Learning Objectives

After completing this chapter, you will be able to:
✓ Understand and work with different types of surface models.
✓ Create procedural surfaces.
✓ Create NURBS surfaces.
✓ Create network surfaces.
✓ Create surface models from existing surfaces.
✓ Blend and patch surfaces.
✓ Offset, fillet, extend, and trim surfaces.
✓ Convert existing models to NURBS surfaces.
✓ Edit NURBS surface control vertices.
✓ Convert 2D objects to surfaces.
✓ Thicken a surface into a solid.
✓ Sculpt watertight surfaces into solids.

Overview

This chapter describes advanced surface modeling techniques and workflows used in AutoCAD. Surface modeling provides the ability to create a more freeform shape with tools that solid modeling cannot provide. You have been introduced to basic surface modeling techniques in previous chapters. As you have learned, one way to create surface models is to extrude, revolve, sweep, or loft profiles. This chapter builds on those techniques. In this chapter, you will develop an understanding of surface modeling techniques that can stand alone in the design process or work in combination with other modeling techniques.

As discussed in previous chapters, a solid model is created with a closed and bounded profile and has mass and volume properties. A mesh consists of vertices, edges, and faces that define the 3D mesh shape. A mesh does not have mass or volume. A surface model can be thought of as a thin-walled object with no “Z” depth. A surface model does not have mass or volume.
Advanced Surface Modeling

Chapter 11

AutoCAD and Its Applications—Advanced

276

As discussed in Chapter 8, selecting a command from the ribbon. See geometry maintains the associative relationship. If you pick on the surface and then tab of the surface. Modifying the profile not 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

there are a number of workflows in AutoCAD available to the 3D designer. The following approaches can be considered depending on the nature of the work or the requirements of a specific application:

- Creating 3D models as solids, meshes, procedural surfaces, or NURBS surfaces (procedural surfaces and NURBS surfaces are discussed in the next section).
- Using Boolean operations on solids to create composite solids.
- Slicing composite solids using surfaces.
- Converting solids to mesh models.
- Converting solids to surface models.
- Converting surface models to NURBS surfaces.

These are just a few of the possible workflows. Editing techniques are also available and often play a significant role in surface modeling.

Understand 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 (as discussed in previous chapters). 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. See Figure 11-1. As discussed in Chapter 8, selecting a command from

Figure 11-1. The Surface tab of the ribbon.

Picking this button sets the value of the SURFACEASSOCIATIVITY system variable.

Picking this button sets the value of the SURFACEMODELINGMODE system variable.

Planar surface creation is discussed in Chapter 2. Extruded and revolved surfaces are discussed in Chapter 9. Swept and lofted surfaces are discussed in Chapter 10.

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.

An example of creating a procedural surface model is shown in Figure 11-2. In this example, an associative surface is created from a series of cross-sectional profiles using the LOFT command. As discussed in Chapter 9, a loft is created by selecting the cross-sectional profiles in order. In Figure 11-2A, four open profile curves are selected to create the loft. By default, this creates a surface model. In Figure 11-2B, the object is shown after lofting. In Figure 11-2C, the object is shown after moving and scaling the third open profile. Because the surface model is associative, the model updates and adjoining surfaces adjust to conform to the new curve shape.

In Figure 11-3, a loft surface is created from three closed cross-sectional profiles (circles). The surface is created by using the Mode option or by selecting the command from the Surface tab in the ribbon. In Figure 11-3B, the object is shown after moving and scaling the third closed profile at the top. Because the surface is associative, the model updates and conforms to the new curve shape.

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. See Figure 11-4. By default, the 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.

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, as discussed later in this chapter. 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.
The following sections discuss surface modeling commands and techniques available in AutoCAD. Procedural surfaces are shown in examples as the default creation method. NURBS surface creation and editing techniques are covered later in this chapter.

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. See Figure 11-5. A network surface is created as an associative surface by default.

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.

Figure 11-6 shows examples of creating network surfaces from similar sets of profile curves. In Figure 11-6A, a series of connected profile curves defines the network surface. In Figure 11-6B, two of the curves do not intersect with other profiles. Notice the differences between the resulting surface models.

Creating a network surface from region edges, surface subobject edges, and solid subobject edges is shown in Figure 11-7. Creating a network surface in this manner may result in some unexpected surface shapes. 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. In Figure 11-7, the four objects are located at different “Z” heights. The network surface is created from the four non-intersecting edges.

Figure 11-5.
Creating a network surface. Splines are used as the profiles in this example. A—The profiles used to define the surface are selected in the numbered order shown. Profiles 1–3 define the first direction and are shown in color. Profiles 4–8 define the second direction. B—The resulting surface model.
Creating Surfaces from Existing Surfaces

In addition to creating surfaces from profile curves, you can create surfaces from existing surfaces using the SURFBLND, SURFPATCH, SURFFILET, and 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 surfaces that do not meet or touch. The SURFBLND command is used to create a blend surface between two surface edges or two solid edges. When blending surfaces, you select the surface edges to blend, not the surfaces.

