Compelling Features of 3D Surface Modeling in AutoCAD® 2013

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AC2292-L This intermediate-to-advanced hands-on lab offers AutoCAD software 3D veterans and experienced 2D users a chance to explore surface modeling techniques in AutoCAD 2013 to create compelling 3D surface models. Tools for 3D 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 surface models. We will explore creating procedural, networked, blended, patched, offset, filleted, and trimmed surfaces. We will convert a surface to NURBS and edit surfaces by thickening and sculpting to create solids. If you have used AutoCAD 3D in the past, attend this lab and get ready to be surprised!

Learning Objectives
At the end of this class, you will be able to:

- Create procedural surfaces
- Create network surfaces
- Create blend and patch surfaces
- Convert and edit surface models

About the Speaker
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Understanding Surface Model Types

There are two basic types of surface models in AutoCAD: **procedural surfaces** and **NURBS surfaces**. A **procedural surface** is a standard surface object without control vertices. By default, a procedural surface, when created, is an **associative surface**. This means that the surface maintains associativity to the defining geometry or to other surrounding surfaces. Editing the defining geometry of an associative surface, or an adjacent surface in a “chain” of associative surfaces, modifies the surface.

A **NURBS surface** is based on splines or curves. The acronym **NURBS** stands for non-uniform rational B-spline. NURBS surfaces are based on a mathematical model and are used to create organic, freeform shapes. NURBS surfaces have control vertices that can be manipulated to edit the shape of the surface with great precision. Unlike a procedural surface, a NURBS surface cannot be created as an associative surface.

A third type of surface in AutoCAD is a **generic surface**. A generic surface has no associative history and no control vertices.

The type of surface model created is controlled by the **SURFACEMODELINGMODE** system variable. The default setting, 0, creates procedural surfaces. If the **SURFACEMODELINGMODE** system variable is set to 1, NURBS surfaces are created.

When creating a procedural surface, the **SURFACEASSOCIATIVITY** system variable setting determines whether an associative surface is created. The default setting, 1, creates associative surfaces. This system variable has no effect when creating NURBS surfaces.

Surface models can be created from either closed and bounded geometry or open profile geometry. When using modeling commands such as **EXTRUDE**, **REVOLVE**, **Sweep**, and **LOFT**, the **Mode** option determines whether a surface model or solid model is created. Open profile curves always create surfaces, regardless of the **Mode** option setting.

The advantage to a procedural surface is the ease with which the surface can be created based on common shapes. In addition, working with procedural surfaces allows the designer to take advantage of associative modeling. Based on the design intent, the designer can use profile curves such as lines, circles, arcs, ellipses, helices, points, polylines, 3D polylines, and splines as the basis for the model. A procedural surface model can then be created using commands such as **EXTRUDE**, **REVOLVE**, **Sweep**, **LOFT**, or **PLANESURF**. If created as an associative surface, the model is linked to the defining geometry and can be modified by editing the geometry.

The commands used to create surface models are located in the **Surface** tab of the ribbon.
PROFESSIONAL TIP:

Procedural surfaces are also referred to as explicit surfaces. The terms procedural surface and explicit surface are interchangeable. NURBS surfaces are used in creating models that represent sophisticated freeform shapes. Procedural surfaces can be converted to NURBS surfaces in order to model more sophisticated surface model shapes. In the exercise you will create procedural surface models.

OPEN: Exercise 1.dwg

In this exercise you will create procedural surfaces using EXTRUDE, SWEEP, LOFT REVOLVE, PLANAR Surface and NETWORK surface. Understanding procedural surfaces creation gives the designer workflow concepts of how surfaces are created and can be used for creating more complex free-form surfaces models.

Complete Exercise 1 as a warm up exercise. The Surface commands are similar to the solids command. Read the command options carefully.

1. From the Surface tab>Create panel, select EXTRUDE. Extrude the circle to create surface cylinder 10 units high using the MOde: Surface.
2. From the Surface tab>Create panel, select SWEEP. Sweep the polygon alone the polyline sweep path using the MOde: SUrface.
3. From the Surface tab>Create panel, select LOFT. Loft the three cross section profiles using the MOde: SUrface.
4. From the Surface tab>Create panel, select REVOLVE. Revolve the polygon 360 degrees alone the red centerline axis using the MOde: SUrface
5. From the Surface tab>Create panel, select Planar Surface. Select polygon object to create a planar surface.
6. From the Surface tab>Create panel, select Network surface. Select curves A,B,C in order and press ENTER. Select curves C and D in order and press ENTER.
7. SAVE the drawing as Surface1.dwg.

