 4. Now, switch to the 3D SW Isometric view and separate the three profiles evenly in size order, as seen in Fig. 24.32. Be careful, you may have to 3Drotate all the profiles to get them to "stand upright." Also, be sure to check the side profiles to make sure they are spaced evenly. Use Ortho, and when done, 3Dorbit around the profile to check that they are located just right.

Step 5. Finally, you are ready to execute loft. Fig. 24.33 shows the result.

As you can see from this and the previous examples, loft is a very versatile and powerful command. You can represent virtually any object as long as you are able to visualize and correctly draw a suitable profile.

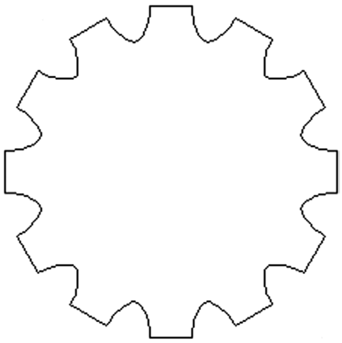


FIGURE 24.29 Drill bit, Step 1.

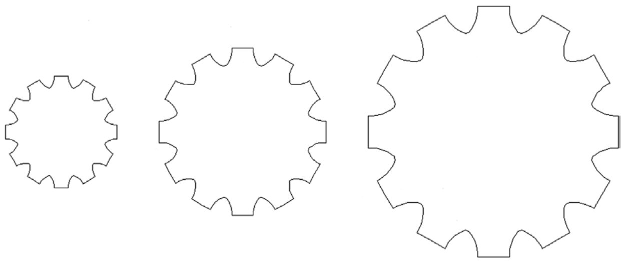


FIGURE 24.30 Drill bit, Step 2.

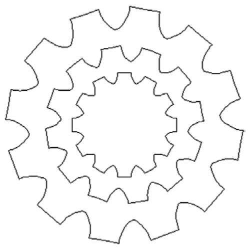


FIGURE 24.31 Drill bit, Step 3.

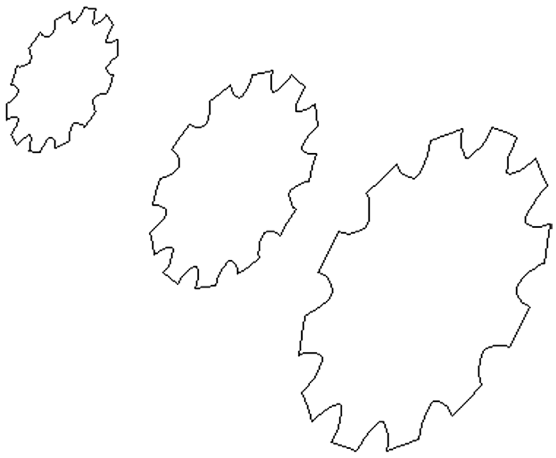


FIGURE 24.32 Drill bit, Step 4.

