Compelling Features of 3D Solid Modeling in AutoCAD® 2013

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**AC2287-L**  This intermediate-to-advanced hands-on lab offers 3D AutoCAD® software veterans as well as AutoCAD® 2D users a chance to explore using solids to create compelling 3D part models in AutoCAD 2013. Tools for creating 3D solids with surface modeling 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, loftes, extrudes, PressPull operations, helix, surface slicing, and the use of gizmos to edit your 3D solid models. We will also edit models in Autodesk® Inventor® Fusion software. We will create assemblies by aligning solid parts. 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:

- Create 3D solid models
- Use surfaces as slicing tools
- Edit 3D models
- Create a 3D assembly

**About the Speaker**
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1. **Set your Tools**

1. Set your workspace to 3D modeling.

![Workspace Setting](image)

2. **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!
3. **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.

4. Turn on **SOLIDHIST**. Solid history is very important to achieve design intent. It controls whether new composite solids retain a history of their original solid components. You WILL be editing solids during a design cycle. Set the **SOLIDHIST** variable to 1. This sets the History property to Record for new solids created and will retain a history of the objects created. You can also turn on Solid History from the Primitive panel on the Solid tab.

5. Turn on **CULLING**. From the Selection panel on the Solid tab turn on Culling so objects that are hidden form the current view (i.e. in the background) are highlighted when you hover over the solid.
2. **New Direct Manipulation of the UCS**

In AutoCAD 2012/2013 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.

**OPEN: Dynamic-UCS.dwg**

Position the UCS by using the UCS grips. Set the XY plane parallel to the inclined surface face.
3. **Xedges – Projecting Geometry**

The **XEDGES** command creates copies of, or extracts, all of the edges on a selected solid. You can also select edges of a surface, mesh, region, or subobject. Once the command is initiated, you are prompted to select objects. Select one or more solids and press [Enter]. The edges are extracted and placed on top of the existing edges. (Note: If you are an Autodesk Inventor user, think of this as Projected Geometry). The new objects are created on the current layer. Straight edges and the curved edges where cylindrical surfaces intersect with flat or other cylindrical surfaces are the only edges extracted. Spheres and tori have no edges that can be extracted. The round bases of cylinders and cones are the only edges of those objects that will be extracted.

**OPEN: Xedges base.dwg**

We are going to project geometry to drill the holes in the top of the part. What we do not want to do is
just extend the cylinder from the bottom of the part into the top to subtract the top holes. Think of how Design Intent will have an effect on this part.

1. Change your visual style to 2D wireframe and use the XEDGES command to project the top of the two bottom cylinder edges. Use your [CTRL] key for selection cycling.

2. Set the visual style to X-Ray and select one of the red circles to project. Use the Gizmo to move the circle to the bottom of the top plate. Object snap to the endpoint (corner) of the top plate to set the circles position.
3. Use the **PRESSPULL** command to subtract the holes. **PRESSPULL OR EXTRUDE** to the endpoint of the corner of the top of the part. Reset your visual style to Conceptual. Do you need to subtract the new cylinders to create the holes? Maybe Yes, Maybe No! Depends on how you created the cylinders!

![Image](image.png)

4. **Projecting Geometry to create a duplicate image of a surface.**

   **OPEN: ABS_CAP-Base.dwg**

   We need to create a cap for this part. Set your in-canvas visual styles to Shades of Gray. Use the **XEDGES** command to project the top surfaces of the object.

   ![Image](image.png)

   1. Use the **XEDGES** command and select the top faces as shown below. Press and hold your [CTRL] key when selecting.
2. Set the CAP layer current.
3. Set your visual style to realistic. Turn selection cycling on.
4. **EXTRUDE** the outside top circle .500 units in the positive Z direction. (Upward).

![Image of extruded object](image1)

5. Freeze the Base layer.
6. Set your visual style to **2D Wireframe**.
7. **EXTRUDE** the (yellow) projected circles through the top of the cap (.500).
8. Use the SUBTRACT command to subtract the cylinders from the cap. (Could you have used PRESSPULL instead of EXTRUDE?)
9. Then re-set your visual style to **Conceptual**.

![Image of subtracted objects](image2)

10. Use the **PRESSPULL** command to subtract .250 into the CAP the project geometry that forms the three faces of the around the circle.