Working with Associative Surfaces

Procedural surfaces, when created, are associative by default. An associative surface changes shape or adjusts to the modifications made to the defining profile geometry (or other adjoining surfaces). This provides flexibility in the design. However, it is important to remember that when modifying the shape of an associative surface, you modify the profile geometry, not the surface. Modifying the profile geometry maintains the associative relationship. If you pick on the surface and then attempt to modify it, AutoCAD issues a warning that the surface will lose its associativity with the defining curve, surface, or parametric equation. If you choose to continue with the operation, the associativity is lost. You can cancel the operation to preserve the associativity.
Set the **SURFACEASSOCIATIVITY** system variable to 1 to create associative surfaces. If the system variable is set to 0, surfaces that are created have no associativity to defining profile curves or other surfaces.

**PROFESSIONAL TIP:**

The **DELOBJ** system variable is ignored if the **SURFACEASSOCIATIVITY** system variable is set to 1. Also, when creating a NURBS surface, the surface is a NURBS surface and is not associative.

**OPEN: SURFACE1.dwg (or use the Exercise 1 drawing that is currently open).**

In this exercise you will edit the profiles that created the associated surfaces in Exercise 1.

1. Select the profiles that created the surfaces in Exercise 1. Use the grips by stretching the profile to create unique surface shapes.
2. **Save the drawing as Surface2.dwg.**

**Removing Surface Associativity**

The surface associativity can be removed from a surface once the surface has been created. This can be done by selecting the surface and opening the Properties palette. The Maintain Associativity property in the Surface Associativity category controls the associativity of the surface. The default property is set to Yes. Selecting Remove from the drop-down list removes the associativity and changes the property to None. This converts the surface to a generic surface.

The Show Associativity property in the Surface Associativity category controls whether adjoining associative surfaces are highlighted when a surface is selected in order to indicate dependency. When this property is set to Yes and a surface is selected, AutoCAD highlights other surfaces to which the surface is dependent. This can be useful for identifying associative relationships in a chain of surfaces.

**PROFESSIONAL TIP:**

When moving, scaling, or rotating an associative surface, be sure to select the underlying curve geometry defining the surface. Failure to select the underlying geometry will result in the loss of the associativity.

**Determining Modeling Workflows for Procedural Surfaces and NURBS Surfaces**

When the design of a 3D model requires a freeform shape that would be difficult to create using solids, start by creating a procedural surface. You can convert the surface as required. A practical application is creating a surface model of a car fender. Start with a lofted surface based on four guide curves. Finish creating the fender by creating several procedural surfaces or patches. Then, convert the fender surfaces to NURBS surfaces as needed and add further editing techniques for a more freeform sculpted shape.
Different factors determine when to use procedural surface modeling and NURBS surface modeling. For example, create procedural surfaces when it is important to maintain associativity and you plan to edit the original geometry. On the other hand, NURBS surfaces have control vertices that typically permit greater flexibility when editing. NURBS surfaces are often very useful for modeling organic shapes. The extent to which the design will require further editing can serve as a guideline for determining the best modeling approach.

**Creating Network Surfaces**

A network surface is a surface model created by a group or “network” of profile curves or edges. A network surface is similar to a loft surface. As with a loft, the defining profiles can be open or closed curves, such as splines. The defining profiles can also be the edges of existing objects, including region edges, surface edge subobjects, and solid edge subobjects. The curves or edges selected can intersect at coincident points, but do not have to intersect.

The **SURFNETWORK** command is used to create network surfaces. After selecting this command, select the curves or edges defining the first direction of the surface. Make sure to select the curves in the order of surface creation. Then, press [Enter]. Next, select the curves or edges defining the second direction of the surface. Press [Enter] when you are done selecting the profiles. This creates the surface and ends the command.

The curves selected for the two directions define the U and V directions of the surface. The U and V directions can be thought of as the local directions of the surface and can be defined in either order. The U and V directions define the “flow” of the surface.

You can create a network surface from region edges, surface subobject edges, and solid subobject edges. To select the profile edges, press and hold the [Ctrl] key. You can also use the edge subobject filter by selecting **Edge** from the **Selection** panel in the **Home** tab of the ribbon.

