AUTODESK UNIVERSITY 2013

Compelling Composite 3D Solid and Surface Modeling Techniques in Autodesk® AutoCAD® 2014

J.C. Malitzke - Digital JC CAD

AC2829-L This intermediate-to-advanced hands-on lab offers 3D AutoCAD software veterans as well as AutoCAD 2D users a chance to explore 3D solid and surface modeling techniques for creating compelling composite 3D solid and surfaced part models in AutoCAD 2014. Tools for creating 3D solids with surfaces 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. In this lab, we explore solid and surface extrudes, sweeps, lofts, and revolves. We also explore creating procedural, networked, blended, patched, and trimmed surfaces using surface slicing workflows. 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:

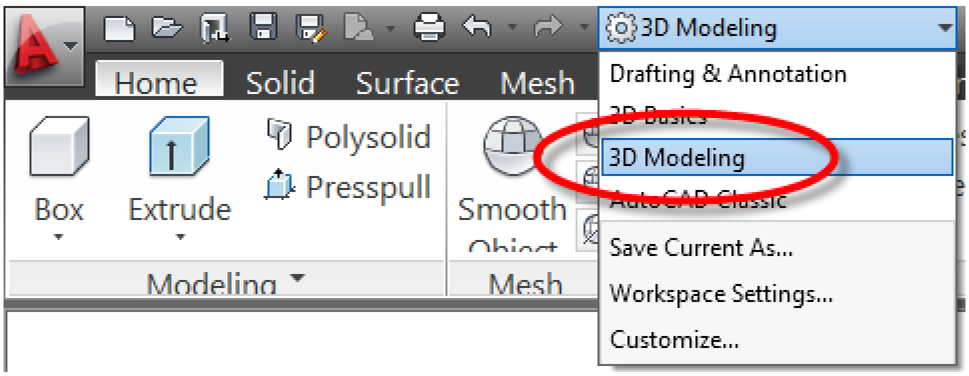
- Create 3D solid models
- Create 3D surface models
- Edit solid and surface models to create 3D composites
- Use surface slicing techniques to produce 3D composite 3D models

About the Speaker

J.C. Malitzke is President of Digital JC CAD Services Inc. and is the former department chair of Computer Integrated Technologies and a faculty member at Moraine Valley Community College in the greater Chicago area. He managed and taught for the college’s Autodesk Authorized Training Center. He has been using and teaching Autodesk® products for 28 years. J.C. is co-author to Good-Heart Wilcox Publisher for AutoCAD® and Its Applications Advanced. He is the recipient of several educator awards, including, Professor of the Year, and the Illinois Trustee Association's Faculty Member of the Year, and a top presenter award winner at Autodesk University. J.C. is a Certified Autodesk Instructor for AutoCAD® and Autodesk® Inventor and is an Autodesk Certification Evaluator. This is his 19th year presenting at Autodesk University. He holds a BS degree in education and a MS in industrial technology from Illinois State University. Contact J.C. at: digitaljc@digitaljccad.com

1. Set your Tools

a. Set your workspace to 3D modeling.



b. 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!

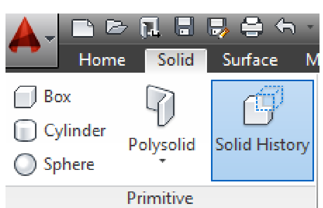
C. A special note on the DELOBJ system variable

From the AutoCAD Help menu

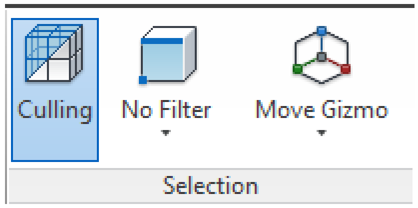
When you create a unique profile for extrudes, sweeps, revolves and lofts you may want to keep the unique geometry for future editing purposes. Set the DELOBJ variable as noted below. DELOBJ control whether the geometry is retained or deleted.

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.

2. 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.

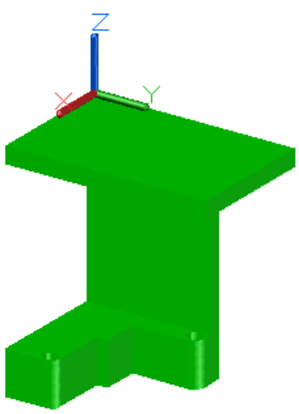


3. Turn on **CULLING.** From the Selection panel on the Solid tab turn on Culling so objects that are hidden form he current view (i.e. in the background) are highlighted when you hover over the solid.