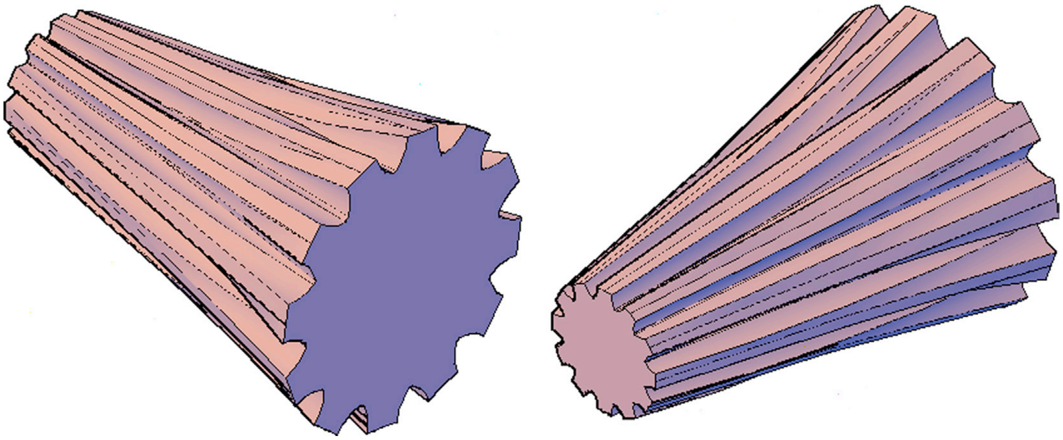


FIGURE 24.33 Drill bit, Step 5.

Path Extrusion

Path extrusion and sweep, which follow next, are commands with a similar mission but different approaches. Path extrusion is the older command and still quite useful in certain situations, while the newer sweep just expands and simplifies the same idea.

The central idea here is to allow extrusion along a given path, which can be curved and otherwise nonlinear (recall that the basic extrude command was linear in nature). Path extrusion is technically not a new command, as the option to select a path while extruding was actually always there in the command's submenus, so all we are doing here is finally trying it out.

Draw a circle of any diameter or radius. Then, draw an L-shaped set of lines, as shown in Fig. 24.34, and add a small fillet to the intersection of the lines; finally pedit the lines and arc.

This line is the path, and the circle is the profile we would like to extrude. There is one more step before you can do the path extrusion, however. This command requires that the profile (circle) be turned perpendicular to the path (line). Use 3Drotate to rotate the circle 90°. It will look like the one shown in Fig. 24.35. Now switch to 3D. You may need to 3Drotate the pieces into an upright position, as seen in Fig. 24.36.

FIGURE 24.34 Basic path.

FIGURE 24.35 Basic path, circle rotated.

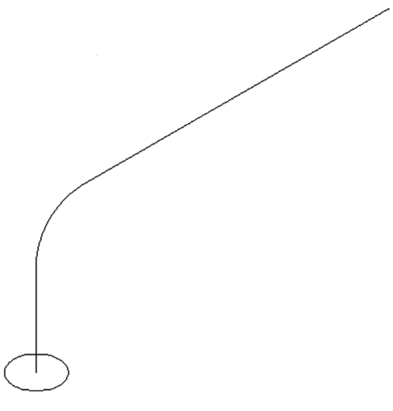


FIGURE 24.36 Basic path in 3D.

We can now do the path extrusion. There is no separate path extrusion command, rather it is a submenu of the regular extrude.

Step 1. Start up extrude via any preferred method.
- AutoCAD says: `Current wire frame density: ISOLINES = 4 Select objects to extrude:`

Step 2. Select the circle and press Enter.
- AutoCAD says: `Specify height of extrusion or [Direction/Path/Taper angle]`

Step 3. Select`p` for `Path` and press Enter.
- AutoCAD says: `Select extrusion path or [Taper angle]`

Step 4. Click to select the path itself. You see what is shown in Fig. 24.37, in wireframe.

The extrusion is then shaded and colored to give the final result, as seen in Fig. 24.38. This view is also switched to SE Isometric to show the other side of the pipe.

Fig. 24.39 shows the same path extruded pipe with a hollow center, to actually make it a pipe. This was done by going back to the original profile and simply adding another (smaller) circle profile and extruding it along the same path. Then, a subtraction was done, subtracting one extrusion from the other. As an exercise, go ahead and recreate this yourself.

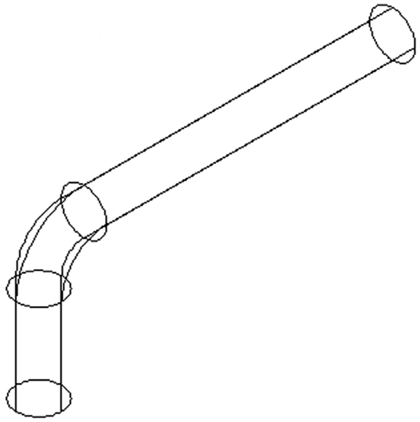


FIGURE 24.37 Path extrusion of the pipe.