**PROFESSIONAL TIP:**

Surface associativity plays an important role in the creation of network surfaces. When editing a network surface with associativity, select a curve that forms the basis for the surface and modify it as needed. The result will be a new network surface shape.

**OPEN: Exercise 3.dwg**

In this exercise, you will create a procedural Network surface with surface associatively on.

1. Using the **SURFNETWORK** command, create a network surface of the car bumper. Select curves A, B and C as the first direction surfaces and curves D and E as the second direction surfaces.
3. Select different curves that form the network surface and stretch the curves to form a new car bumper shape. These new shapes are possible because surface associativity is on.

![Network Surfaces Diagram]

3. Save the drawing as Networked.dwg.

Creating Surfaces from Existing Surfaces

In addition to creating surfaces from profile curves, you can create surfaces from existing surfaces using the \texttt{SURFBLEND}, \texttt{SURFPATCH}, \texttt{SURFOFFSET}, \texttt{SURFFILLET}, and \texttt{SURFEXTEND} commands. These commands are discussed in the following sections. Surface models created with these commands are created as associative surfaces by default. This maintains associativity between the surfaces used to create the surface and the resulting surface.

Blend Surfaces

When working with surface models, there are situations when you need to “blend” together \textit{surfaces} that do not meet or touch. The \texttt{SURFBLEND} command is used to create a \textit{blend surface} between two surface edges or two solid edges. When blending surfaces, you select the surface edges to blend, \textit{not} the surfaces.
SURFBLEND

Select the SURFBLEND command and then select the first edge to blend. Press and hold the [Ctrl] key to select the first edge or use the edge subobject filter. You can select multiple edges or use the Chain option to select a chain of continuous edges. Press [Enter] after defining the first edge. Next, select the second edge. Select a single edge or multiple edges, or use the Chain option to select a chain of continuous edges. When you press [Enter], a preview of the surface appears. You can press [Enter] to accept the default settings, as shown above, or you can use the Continuity and Bulge magnitude options to specify the continuity and bulge magnitude settings at the edges. Different settings can be applied at each edge. Continuity defines how the surfaces blend together at the starting and ending edges. The following options are available:

• **G0 (positional continuity)**. This option creates a sharp transition between surfaces. The position of the surfaces is maintained continuously at the surface edges. This option is used for creating flat surfaces.

• **G1 (tangential continuity)**. This option forms surfaces so that the end tangents match at the edges. The two surfaces blend together tangentially.

• **G2 (curvature)**. This option creates a curvature blend between the surfaces. The surfaces share the same curvature.

**Bulge magnitude** defines the size or “bulge” of the radial transition where the surfaces meet. See below. Valid values range from 0 to 1. A greater value is valid, but results in a larger roundness to the blend. Using different surface modeling techniques instead of entering a value greater than 1 is recommended. If the surface is set to G0 (positional continuity), changing the default bulge magnitude value has no effect.

Using different continuity and bulge magnitude settings modifies the surface and provides a way to create different blend surface shapes. You can change the continuity by using the grips that appear when creating the blend surface. Picking on a grip displays a menu with the continuity options. The same grips appear when selecting a blend surface after it has been created. You can also use the

**Properties** palette to edit the settings of a blend surface.
OPEN: Blend1.dwg

1. Use the BLENDS command to create the blended tubes as shown above.
2. Do not save the drawing.

OPEN: Blend2.dwg

1. Use the BLENDS command with different bulge magnitudes to create the caps at the end of the surface arcs.
2. Do not save the drawing.

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

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.

![Tent Patches](image)

**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.

PROFESSIONAL TIP:

Surfaces created using the SURFPATCH command may differ from those created with the LOFT command. In addition, you may come across design situations where curves used with the LOFT command will not create a surface, but will when used with the SURFPATCH command.

PROFESSIONAL TIP:

The PREVIEWCREATIONTRANSPARENCY system variable controls the transparency of surface previews when using the SURFBLEND, SURFPATCH, and SURFFILLET commands. The default setting is 60. Setting a higher value increases the transparency of the surface preview.

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. Then, press [Enter]. You are then prompted to select the surface areas to be trimmed. Select one or more areas. As you select each area, it is trimmed by the cutting object(s).
If you trim an area that you wish to restore, use the Undo option. 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.