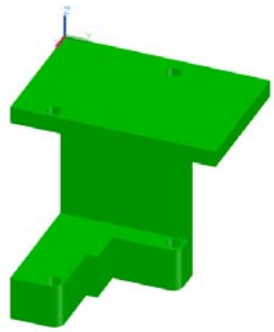
2. Xedges – Projecting Geometry - Solids

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 a 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.

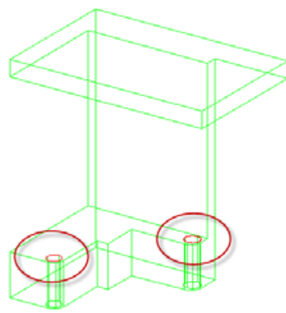


OPEN: Xedges base.dwg

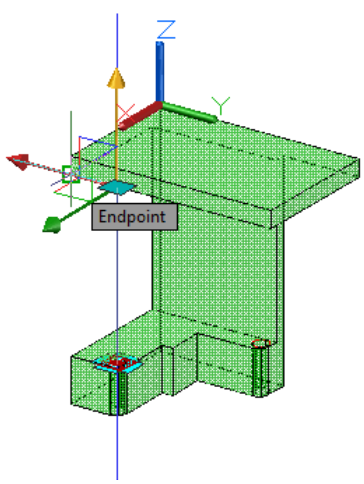
We are going to project geometry to drill the holes in the top of the part. What we do not want to do is just extend the cylinder from the bottom of the part into the top to subtract the top holes. Think of how Design Intent will have an effect on this part.

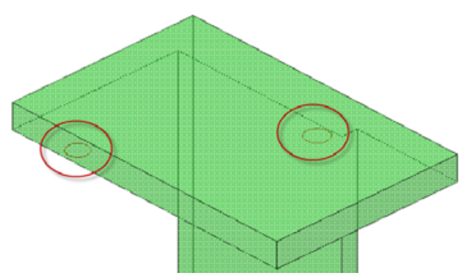


1. Change your visual style to 2D wireframe and use the **XEDGES** command to project the top of the two bottom cylinder edges. Use your [CTRL] key for selection cycling.



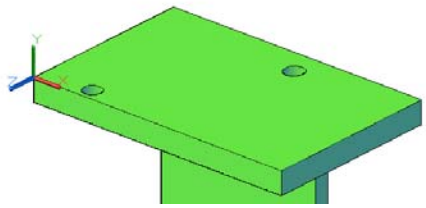
2. Set the visual style to X-Ray and select one of the red circles to project. Use the Gizmo to move the circle to the bottom of the top plate. Object snap to the endpoint (corner) of the top plate to set the circles position.





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

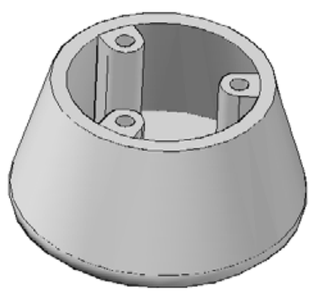
need to subtract the new cylinders to create the holes? Maybe Yes, Maybe No! Depends on how you created the cylinders!



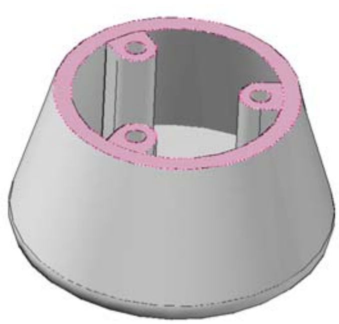
3. Projecting Geometry to create a duplicate image of a surface.

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.

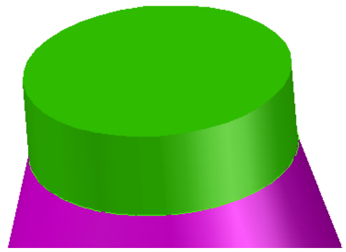


1. Use the **XEDGES** command and select the top faces as shown below. Press and hold your **[CTRL]** key when selecting.