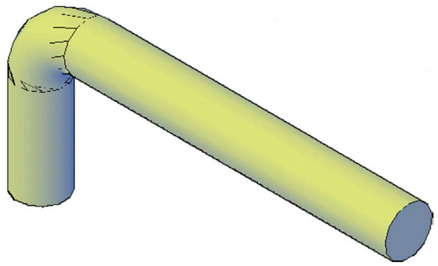


FIGURE 24.38 Path extruded pipe, shaded, and colored.

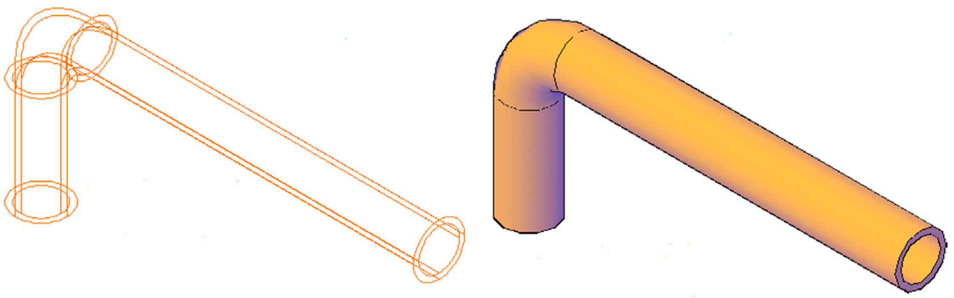


FIGURE 24.39 Path extruded pipe with hollow center.

Sweep

As already mentioned, sweep is a variation on the same theme but with some enhancements. Once again, we have a profile and a path, but instead of carefully lining up the circle to the path, we can select almost any profile, anywhere on the screen, and click on almost any path to generate a sweep. Therefore, it is a much more flexible command. Fig. 24.40 shows an example with a rectangle used to create a sweep along essentially the same type of path. Notice the profile is nowhere near the path. Go ahead and set this up via the following steps:

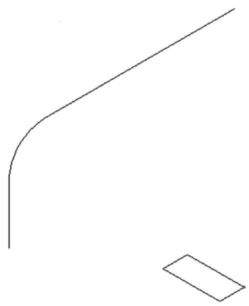


FIGURE 24.40 Rectangle and sweep path.

Keyboard: Type in **sweep** and press Enter.
Cascading menus: **Draw→Modeling→Sweep**
Toolbar icon: **Modeling** toolbar→
Ribbon: **Solid** tab→**Solid**

Step 1. Start the sweep command via any of the preceding methods.

- AutoCAD says:

```
Current wire frame density: ISOLINES = 4, Closed profiles creation mode = Solid
Select objects to sweep or [MOde]:_MO
Closed profiles creation mode [SOlid/SUrface] <Solid>:_SO
Select objects to sweep or [MOde]:
```

Step 2. Pick the rectangle and press Enter.

- AutoCAD says: `Select sweep path or [Alignment/Base point/Scale/Twist]:`

Step 3. Select the path and the sweep is produced, as seen shaded and colored in the SE Isometric view in Fig. 24.41.

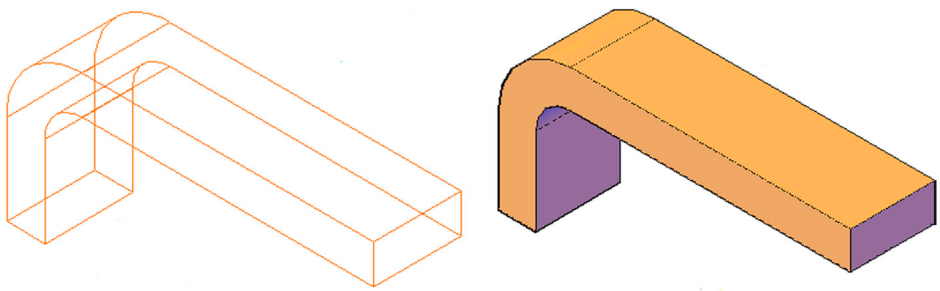


FIGURE 24.41 Rectangle sweep.