Using Projected Geometry to Trim Surfaces

With the SURFTRIM command, you can trim surfaces using cutting objects other than existing surfaces. Objects used in this manner are referred to by AutoCAD as curves. Selecting a curve, such as an arc or circle, allows you to project the geometry onto the surface and use it as the cutting edge. In the example shown, the cell phone case is selected as the surface to trim. The arcs located above the cell phone case are selected as the cutting edges. When selected, the arcs are projected onto the surface by AutoCAD. The areas to be trimmed are then selected to complete the sides.

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.

Projecting geometry

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.

PROFESSIONAL TIP:

The SURFTRIM command defaults to the Automatic option. Automatically projected geometry is used in most trim situations.

OPEN: Projectgeometry.dwg

1. Project the ellipse onto the top of the tent.
2. Trim a hole in the tent.
3. Undo the trim.
4. Extrude the projected geometry upward to create a new 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. Do not save the drawing.

SURFUNTRIM

If you need to restore a trimmed surface back to its original shape, use the SURFUNTRIM command. After selecting this command, select the edge of the surface area to untrim. If the surface has multiple trimmed edges, you can use the Surface option. The SURFUNTRIM command untrims surfaces trimmed by the SURFTRIM command. It does not untrim surfaces trimmed using the PROJECTGEOMETRY command.

Challenge Exercise

OPEN: CarBody Project Geometry.dwg

1. Use the PROJECTGEOMETRY command to project the top blue ellipse to the top of the car.
2. Rebuild the projected ellipse.
3. Trim or Extrude the opening.
4. Patch the opening as needed.
5. So not save the drawing.

Challenge exercise

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. Save the drawing as 3D Mouse.dwg

SURFEXTRACTCURVE

Creating the underlying wireframe geometry required in surface or solid modeling can sometimes be difficult. When working with curved surfaces, it may help to create a profile based on the contours of an existing model. The SURFEXTRACTCURVE command allows you to extract isoline curves from an existing
surface in the local U and V directions of the surface. Surface curves can be extracted from a surface model, solid model, or face of a solid. This provides a way to experiment with wireframe geometry and get a better idea of what the resulting model will look like. The `SURFEXTRACTCURVE` command creates objects such as lines, circles, arcs, polylines, and splines, depending on the existing model. Once the new curves are created, the designer can use the geometry for model edits or conceptual design purposes.

To extract isoline curves from an existing model, enter the `SURFEXTRACTCURVE` command and select a surface. An isoline curve matching the surface contour appears attached to the crosshairs. You can move the crosshairs to other surfaces to preview different curves. The default direction for the curve extraction is in the U direction. You can change the direction to the V direction by selecting the `Direction` option. Move the crosshairs until you locate the desired curve and pick to create the new curve.

The `Chain` option is used to extract curves from a “chain” of adjacent surfaces.

The `Spline points` option allows you to extract a spline by selecting points on a model surface. The spline passes through all specified points. If needed, use the `Close` option to create a closed spline. The `SURFTRIM` command is used to form the opening by trimming the center area. The spline is selected as the cutting curve in the guitar below.

**OPEN: Guitar_Curves.dwg**

1. Use the `SURFEXTRACTCURVE` command to draw two horizontal and three longitude lines on the face of the guitar.
2. Use the `SURFEXTRACTCURVE` command to draw spline points to create the elliptical shape as shown.

3. Use the `SURFTRIM` command to trim out the center portion of the guitar. Do not save the drawing.
SURFOFFSET or Thicken

The SURFOFFSET command allows you to offset a surface to create a new, parallel surface at a specified distance. You can offset a surface in one direction or in both directions from an existing surface. You can also offset a region to create a new surface. You can also create a new solid by offsetting surfaces.

The THICKEN command create a solid from a surface. First, create a complex surface, then use the Thicken to create a complex solid.

OPEN: SurfaceOffset.dwg

1. Use the SURFOFFSET command to create two offset surfaces. Set the offset distance to .25.
2. Use THICKEN command on the second surface to create a solid object.
3. Do not save the drawing.

Creating Fillet Surfaces
The **SURFFILLET** command is used to create a fillet between two existing surfaces. The fillet created is a rounded surface that is tangent to the existing surfaces. You can create fillet surfaces from existing surfaces or regions. Using the **SURFFILLET** command is similar to using other fillet commands in AutoCAD.