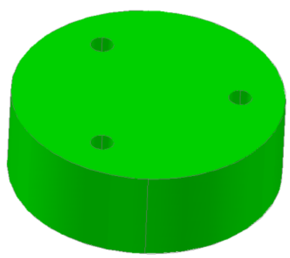


2. Set the CAP layer current.

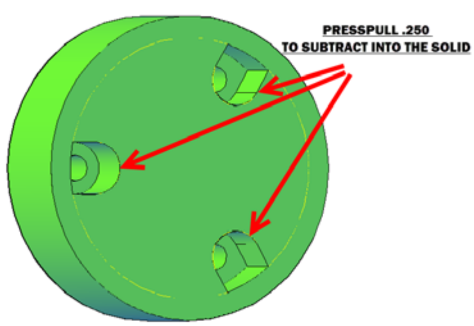
3. Set your visual style to realistic. Turn **selection cycling** on.
4. **EXTRUDE** the outside top circle .500 units in the positive Z direction. (Upward).



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**.

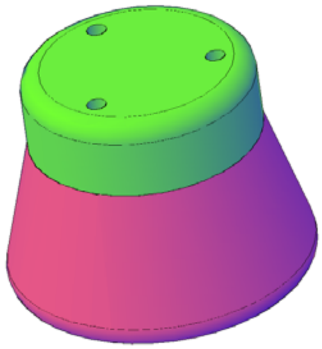


10. Use the **PRESSPULL** command to subtract **.250 into the CAP** the project geometry that forms the three faces of the around the circle.



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.

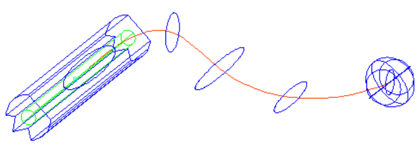
14. Erase any leftover projected lines, circles or arc.

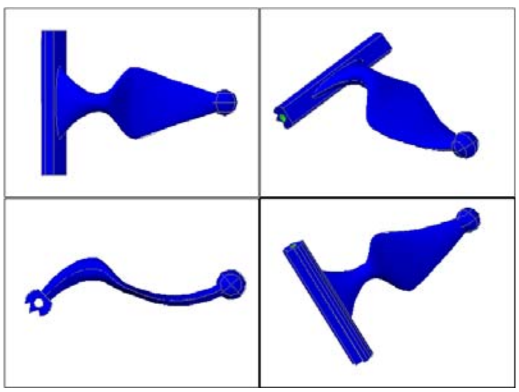


4. Creating Models from 2D Profiles - Loft, Shell, Sweep, Slice and Helix

A ***loft*** is an object created by extruding between two or more 2D profiles. The shape of the loft object blends from one cross-sectional profile to the next. The profiles can control the loft, or the loft can be controlled by one path or multiple guide curves.

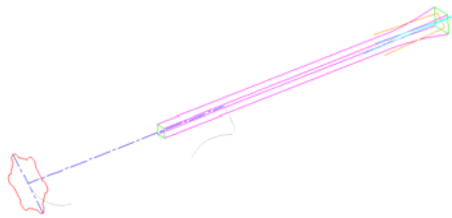
OPEN: Shaver-Loft.dwg



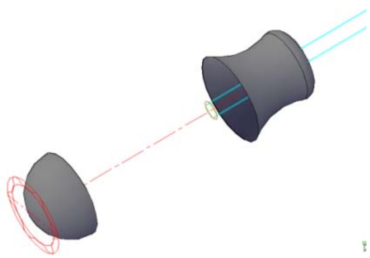


5. Solids and Surfaces

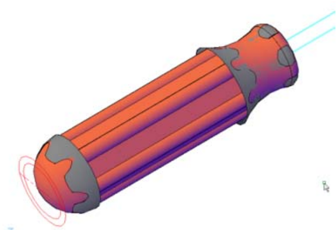
OPEN: Screw Driver-Lofted-Surfaces.dwg



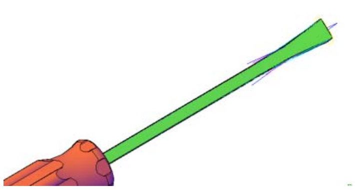
1. We start by creating a **revolved surface**.



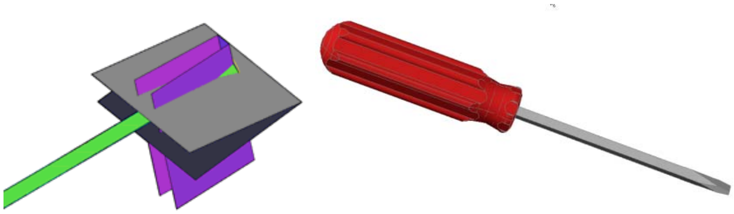
2. **SWEEP** the main part of the handle. Use the revolved surfaces as the slice tool.



