Compelling Features of 3D Solid Modeling in AutoCAD®

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AC5675-L This intermediate-to-advanced hands-on lab offers AutoCAD 3D veterans a chance to explore using solids to create compelling 3D part models in AutoCAD 2012. Tools for 3D solid with surface model creation will accelerate your design workflow! Learn new techniques that will supersede your old-school techniques, and develop a new level of understanding for creating and editing 3D models. We will explore user coordinate system flexibility, sweeps, extrudes, lofts, presspull, helix, surface slicing, and the use of gizmos to edit your 3D solid models. We will also create a simple assembly. If you used AutoCAD 3D in the past, attend this class, and get ready to be surprised!

Learning Objectives
At the end of this class, you will be able to:
- Explain the flexibility of the user coordinate system
- Create 3D solid models
- Use surfaces as slicing tools
- Edit 3D models

About the Speaker
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1. Techniques, Techniques, Techniques

In AutoCAD 2012, new 3D techniques are added to AutoCAD’s 3D world! If you are an experienced AutoCAD 3D user, you will really appreciate the new 3D commands and features in AutoCAD 2012. Use these new techniques. Remember, to set your Workspace to 3D Modeling and turn on Selection Cycling.

Turn on Selection Cycling!

When selecting objects that are on top of each other or occupy the same space, selection cycling is the preferred method to select one of the objects. When editing, you may need to erase, move, or copy one of the objects that overlap in order to select the correct object. The [Shift] key and spacebar can be pressed at the same time to cycle through objects at a pick point. When you need to cycle through objects:

1. At the “select objects” prompt, hold down the [Shift] key and spacebar, then click to select the object you want.
2. Keep clicking until the object you want to select is highlighted.
3. Press the [Enter] key.

You can use the SELECTIONCYCLING system variable to turn on selection cycling instead of using the [Shift] key and spacebar. The Selection Cycling button on the status bar is used for toggling selection cycling. There are three settings for the system variable:

• Off (0).
• On, but the list dialog box does not display (1).
• On and the list dialog box displays the selected objects that can be cycled through (2).

It is recommended that you turn on selection cycling!

New Direct Manipulation of the UCS

In AutoCAD 2012 the UCS has been enhanced to allow direct manipulation. Simply put, you can use the new multi-function grips to change the UCS. Move the origin, rotate the UCS around the X, Y or Z axis using grips, align the UCS with other objects including curves and solids. Select the UCS icon and then hover over the center grip. You have three selections to choose.
Select the UCS icon and hover over one of the three multi-function grips. You have three selections for each multi-function grip per axis.

If you have the Properties palette open, you can set various options.
If the UCS dialog box is opened from the Coordinates Panel of the View tab, you can also set various options.

![UCS dialog box](image)

2. Array using 3D Objects

An array is an arrangement of objects in a 2D or 3D pattern. An array can be created as a rectangular, polar, or path array. A 2D array is created on the XY plane of the current UCS. A 3D array is an arrangement of objects in 3D space. The ARRAYRECT, ARRAYPOLAR, and ARRAYPATH commands can be used to create both 2D and 3D arrays. These commands provide the same functions as the Rectangular, Polar, and Path options of the ARRAY command and are available in the drop-down menu located in the Modify panel of the Home tab on the ribbon.

An array can be created as an associative or non-associative array. Creating an associative array creates an array object, which can be modified as a single entity. For example, you can edit the source object to change all of the items in the array at once. You can also perform other modifications, such as deleting one or more items in the array, while maintaining the associativity of the arrayed items.

When creating a 3D array, the information you specify depends on the type of array being created. Many of the options are similar to those used when creating a 2D array. The following sections discuss 3D rectangular, polar, and path arrays.
3D Rectangular Arrays

In a 3D rectangular array, as with a 2D rectangular array, you must enter the number of rows and columns. However, you must also specify the number of levels, which represents the third (Z) dimension. The command sequence is similar to that used when creating a 2D array.

An example of where a 3D rectangular array may be created is the layout of structural columns on multiple floors of a commercial building. You can see two concrete floor slabs of a building and a single structural column. It is now a simple matter of arraying the column in rows, columns, and levels.

Open: Rectangular Array.dwg.

Use the ARRAYRECT command to select the one vertical column.

Follow the command prompt sequence as shown below.