**OPEN: FilletSurface.dwg**

1. Using the **SURFFILLET** command, create a fillet between the two tubes. Use a radius of .250 with Trim, **no**.

2. Fillet the two planar surfaces that do not meet.

3. Do not save the drawing.

![Image of fillet surfaces](image)

To create a fillet surface, select the **SURFFILLET** command. First, set a radius using the **Radius** option. Then, select two surfaces. By default, the existing surfaces are trimmed to form the new surface. The surface trimming mode can be set by selecting the **Trim surface** option. AutoCAD stores the radius you specify as the setting for the **FILLETRAD3D** system variable. If you do not specify a radius, the current **FILLETRAD3D** system variable setting is used.

**PROFESSIONAL TIP:**

You can use the **UNION** command to union surfaces. However, it is not recommended. You will lose the surface associativity between the surfaces and the defining profile curves. Use surface editing commands instead.

**Extending Surfaces**

You can add length to an existing surface using the **SURFEXTEND** command. When extending a surface, you can specify whether the new surface is created as a continuation of the existing surface or as a new surface. The surface extends to a new length using the specified distance.
Select the **SURFEXTEND** command and select one or more surface edges to extend. After selecting an edge, press [Enter]. A preview of the extended surface appears and you can drag the cursor dynamically to set the distance. You can also enter a distance by typing a value.

Before specifying the extension distance, you can use the **Modes** option to specify the extension mode. The two options are **Extend** and **Stretch**. The default **Extend** option is used to extend the surface in the same direction as the existing surface and attempt to maintain the surface shape based on the surface contour. The **Stretch** option is also used to extend the surface in the same direction as the existing surface. However, the resulting extension may not have the same surface contour.

After specifying the extension mode, the **Creation type** option can be used to set the type of surface created. The two options are **Merge** and **Append**. The default **Merge** option is used to extend the surface as one surface. The **Append** option is used to create a new surface extending from the original surface. This option results in two surfaces instead of one merged surface. After creating the new surface, you can use the **Properties** palette to edit the extension distance.

**OPEN: ExtendSurface.dwg.**

1. Use the **SURFEXTEND** command to extend the surfaces.
2. Do not save the drawing.

**NURBS Surfaces**

When creating a NURBS surface, you use many of the same commands that you would use to create a procedural surface. You can use splines and various curve shapes to create NURBS surfaces. In addition, you can convert procedural surfaces into NURBS surfaces.

A NURBS surface is created when the **SURFACEMODELINGMODE** system variable is set to 1. In addition, NURBS surfaces are non-associative. The setting of the **SURFACEASSOCIATIVITY** system variable has no effect when creating a NURBS surface.

The advantage of working with NURBS surfaces is that you use control vertices to control or influence the shape of the surface. The ability to edit control vertices provides significant flexibility in creating and sculpting freeform, organic shapes.
NURBS Surface Modeling Workflows

There are two common workflows used in NURBS surface modeling. You can begin by creating procedural surfaces and then convert them to NURBS surfaces, or you can create the initial surfaces as NURBS surfaces.

When you start the modeling process by working from procedural surfaces, the following workflow is common:

- Create procedural surfaces using commands such as **EXTRUDE**, **REVOLVE**, **SWEEP**, **LOFT**, **PLANESURF**, and **SURFNETWORK**. The **SURFACEMODELINGMODE** system variable should be set to 0.
- Create other surfaces, such as blend surfaces, patches, fillets, and offset surfaces. Use the commands presented in this chapter.
- Convert the surfaces into NURBS surfaces.
- Edit the NURBS surfaces as needed to create the desired sculpted shape.

When you start the modeling process by creating NURBS surfaces, the following workflow is common:

- Set the **SURFACEMODELINGMODE** system variable to 1 (on). With this setting, NURBS surfaces are created.
- Create the surfaces needed to create the desired model shape. When using this approach, you use splines or curves to define the surface profile. Splines are created using the **SPLINE** command. Splines used for NURBS surface models are typically created with the Method option of the **SPLINE** command set to CV. This creates splines with control vertices (CVs), also known as CV splines. Control vertices play a major role in editing NURBS surfaces.
- Edit the NURBS surfaces as needed to create the desired shape.