There are some limits to this, of course, and those have to do mostly with the size of the profile or path. If the profile does not physically fit the path, AutoCAD tells you that it is unable to produce the sweep. Note that the sweeping action destroys the original profile, so if you think you may need it again, keep a copy of it. This is an important point in creating multiple sweeps later on.

A major advantage of this command over path extrusion is that a sweep can be produced even if the profile is on the same plane as the path, as seen with an arc and a circle in Fig. 24.42.

The sweep command is not bothered by the similar plane. It just takes whatever profile is selected and applies it perpendicularly to whatever path you choose.

This concludes the advanced 3D tools. Practice and learn to apply everything thus far. We have an additional chapter with a few more tools to complete your 3D construction knowledge, but the ones presented in this chapter are the primary ones.

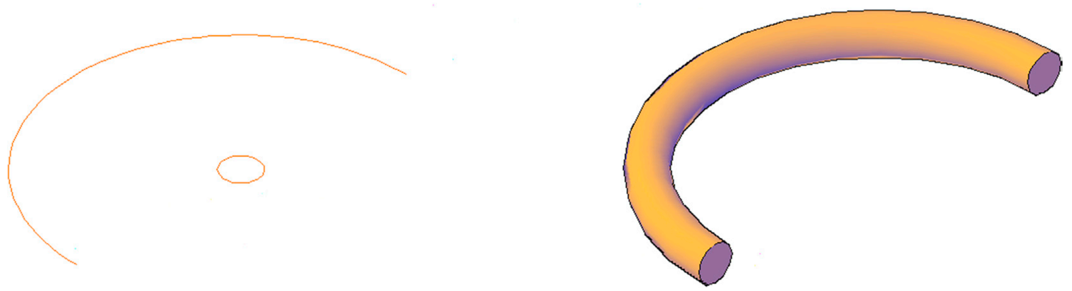


FIGURE 24.42 Circle sweep.

Drawing Challenge: Helical Coil

Using what you know so far, draw the helical coil shown in Fig. 24.43. You have all the tools at your disposal at this point. Think carefully, and do not worry about size or how many coils to do, only the shape. The answer is featured at the end of the next chapter, but do not look it up right away; give it your best shot first. Also, do not be tempted to use AutoCAD's helix tool, which is covered in the next chapter. It is relatively new (introduced in AutoCAD 2007), and the results may not necessarily look like the example shown here anyway. There is a reason why you should try to figure this out, and it is mentioned along with the answer in the next chapter. Good luck.