1. Specify opposite corner for number of items or [Base point/Angle/Count] <Count>: Enter

2. Enter number of rows or [Expression] <4>: 3

3. Enter number of columns or [Expression] <4>: 5

4. Specify opposite corner to space items or [Spacing] <Spacing>: enter

5. Specify the distance between rows or [Expression] <1'-6”>: 10'

6. Specify the distance between columns or [Expression] <1'-6”>: 10'

7. Press Enter to accept or [ASsociative/Base point/Rows/Columns/Levels/eXit]<eXit>: AS

8. Create associative array [Yes/No] <Yes>: Enter

9. Press Enter to accept or [ASsociative/Base point/Rows/Columns/Levels/eXit]<eXit>: L

10. Enter the number of levels or [Expression] <1>: 2

11. Specify the distance between levels or [Total/Expression] <15'>: 12'8"

12. Press Enter to accept or [ASsociative/Base] Enter

13. Do not save the drawing.
Double click on one of the associative array columns and change the Columns and Column spacing as shown in the properties palette. (8 columns, 5’6” Column spacing)

By default, an associative array is created. You can create a non-associative array by selecting the **Associative** option.

As shown in the previous sequence, when setting the distance between rows, you can also define an elevation increment between rows when the Specify the incrementing elevation between rows or [Expression] <0”>: prompt appears. The elevation increment is different from the distance between levels. The elevation increment sets the spacing between rows along the Z axis so that each successive row is drawn on a higher or lower plane. This option can also be used when creating a 3D polar array.

**3D Polar Arrays with Sub-Object editing**

A 3D polar array is similar to a 2D polar array. However, the axis of rotation in a 2D polar array is parallel to the Z axis of the current UCS. In a 3D polar array, you can define a centerline axis of rotation that is not parallel to the Z axis of the current UCS. In other words, you can array an object in a UCS different from the current one. In addition, as with a 3D rectangular array, you can array the object in multiple “levels” along the Z axis. The `ARRAYPOLAR` command can be
used to create a 3D arrangement in rows, levels, or both rows and levels.

The **Base point** option is used to define the base point for the array. By default, the base point defined by AutoCAD is the centroid of the object(s) selected. You may want to select a more logical base point, such as an endpoint of an edge or the center point of a circular face. If any of the properties of an associative array require changes, use the **Properties** palette to edit the array. The arrayed items will update based on the changes.

To create a 3D polar array, select the **ARRAYPOLAR** command. Select the object to array and press [Enter]. Then, pick the center point of the array or use the Axis of rotation option to select a centerline axis. The Axis of rotation option allows you to select a centerline axis that is different from the Z axis of the current UCS. Using this option requires you to pick two points to define the axis. The leg attached to the hub must be arrayed about the center axis. The center axis is drawn as a construction line. Use the Axis of rotation option to pick the two endpoints of the axis. Once you define the axis, items in the array generate dynamically. You can then use the command options to adjust the array. If the design intent is to add or subtract the legs and feet, use the **ASsociative** option to make the array associative.

**OPEN: BaseSupport.dwg.**

Use the **ARRAYPOLAR** command to create the array.

1. Select the leg. Enter.
2. Specify center point of array or [Base point/Axis of rotation]: A
3. Specify first point on axis of rotation: **Select one end of the centerline.**
4. Specify second point on axis of rotation: Select the other endpoint of the center line.

5. Enter number of items or [Angle between/Expression] <4>: 5

6. Specify the angle to fill (+=ccw, -=cw) or [EXpression] <360>: 360

7. Press Enter to accept or [ASsociative/Base point/Items/Angle between/Fill angle/ROWs/Levels/ROTate items/eXit]<eXit>: AS

8. Create associative array [Yes/No] <Yes>: Enter

9. Press Enter to accept or [ASsociative/Base point/Items/Angle between/Fill angle/ROWs/Levels/ROTate items/eXit]<eXit>: Enter

10. The part has five legs as shown in Figure 2.

11. Set the Visual Style to 3D Wireframe.

12. Select the associative array of the legs. Notice the new Array tab in the ribbon.

13. In the Items panel, change the number of Items to 6.

14. In the Options panel, select Edit Source and select the array, then select the cylinder in the foot. Use CTRL and pick the cylinder in the source leg. (This method of selecting the cylinder in the foot by using the CTRL key and picking the cylinder is called “Sub-object editing”)

15. Use one of the parameter grips to stretch the cylinder. Type .0625 as the radius stretch distance. In the Edit Array panel, Save Changes to the array. What happens to the other cylinders in the associative array?
16. Thaw the New_Leg layer.