There are several important points to keep in mind when you are working with NURBS surfaces. Once a procedural surface is converted to a NURBS surface, the NURBS surface cannot be converted back to a procedural surface. In addition, once a NURBS surface is created, it cannot be converted to a procedural surface. The design workflow you use is important. Make sure to plan ahead so that your modeling process is suitable for the design.
Creating and Editing NURBS Surfaces

3DEDITBAR

For greater control when editing the control vertices of a NURBS surface, you can use the 3DEDITBAR command. Select this command and select the NURBS surface to edit. You are then prompted to select a point on the surface. When you select a point, the 3D edit bar gizmo appears. This gizmo is similar to the move gizmo that appears when working with 3D objects. However, it contains additional grips for setting the tool options and modifying the tangencies of the surface. The grips include a square grip, triangle grip, and tangent arrow grip. The square grip represents the initial base point of the edit. Picking on the grip and dragging reshapes the surface from the base point. The triangle grip is used to specify the method for reshaping the surface. Picking on the grip displays a shortcut menu with the Move Point and Tangent Direction options. The Move Point option is used to reshape the surface by moving the base point. The Tangent Direction option is used to adjust the magnitude or bulge of the tangency at the base point. The tangent arrow grip is used to dynamically modify the tangency.

In this exercise, a NURBS surface model of a game controller will use the 3DEDITBAR command and 3D edit bar gizmo to modify two surfaces.

OPEN: Game Controller Nurbs.dwg

In the example, the game controller was converted to a NURBS surface for further editing of different surfaces.

1. Use the CONVTONURBS command to convert the game controller to a NURBS surface.
2. Type CVSHOW and select one of the surfaces of the game controller. Notice the Control Vertices. Type CVHIDE and the control vertices will hide.
3. Using the 3DEDITBAR command, select the left side of the game controller to edit. Select the middle of the surface.
4. Select the square grip to move the base point downward in the -"Z" direction .375 units. This creates a small indentation in the side of the game controller.
5. Repeat the procedure on the right side.
6. Continue editing the surfaces by pulling or dragging using the 3D edit bar gizmo until you have created a new design shape for a game controller.
7. Do not save the drawing.

Show and edit CV's (Control Vertices)

In this exercise, we will fine tune a NURBs surface by editing the CV's to create a more refined sculptured surface. Use the CVREBUILD command to rebuild the surface and make modifications to the surface.

OPEN: NURBS-Edit.dwg

1. Use the CONVTONURBS command to convert the surface to a NURBS surface.
2. Type CVSHOW and select the surface. Notice the Control Vertices. Type CVHIDE and the control vertices will hide.
3. Select a grip or hold the [CTRL] key to select multiple grips to edit the CV's.
4. Use the CVREBUILD command and select the surface.
5. Make the following changes. Change the amount of CV's. (U and V direction) to 14 and 8 as shown below.
The surface can be previewed to see if the changes are acceptable in terms of maximum deviation. (Standard deviation: simple put, is comparing the points in the original and the edited surface, and show the maximum difference between the two. This is what we can call Maximum Deviation. Maximum Deviation as shown in the dialog box is the longest distance between two common points in both surfaces, expressed in whichever unit you are using).

6. Add some rows of CV's to the surface. (U or V direction) Select the CVADD command to add some rows. You can change the Direction of the CV rows (on the top of the surface) or add Knots (Turns off the display of control vertices and allows you to place a point directly on a surface).

7. Do not save the drawing.
SURFSCULPT - Surface to Solid

When designing using surfaces, you can sculpt a surface into a solid using the SURFSCULPT command. This is similar to using the CONVTOSOLID command. The main use for sculpting is to create a solid from a watertight area by trimming and combining multiple surfaces. The command can also be used on solid and mesh objects.

Which command should you use, CONVTOSOLID or SURFSCULPT? The difference is very subtle.

- CONVTOSOLID works the best when you have a watertight mesh and want to convert it to a solid. Also, it works well on polylines and circles with thickness.
- SURFSCULPT works the best when you have watertight surfaces or solids that completely enclose a space (no gaps).

OPEN: SurfaceSCULPT.dwg or Game Controller Surface to Solid.dwg

1. Use the SURFCULPT command to create a solid.
2. Do not save the drawing.

1. Open: Guitar.dwg. (To sculpt a surface to a solid) or OPEN: GUITAR_Patch_Sculpt.dwg (To Patch a side of the guitar and then sculpt the guitar).
2. Set the DELOBJ system variable to zero (0).
3. Enter the **SURFSCUPLT** command and window the Guitar as the selection set.

4. Move the solid Guitar to the right of the watertight surface as shown. Do not save the drawing.

**Surface Slicing - Surfaces as cutting tools to solids**

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.
Challenge Exercise

OPEN: Car Body Design.dwg

1. Use Network, Loft, Patch and Blend surfaces to create the car body. Create a trim profile for the top hole as was done in a previous exercise. Create a profile or cylinders to trim out the wheel wells.

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