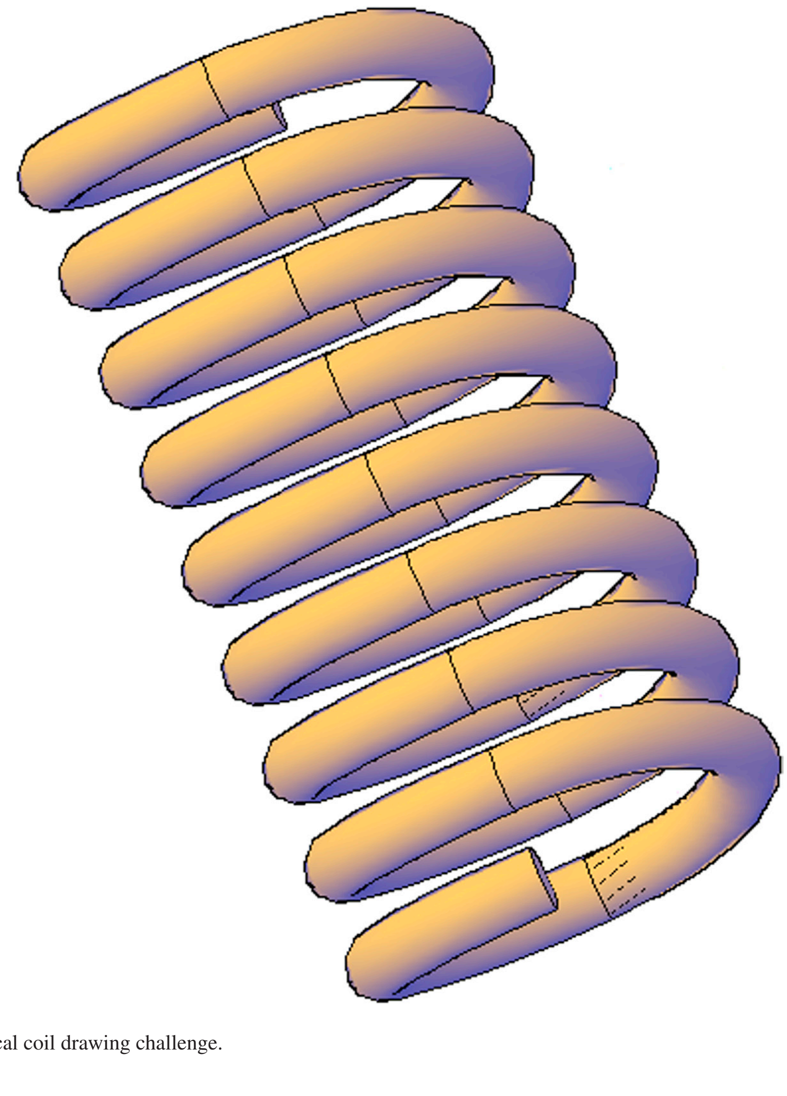


FIGURE 24.43 Helical coil drawing challenge.

24.2 SUMMARY

You should understand and know how to use the following concepts and commands before moving on to Chapter 25, Advanced Solids, Faces, and Edges:

- Revolve
- Shell
- Taper
- Loft
- Path extrusion
- Sweep

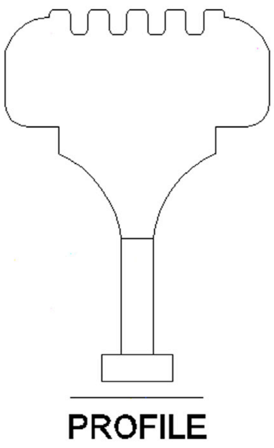
Review Questions

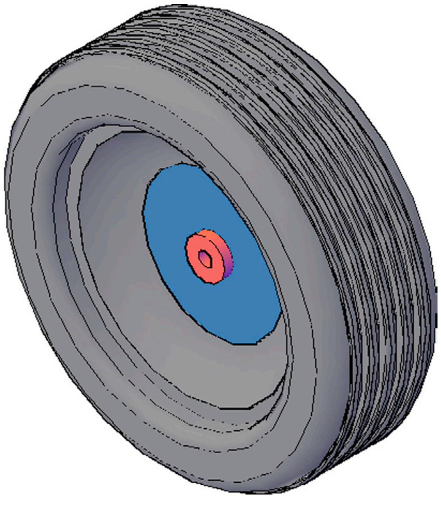
Answer the following based on what you learned in this chapter:

1. Describe the revolve command.
2. Does the profile need to be open or closed? Describe the effect of each.
3. Describe the shell command.
4. How many surfaces can you select with shell?
5. Describe the taper command.
6. Describe the loft command.
7. Describe the path extrusion command.
8. Describe the sweep command.
9. Why is sweep better than path extrusion?