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.

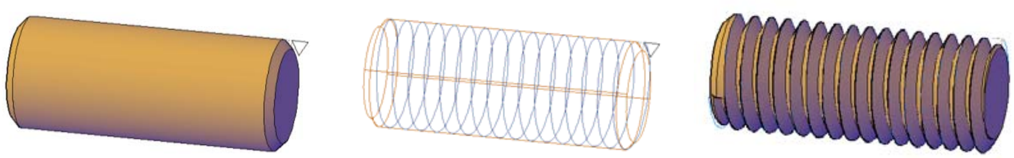


6. HELIX and SWEEP commands

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.

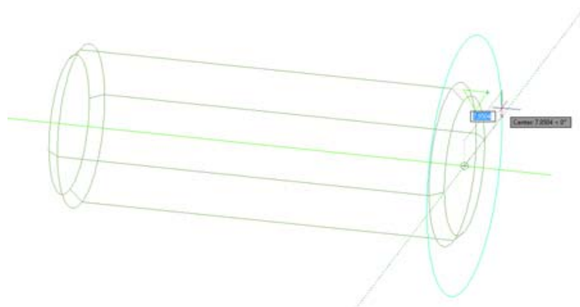
A ***sweep*** is an object created by extruding a single 2D profile along a path object. Sweeping an open shape along the path results in a surface object. If a closed shape is swept, a solid or surface object can be created. The **SWEEP** command creates a more free formed shape. Swept models have a 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.

OPEN: Helix with Threads.dwg

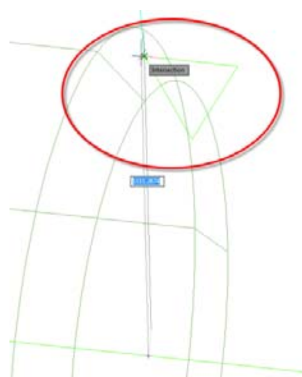


1. We are going to cut thread to an M10 x 1.5.

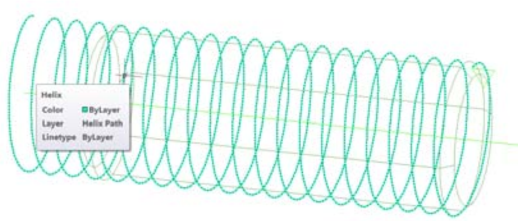
2. Set the Helix Path layer current. Use the **HELIX** command and snap to the Center point of the right end of the shaft.



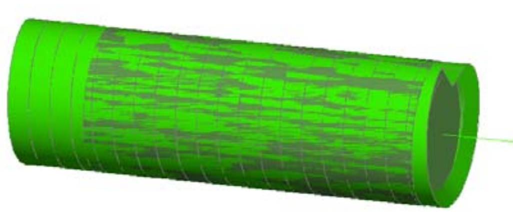
3. Then select the Intersection of the corner of the cutting tool as the base radius. Press enter to accept the same distance as the top radius.



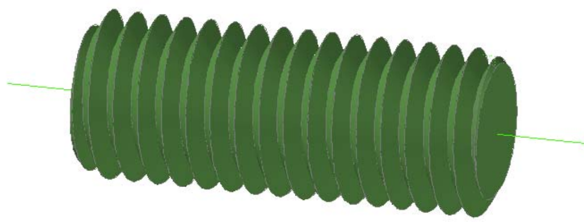
4. Set the turn **Height** and enter a height of **1.5** for the pitch of the thread.
5. Select **Axis endpoint** and select the end of the centerline at the opposite end of the shaft.



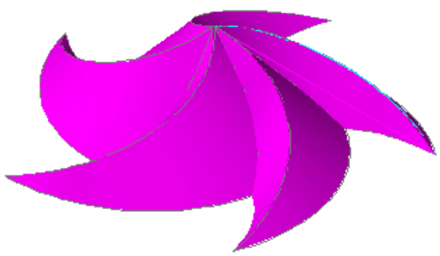
6. Set the Threading Tool layer current. Use the **SWEEP** command and select the triangle as the cutting tool. Set **Alignment to No**. Then select the helical path.