![Image of presspulled object](image3)

11. Thaw the Base layer.
12. Set the visual style to X-Ray to look inside the two parts.
13. Use the **FILLET** command with a radius of **.125** to add a round to the top edge of the top Cap.
14. Erase any leftover projected lines, circles or arc.

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

Example: The **LOFT** command is use 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-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 0.100" 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.

6. Solids and Surfaces

Use for the LOFT tool is to create a surface for use as a cutting tool. LOFT 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.

**6. 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**

1. Create a small circle next to the assembly and **SWEEP** the circle using the subcommand **Alignment>Yes** to align the circle along the helical path. Select the helical path as the sweep path.
Creating threads using the **HELIX** and **Sweep** commands, then subtract the sweep.

**OPEN:** Helix with Threads.dwg

1. We are going to cut thread to an M10 x 1.5.

2. Set the Helix Path layer current. Use the **HELIX** command and snap to the Center point of the right end of the shaft.
3. Then select the Intersection of the corner of the cutting tool as the base radius. Press enter to accept the same distance as the top radius.

4. Set the turn **Height** and enter a height of 1.5 for the pitch of the thread.
5. Select **Axis endpoint** and select the end of the centerline at the opposite end of the shaft.

6. Set the Threading Tool layer current. Use the **SWEEP** command and select the triangle as the cutting tool. Set **Alignment to No.** Then select the helical path.

7. Use the **SUBTRACT** command to subtract the Sweep of the threads from the shaft.
We are going to create this spiral Star using the **HELIX** and **SWEEP** commands.

**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. **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 20 unit axis height at point C. (Star2.dwg)

5. **Set the UCS at the beginning of the helix as shown. Use the ZAxis > Object method.** (Star3.dwg)

6. **Set the layer Wires current.** 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.** (Star3.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. (Star4.dwg)

![Image](Star4.dwg)

9. Set the current layer to Star. **SWEEP** the triangle along the helix as shown. Set the Visual Style to Realistic. (Star5.dwg)

![Image](Star5.dwg)

10. Type **UCS** and set to World. Set the current Layer to Surface. **EXTRUDE** the 162° line, 5 units high as shown. Type **PLANESURF**, Object option and select the circle. (Star6.dwg)

![Image](Star6.dwg)
11. Type **SLICE** and use the two surfaces as your slicing tools. Keep the swept solid as shown. (You may want to slice off the bottom first!) (Star7.dwg)

12. Erase the two green surfaces. And any remaining construction geometry.

13. Create an associative **Polar Array** of the solid with 5 copies. (or use **3DARRAY** to create the array)  (Star_Final.dwg)

14. **Edit the associative array to 4 copies.**

15. **UNION** the solids together. But what do you need to do to the associative array.

**7. Intersection**

**OPEN: Intersection_Curved_Hand_Shoovel.dwg**

1. Create a cylinder at 0,0 with a diameter of 6 and a height of -8. (Shovel1.dwg)
2. Set the current layer to Cylinder2. Create another cylinder at 0,0 with a diameter of 5.875 and a height of 8. (Shovel2.dwg)

3. Subtract the small cylinder from the larger cylinder. Set your visual style to Realistic. (Shovel3.dwg)

4. Set the Shovel Blade layer current. **EXTRUDE** the Shovel blade with a height of 5.

5. Use the **INTERSECT** command and select the cylinder and the shovel blade. (Shovel_Finished.dwg)
8. 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.
9. Editing the Solid

OPEN: BaseBracket-Fusion Detail.dwg

2. Open the Properties palette. Use sub-objecting editing techniques to edit the hole and the fillets. [CTRL] pick the feature. Make changes in the Properties palette.

3. Edit the other cylinders as shown. What happens?

4. Autodesk Inventor Fusion to the rescue! Launch Autodesk Inventor Fusion and select each circle one at a time. Use the PRESSPULL tool to change the diameters (radius x 2). Return to AutoCAD when finished editing.
5. Except the changes in AutoCAD. Why did we need to use Autodesk Inventor Fusion to edit our AutoCAD 3D model? ................. SOLIDHIST was set to zero (off). No solid history of the part to edit inside of AutoCAD.

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

Special thanks to Dr. J.D. Mather, Assistant Professor of CAD and Product Design
Pennsylvania College of Technology, Williamsport, Pennsylvania

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