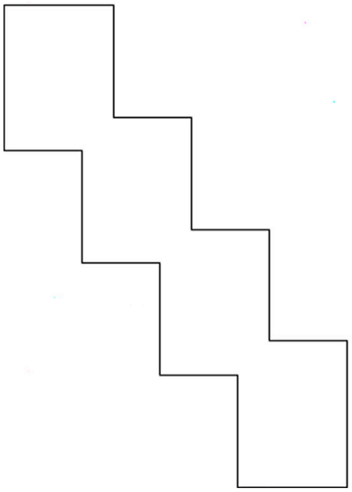
Exercises

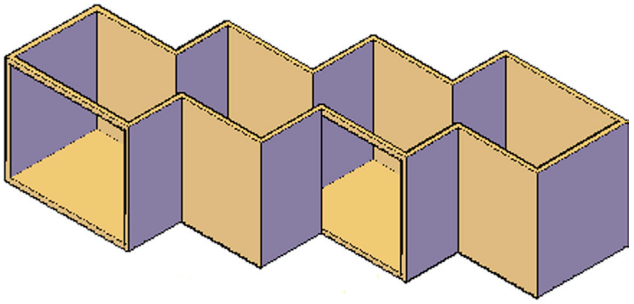
1. Use the following profile and axis of rotation (the line below the profile) to revolve it 360° to create the model of a basic tire and rim. Sizing and exact profile shape is up to you, and you need various drawing and editing commands, including pedit, prior to revolve. (Difficulty level: Moderate; Time to completion: 25 minutes.)



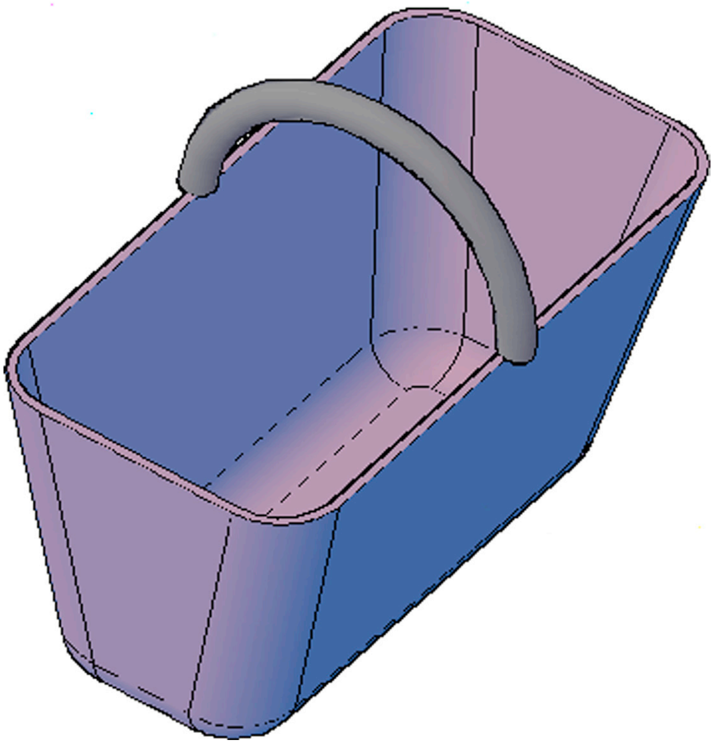


2. Create four 4.5″ × 6″ rectangles, then explode, trim, and pedit them together into an outline as shown in the profile on the left. Next, extrude them to a height of 5.5″ and shade and color them. Finally, use the shell command (0.2″ thickness) to remove the faces, as shown in the profile on the right. (Difficulty level: Moderate; Time to completion: 10 minutes.)