7. Use the SUBTRACT command to subtract the Sweep of the threads from the shaft.

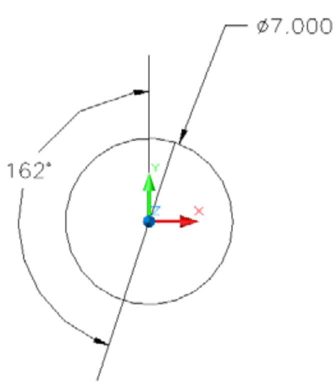


We are going to create this spiral Star using the **HELIX** and **SWEEP** commands.

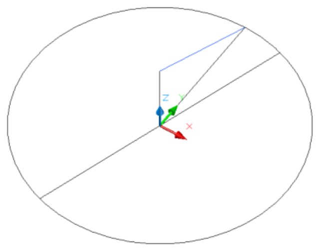


OPEN: Star.dwg.

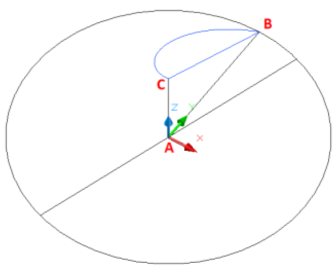
1. Type **DELOBJ** and set to 0.
2. Make the Visible layer current.
3. The circle with a diameter of 7 and the two lines has been drawn for you. (The angle comes from the formula 90° + 360°/5)



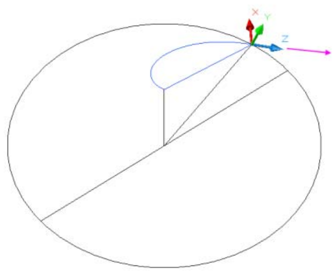
1. Type **UCSICON** and turn the Ucsicon off.
2. Set your view to the Home position.
3. Create a 2 unit vertical line and angled line as shown. (0,0,2) (Star1.dwg)



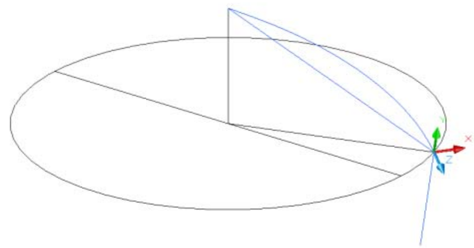
4. Make the Helix Layer current. Create a Helix with the base centered at point A, the radius at the intersection at point B. The top radius is 0. Turns 1/5 and the turn Height to the 20 unit axis height at point C. (Star2.dwg)



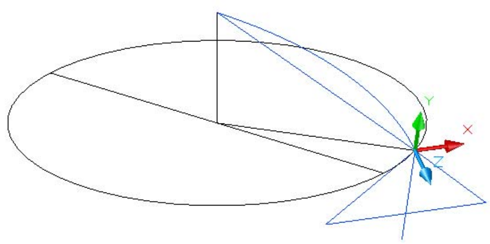
5. Set the **UCS** at the beginning of the helix as shown. **Use the ZAxis > Object** method. (Star3.dwg)



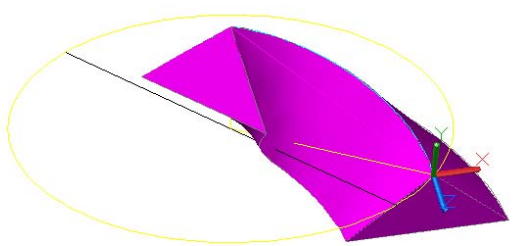
6. Set the layer Wires current. Draw a line 2 units starting at the end of the helix in the –Y direction. (0,-2,0)
7. Rotate your view as shown. (Star3.dwg)



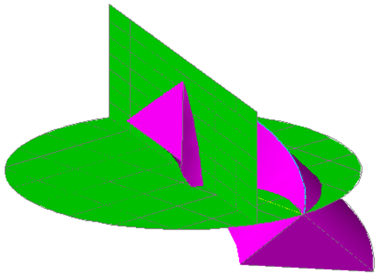
8. Use the **ROTATE** command and copy the 2 unit line to 45° and -45°. Use the LINE command and connect the endpoints. Use the **PEDIT** command and Join the three lines to create a polyline that forms the triangle as shown. (Star4.dwg)