17. Select the associative array. In the Options panel, select Replace Item and select the new leg to the right. Press enter to except the new item. Then press Enter to except the default centroid. Type S to select the Source item in the array to replace. Press Enter to end the command. Press Esc. As shown below, you have replaced the source legs with the new leg!

18. **UNION** the associative array of the legs and feet to the hub. What happens?

19. **Explode** the associative array of the legs and feet to the hub.

20. **UNION** the associative array to the legs and feet to the hub. What happens?

21. Do not save the drawing.

The **Rows** option is used to create multiple rows of arrayed objects. After selecting this option, enter the number of rows and the distance between rows. Then, set an elevation increment value to control the spacing along the Z axis between each successive row. An array of seats in a theater can be created in this manner. **Figure 3** shows an example of arraying a single seat to create multiple rows of seats. To create this array, select the **ARRAYPOLAR** command, select the first seat, and use the following command sequence. If needed, use the **Associative** option to create an associative array. For this example, the default **Center point** option is used to set the center point of the array. Using this option is sufficient because the Z axis of the current UCS is parallel to the required axis of rotation.
OPEN: 3D Chairs.dwg.

Use the ARRAYPOLAR command to create the array.

1. Select the 3D Chair. Enter

2. Specify center point of array or [Base point/Axis of rotation]: (Use the Center object snap to pick the center point of the top arc of the first platform)

3. Enter number of items or [Angle between/Expression] <4>: 8

4. Specify the angle to fill (+=ccw, −=cw) or [EXpression] <360>: 80

5. Press Enter to accept or [ASsociative/Base point/Items/Angle between/Fill angle/ROWS/Levels/ROTate items/eXit] <eXit>: ROWS

6. Enter the number of rows or [Expression] <1>: 6

7. Specify the distance between rows or [Total/Expression] <current>: 96

8. Specify the incrementing elevation between rows or [Expression] <0.0000>: 7

9. Press Enter to accept or [ASsociative/Base point/Items/Angle between/Fill angle/ROWS/Levels/ROTate items/eXit] <eXit>: Enter

10. Do not save the drawing.

The result is shown below. Notice that each successive row of seats is situated on a higher plane. In more complex models, the Levels option can be used to create multiple levels of rows.
3D Path Arrays

A **3D path array** is similar to a 2D path array. Objects can be arrayed along a path or a segment of a path. The path, also called a *path curve*, can be a line, circle, arc, ellipse, spline, polyline, helix, or 3D polyline. As with a 3D polar array, you can create a 3D arrangement in rows, levels, or both rows and levels. A 3D path array can be created as an associative or non-associative array.

To create a 3D path array, select the **ARRAYPATH** command. Select the object to array and press [Enter]. You are prompted to select the path curve. The object to be arrayed does not have to intersect the path curve. Once you select the path curve, items in the array generate dynamically as you drag the cursor. As an option, you can select the **Orientation** option to set the array’s base point, orientation, or both. The default base point of the array is the endpoint of the path curve closest to where you select it. This point serves as the start point of the array. Depending on the result you want, you can select a different base point (start point), such as a point on the object. The default orientation of the array is the current orientation of the object. The **2 Points** option can be used to pick two points to define a different orientation. The **Normal** option can be used to align the object “normal” to the path. Using this option aligns the Z axis of the object perpendicular to the path.

After specifying the base point and orientation of the array or using the defaults, continue as follows. You can use the command options to adjust the number of items to be arrayed and the distance between items. As with a 2D path array, you can distribute the object along the path evenly by using the **Divide** option. You can also specify the total distance between the first and last objects, or the distance between each object.
The **Rows** and **Levels** options allow you to create an arrangement in rows and levels. The options are similar to those used with the **ARRAYPOLAR** command.

**OPEN: Ramp.dwg**

1. Use the **ARRAYPATH** command to array 310 ramp guards (the pole and the barrier frame) along the helical path. Do NOT create an associative array. Divide evenly along the path.
2. Modify the array as needed.
3. Do not save the drawing.
3. Simple 3D Primitives and 2D Polyline Extrusions

OPEN: SlideBase.dwg

You will edit the two small cylinders and the two fillets as shown above.