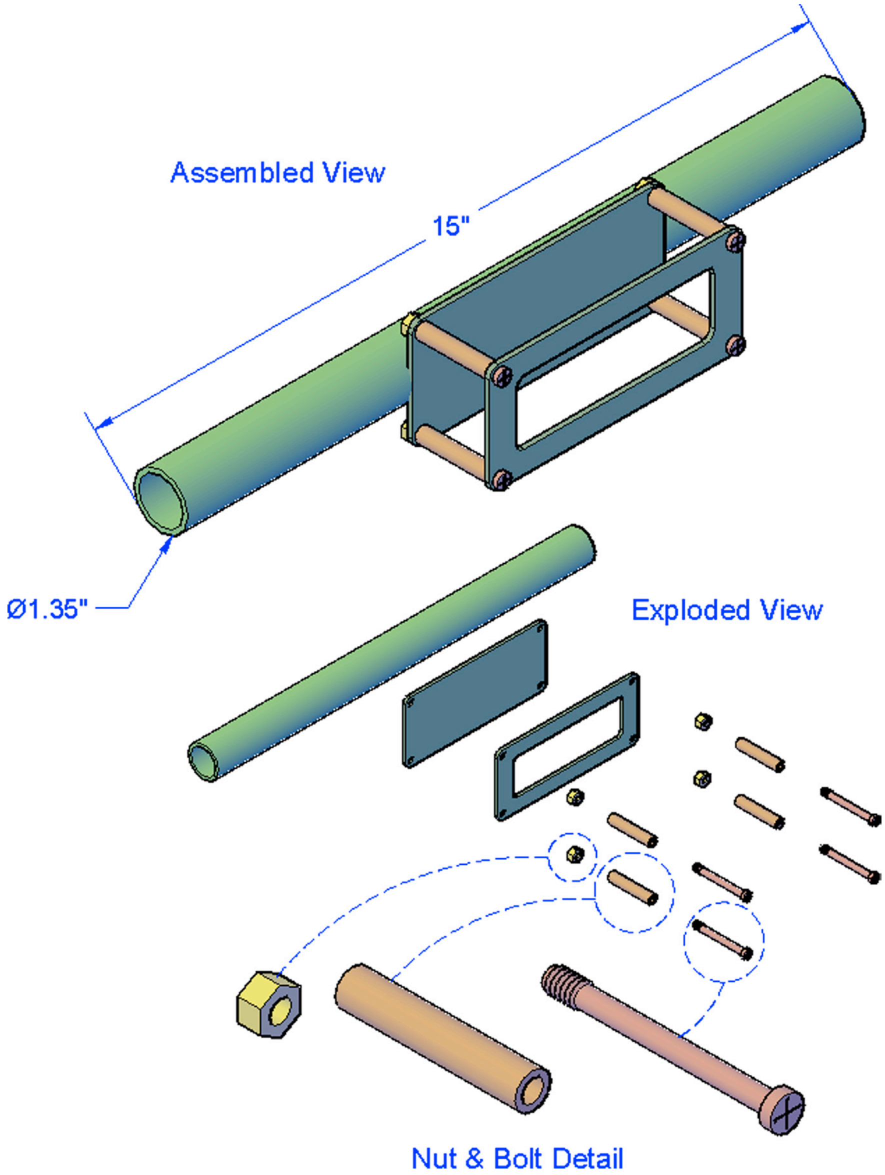




3. Create the following basket using the rectangle (20″ × 10″), extrude (10″), taper (10°), fillet (1.5″), and shell (0.25″) commands. Then, add the handle using arc and sweep (1″ diameter circle). (Difficulty level: Moderate; Time to completion: 15–20 minutes.)



4. Create the following bracket and pole model. An exploded view as well as a breakdown of some individual parts are shown for clarity. A pair of overall dimensions is given, but the rest of the sizing is up to you. (Difficulty level: Moderate; Time to completion: 30–40 minutes.)



5. Create the following simplified fire extinguisher model. The steps involved, including the overall sizing, are shown for clarity, but you have to improvise the rest of the details. You will need pline, pedit, revolve, shade, subtract, fillet, sweep, and other commands. (Difficulty level: Moderate/Advanced; Time to completion: 60 minutes.)