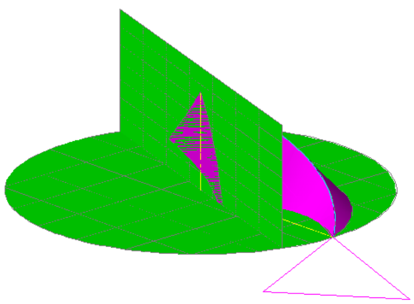
9. Set the current layer to Star. **SWEEP** the triangle along the helix as shown. Set the Visual Style to Realistic. (Star5.dwg)



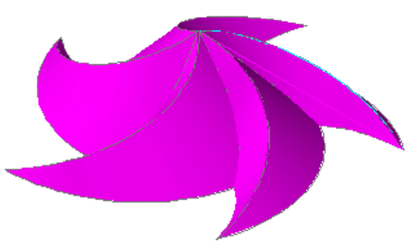
10. Type **UCS** and set to **World.** Set the current Layer to Surface. **EXTRUDE** the 162° line, 5 units high as shown. Type **PLANESURF**, Object option and select the circle. (Star6.dwg)



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. Patch Surfaces

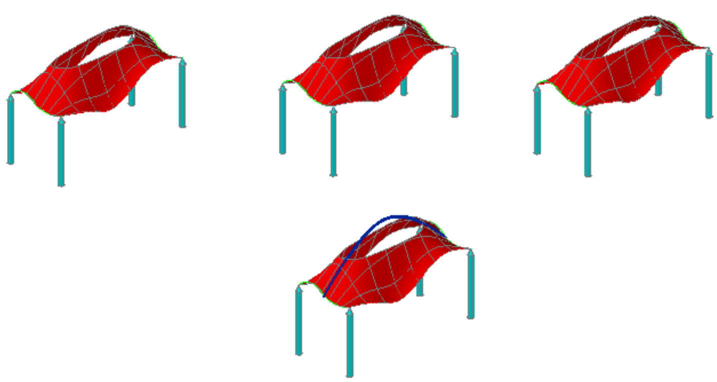
A **patch surface** is used to create a "patch" over an opening in an existing surface. A patch surface is used when it is necessary to close an opening or gap in the model. You can think of a patch surface as one of the many squares making up a quilt.

The **SURFPATCH** command is used to create a surface patch based on one or more edges forming a closed loop. You can select one or more surface edges or a series of curves. As when using the **SURFBLEND** command, you can specify the continuity and bulge magnitude to define the curvature of the surface.

Select the **SURFPATCH** command and then select one or more surface edges defining a closed loop. You can use the **Chain** option to select a chain of continuous surface edges. You can also use the **Curves** option to select multiple curves forming a closed loop. After selecting the edges or curves, press [Enter]. A preview appears and you can press [Enter] to create the surface using the default settings. The **Continuity** and **Bulge** magnitude options can be used to change the default settings as previously discussed. The default continuity setting is G0. The default bulge magnitude setting is 0.5.

The **Guides** option allows you to use a guide curve to constrain the shape of the surface patch. You can select one or more curves to define the guide curve. You can also select points to define the guide curve. When selecting points, use object snaps as needed.

Examples of creating patch surfaces are shown below. The top of the tent requires a patch. To create the patch, the single edge representing the opening in the model is selected with the default **Surface edges** option. When using the **Guides** option, draw a curve to serve as the guide curve prior to selecting the **SURFPATCH** command.

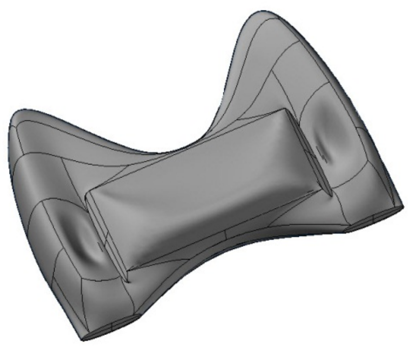


OPEN: Tent Patches.dwg

1. Use the **SURFPATCH** command to patch the top of the 3 tents in a row . In the top three examples, use different bulge magnitudes.

2. In the bottom tent, use the guide option and select the blue guide rail.
3. Do not save the drawing.

The game controller is to be redesigned with a new top shape. The model has been converted from a mesh model to a surface. In the original model (the mesh model), the top faces were deleted. The surface opening has eight continuous surface edges. The **Chain** option is used to assist in selecting edges to create the surface patch. After selecting the **SURFPATCH** command, select the **Chain** option and select one of the edges. The remaining edges are automatically selected. Next, press [Enter]. You can adjust the continuity and bulge magnitude settings or press [Enter] to create the patch surface.