1. Double click on the 3D solid and open the Properties palette. **Turn Show History to Yes.** What do you see that is unique to the solid?
2. Turn Show History to No.
3. **CTRL** and pick one of the small cylinders and open the Properties palette to change the radius to 4.000. Do the same for the other cylinder.
4. **CTRL** and pick the round on the back of the part. Change the radius of the fillet to 4.000.
5. Click on the solid. In the Properties palette, turn **History to None**.
6. Add a **fillet with a 5 unit radius** to the top left side as shown.
7. **CTRL** and pick one of the small cylinders and open the Properties palette to change the radius to 2.000. What happens? Do the same for the other cylinder. What happens? Try the fillets?????
8. Do not save the drawing.

4. A special note on the DELOBJ system variable

**From the AutoCAD Help menu**

When you create a unique profile for extrudes, sweeps, revolves and lofts you may want to keep the unique geometry for future editing purposes. Set the DELOBJ variable as noted below. DELOBJ control whether the geometry is retained or deleted.

0  All defining geometry is retained.
1. Profile curves are deleted, including those used with the EXTRUDE, SWEEP, REVOLVE, and LOFT commands. Cross sections used with the LOFT command are also deleted.

2. All defining geometry is deleted, including paths and guide curves used with the SWEEP and LOFT commands.

3. Prompt to delete profile curves, including those used with the EXTRUDE, SWEEP, REVOLVE, and LOFT commands. Cross sections used with the LOFT command are also deleted.

4. Prompt to delete all defining geometry, including paths and guide curves used with the SWEEP and LOFT commands.

5. Creating Models from 2D Profiles - Loft, Shell, Sweep, Slice and Helix

Example: The LOFT command is used to create freeform shapes. Lofts are created by selecting multiple cross sections causing the model to be created based on transitional size, shape and form from one cross section to another. Closed loop and open loops can be used as cross sections. Guide rails and other options can also influence the lofted shape.

OPEN: Shaver-Loft.dwg
OPEN: Hair Dryer OPEN: Hair Dryer-Loft-Shell.dwg

You can create a shell from your 3D solid object. Hollow out your object! New faces are created by offsetting existing ones inside or outside the model.

1. Loft the Handle first using the centerline as your path. Slice the handle in half and Shell the handle. (We are using .100 shell thickness for this example)

2. Use the LOFT command and loft the main body using the pre-defined cross sections while setting the surface control Normal to the start.
3. Slice then Shell the main body at the handle intersection. Union the parts together. (Use .100” for shell thickness for this example) Make sure you select the main body and the front opening of the hair dryer to remove faces. If you do not select the hole opening as part of your selection to remove the faces you will not have a hole to let the air flow through the hair dryer. In the example below, the open hole face was not selected. How would the air flow through the hair dryer?
4. Add fillets and create the vents at the back of the hair dryer.

**Solids and Surfaces**

Use for the Loft tool is to create a surface for use as a cutting tool. Loft 2 cross sections and 2 guides to create a LOFTEDSURFACE to use as a cutting tool.

**OPEN: Slice-Lofted-Surfaces.dwg**

Use the **SLICE** tool to slice away the top of the box using the LOFTEDSURFACE as the cutting tool.
Use Surfaces and solids to create composite solids.

OPEN: Screw Driver-Lofted-Surfaces.dwg

1. We start by creating a revolved surface.

2. **Sweep** the main part of the handle. Use the revolved surfaces as the slice tool.

3. **Loft** the blade using the rails as your guides.
4. Create **Extruded surfaces** to create the final end shape of the blade.

5. **Slice** the solid blade using the extruded surfaces as cutting tools.

**HELIX and SWEEP commands**

The HELIX command is similar to a Sweep and Loft. A helix can use different paths as shown below. When creating a helix you, specify the diameter of the base of the helix, specify the diameter of the top of the helix, specify the endpoint location for the helix axis, specify the number of turns (revolutions) for the helix, (the number of turns for a helix cannot exceed 500), specify the height of one complete turn within the helix, specify distance between and specify whether the helix is drawn in the clockwise (CW) or the counterclockwise (CCW) direction. Create the Helix and sweep the profile.