OPEN: Game Controller.dwg

1. Use the SURFPATCH command to patch the top of the game controller. Use the **CHain** option to select the eight continuous surface edges. Set different continuities and bulge magnitudes.
2. Do not save the drawing.

8. Trimming Surfaces

The **SURFTRIM** command can be used to trim surfaces or regions using ***other existing surfaces***. You can trim any part of a surface where the surface intersects with another surface, region, or curve. In addition, you can project an existing object onto a surface to serve as a trimming boundary. The object to be trimmed and the cutting object do not have to intersect. When an associative surface is trimmed, it remains associative and retains the ability to be modified by editing the cutting object.

Select the **SURFTRIM** command and then select one or more surfaces or regions to trim. After selecting the objects to trim, press [Enter]. Next, you are prompted to select the cutting objects. Select one or more curves, surfaces, or regions. The **Extend** and **Projection direction** options are available after selecting the **SURFTRIM** command. The **Extend** option determines whether a surface used as a cutting edge is extended to meet the surface to be trimmed. By default, this option is set to **Yes**. The **Projection direction** option specifies the projection method used for projected geometry.

9. Using Projected Geometry to Trim Surfaces

You can select lines, arcs, circles, ellipses, polylines, splines, and helices. The **Projection direction** option of the **SURFTRIM** command can be used to set the projection method used by AutoCAD when projecting curves onto a surface. The following settings are available:

- **Automatic.** The cutting object is projected onto the surface to be trimmed. The projection is based on the current viewing direction. In a plan view, the projection of the cutting object is in the viewing direction. In a 3D view, the projection of a planar curve is normal to the curve, and the projection of a 3D curve is parallel to the direction of the Z axis of the current UCS. The **Automatic** option is set by default.

- **View.** The cutting object is projected in a direction based on the current view.

- **UCS.** The cutting object is projected in the positive or negative direction of the Z axis of the current UCS.

- **None.** The cutting object is not projected and must lie on the surface in order to perform the trim.

10. Projecting geometry – Surfaces and the THICKEN command

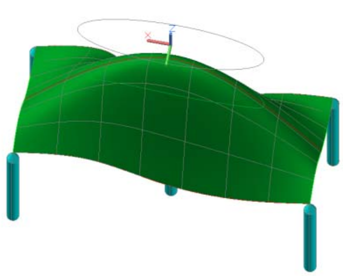
The **PROJECTGEOMETRY** command is used to projects points, lines, or curves onto 3D solid or surface from different directions. You must set the system variable **SURFACEAUTOTRIM 1**.

A surface has no thickness. But, a surface can be quickly converted to a 3D solid using the **THICKEN** command.

To add thickness to a surface, enter the **THICKEN** command. Then, pick the surface(s) to thicken and press [Enter]. Next, you are prompted for the thickness. Enter a thickness value or pick two points on screen to specify the thickness

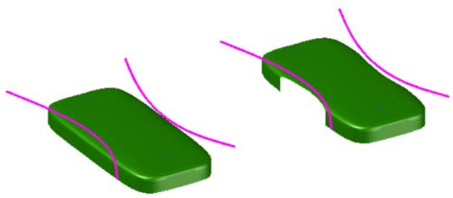
OPEN: Projectgeometry.dwg

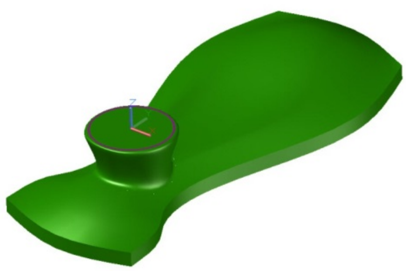
1. Project the ellipse onto the top of the tent.
2. Trim a hole in the tent.
3. Thicken the surface as needed. What happens to the surface?



OPEN: Cell Phone Case.dwg

1. Use the SURFTRIM command to trim the sides of the case. (Be careful as you your viewing direction).
2. Use SURFUNTRIM to untrim the surface.
3. Thicken the surface as needed. What happens to the surface?
4. Do not save the drawing.