The **SWEEP** command creates a more free formed shape. Swept models have a planar shape that follows a defined path that was created by another piece of geometry. Solid sweeps are created using closed loop object. Open loops create surface sweeps. The handle and blade used the PRESSPULL or the EXTRUDE command to add 3D thickness.

**Creating a spring between two caps.**

**OPEN:** Spring Caps.dwg

Creating threads using the **HELIX** and **SWEEP** commands, then subtract the sweep.
OPEN: Helix with Threads.dwg

1. Create the solid shaft.
2. Create the thread profile based on current industrial standards for each unique thread design.
3. Sweep the profile.
4. Subtract the swept profile.

Modeling helical shapes in AutoCAD 2012 is easy. We are going to create this spiral Star using the HELIX command.

OPEN: Star.dwg.

1. Type DELOBJ and set to 0.
2. Make the Visible layer current.
3. The circle with a diameter of 7 and the two lines has been drawn for you. (The angle comes from the formula $90^\circ + 360^\circ/5$)
1. Type **UCSICON** and turn the Ucsicon off.
2. Set your view to the Home position.
3. Create a 2 unit vertical line and angled line as shown. (0,0,2) (Star1.dwg)

4. Turn the **UCSICON** on. Make the Helix Layer current. Create a Helix with the base centered at point A, the radius at the intersection at point B. The top radius is 0. Turns 1/5 and the turn Height to the 2 unit axis height at point C.

5. Set the **UCS** at the beginning of the helix as shown. **Use the ZAxis Object method.** (Star1a.dwg)
6. Draw a line 2 units starting at the end of the helix in the –Y direction. (0,-2,0)
7. Rotate your view as shown. (Star2.dwg)

8. Use the **ROTATE** command and copy the 2 unit line to 45° and -45°. Use the **LINE** command and connect the endpoints. Use the **PEDIT** command and Join the three lines to create a polyline that forms the triangle as shown. (Star3.dwg)

9. Type **UCS** and set to World. Set the current Layer to Section. **EXTRUDE** the 162° line, 5 units high as shown.
10. Type **PLANESURF**, Object option and select the circle. Set the Visual Style to Conceptual. (Star4.dwg)

![Diagram showing the use of PLANESURF](image1)

11. Set the current layer to Visible. **SWEEP** the triangle along the helix as shown. (Star5.dwg)

![Diagram showing the use of SWEEP](image2)

12. Type **SLICE** and use the two surfaces as your slicing tools. Keep the swept solid as shown. (Star6.dwg)

![Diagram showing the use of SLICE](image3)

13. Erase the two magenta surfaces. Any and any remaining construction geometry.
14. Create an associative **Polar Array** of the solid with 5 copies.

15. **Edit the associative array to 4 copies.**
16. **UNION** the solids together.

**Intersection**

Creating an intersection is one of the most difficult concepts to grasp. The resultant of two or more solids intersected creates a new solid. Special thanks to Dr. Joanne R. Reid, Vice President of Corporate Development Associates, Lombard, Illinois for the initial concept.

**OPEN: Intersection Blade.dwg**

1. Create a CYLINDER at 0,0 with a diameter of 14 and a height of 7.
2. Create another cylinder at 0,0 with a diameter of 14.72 and a height of 7.
3. Subtract the small cylinder from the larger cylinder.
4. Set the Visible layer current. Extrude the blade with a height of 15.
5. Use the **INTERSECT** command and select the cylinder and the blade.

The finished blade model.

**Aligning Objects in 3D**

**OPEN: Align Parts.dwg**

AutoCAD provides two different methods with which to move and rotate objects in a single command. This is called **aligning** objects. The simplest method is to align 3D objects by picking source points on the first object and then picking destination points on the object to which the first one is to be aligned. This is accomplished with the **3DALIGN** command, which allows you
to both relocate and rotate the object. The second method is possible with the **ALIGN** command. The **ALIGN** command aligns 2D or 3D objects by selecting three sets of alignment pairs. You first select the source object, then the first source point, and finally the first destination point on the destination object. This technique is repeated two more times to align one object to another. This method of aligning allows you to not only move and rotate an object, but scale the object being aligned. The preferred method for 3D assembly creation is to use the **3DALGIN** command.

For his assistance with some of the above topics in this paper

**Special thanks to Dr. J.D. Mather, Assistant Professor of CAD and Product Design**

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