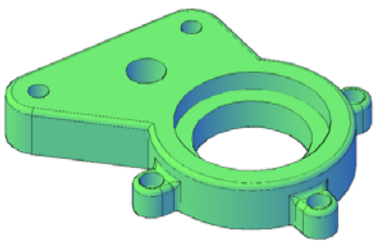
OPEN: 3D CAD Mouse.dwg

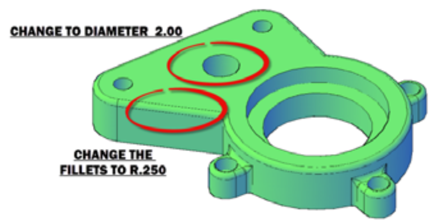
1. Use Lofted surfaces to create the sides of the base.
2. Use a patch surface for top of the mouse with different continuity settings and bulge magnitudes.
3. Surface trim the hole in the base. Use the circle as the trim tool.
4. Add a Planar surface based on the circle at the top.
5. Use Surface blends to blend the Planar Surface to the base surface.
6. Thicken the surface as needed. What happens to the surface?
7. Save the drawing as 3D Mouse.dwg

10. Editing the Solid

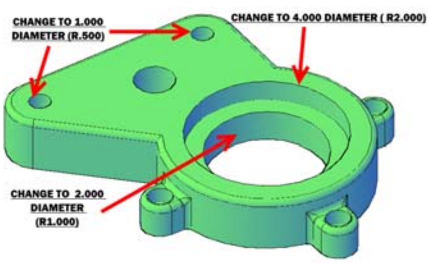
OPEN: BaseBracket-Fusion Detail.dwg

1. 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.

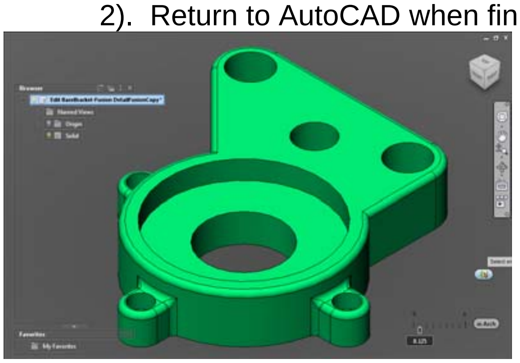




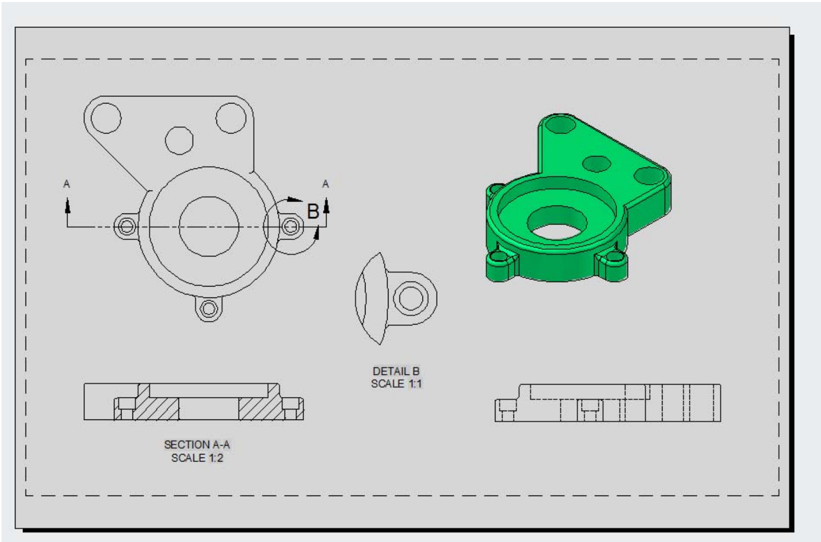
2. Edit the other cylinders as shown. What happens?



3. Autodesk Inventor Fusion to the rescue! Launch Autodesk Inventor Fusion and select each circle one at a time. Use the **PRESSPUL**L tool to change the diameters (radius x 2). Return to AutoCAD when finished editing.



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.



NOTE: Inventor Fusion is NOT Autodesk Fusion 360. Inventor Fusion was a download for install feature of AutoCAD 2013. If you have AutoCAD 2014 models and you need to edit them, install Inventor Fusion 2013 in your AutoCAD 2013 software and edit there. Do check on-line if Inventor Fusion for AutoCAD 2014 is a download program and you can install in AutoCAD 2014. I have not tried this. I edit some of my models in AutoCAD 2013 using Inventor Fusion 2013.

For his assistance with the screwdriver, starring and cut threads topics in this paper

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

Pennsylvania College of Technology, Williamsport, Pennsylvania.