Select the SURFBLND command and then select the first edge to blend. Press and hold the [G] 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 in Figure 11-9, 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. The settings are similar to those when creating a loft, as discussed in Chapter 9.

**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. This is the default option, as shown in Figure 11-9B.
- **G2 (curvature)**. This option creates a curvature blend between the surfaces. The surfaces share the same curvature.

**Bulge magnitude** defines how the surfaces blend together at the starting and ending edges. The default value for this property is 0 (no bulge). Setting a higher value gives you a better understanding of the curvature of the model. When working in a shaded display instead of a wireframe display, you can view the isoline representation by hovering the cursor over the model. The default settings for the system variables can be changed in the 3D objects section of the Options dialog box. The settings can range from 0 to 200.

**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.

**Exercise 11-2**

Complete the exercise on the companion website.

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Referring to Figure 11-6, when working in a wireframe display, isolines appear to represent the curved surfaces of the surface model. The SURFU and SURFY system variables control the number of isolines displayed in the U and V directions of the surface. The number of isolines does not include the lines defining the object’s boundary. The default setting for both system variables is 6. These settings can be changed for a surface by selecting the surface and opening the Properties palette. The U isolines and V isolines properties in the Geometry category control the number of isolines displayed in the U and V directions. See Figure 11-8. Setting a higher value can give you a better understanding of the curvature of the model. When working in a shaded display instead of a wireframe display, you can view the isoline representation by hovering the cursor over the model. The default settings for the SURFU and SURFY system variables can be changed in the 3D objects section of the Options dialog box. The settings can range from 0 to 200.
Examples of blend surfaces with different continuity settings are shown in Figure 11-10.

**Bulge magnitude** defines the size or “bulge” of the radial transition where the surfaces meet. See Figure 11-11. The default setting is 0.5. 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.

Blend surfaces can also be created between the edges of regions or solid objects. An example of creating a blend surface between two solid subobject edges to form a cap is shown in Figure 11-12.

**PROFESSIONAL TIP**
Continuity settings are retained when exporting a 3D model to other 3D CAD modeling applications.

**Figure 11-10.** Blend surfaces created with different continuity settings. A—G0 continuity. B—G1 continuity. C—G2 continuity.

**Figure 11-11.** Bulge magnitude determines the size of the radial transition at the edges where surfaces meet. A—The original model consists of two surfaces created from extruded arcs. B—The model after capping the ends with blend surfaces. For each surface, the continuity is set to G1 and the bulge magnitude is set to 0. C—For each surface, the continuity is set to G1. The bulge magnitude at the left edge is set to 0.5. The bulge magnitude at the right edge is set to 1.

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 in Figure 11-13. In Figure 11-13A, 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. In Figure 11-13B, the patch surface is created with the continuity set to G1. This is the appropriate setting for the patch surface. In this case, you would want the patch to be tangent to the existing surface. The default bulge magnitude (0.5) is used. In Figure 11-13C, the patch surface is created with the continuity set to G1, but the bulge magnitude is set to 1. Notice the different result.

When using the **Guides** option, draw a curve to serve as the guide curve prior to selecting the **SURFPATCH** command. In Figure 11-14A, the top of the tent requires a patch. A new middle post has been added to the model, and a spline is drawn to serve as a guide curve. After selecting the **SURFPATCH** command, select the surface edge. Then, select the **Guides** option, select the spline, and press **[Enter]**. Press **[Enter]** to create the patch surface, or adjust the continuity and bulge magnitude settings as needed. See Figure 11-14B.

**Exercise 11-3**
Complete the exercise on the companion website. www.g-wlearning.com/CAD

**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.

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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.

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Examples of creating patch surfaces are shown in Figure 11-13. In Figure 11-13A, 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. In Figure 11-13B, the patch surface is created with the continuity set to G1. This is the appropriate setting for the patch surface. In this case, you would want the patch to be tangent to the existing surface. The default bulge magnitude (0.5) is used. In Figure 11-13C, the patch surface is created with the continuity set to G1, but the bulge magnitude is set to 1. Notice the different result.

When using the **Guides** option, draw a curve to serve as the guide curve prior to selecting the **SURFPATCH** command. In Figure 11-14A, the top of the tent requires a patch. A new middle post has been added to the model, and a spline is drawn to serve as a guide curve. After selecting the **SURFPATCH** command, select the surface edge. Then, select the **Guides** option, select the spline, and press **[Enter]**. Press **[Enter]** to create the patch surface, or adjust the continuity and bulge magnitude settings as needed. See Figure 11-14B.
Figure 11-13. Creating a patch surface to close an opening in a surface model. A—The original model. The single surface edge indicated forms a closed loop and is selected to generate the patch surface. B—A patch surface created with the continuity set to G1 and the bulge magnitude set to 0.5. C—A patch surface created with the continuity set to G1 and the bulge magnitude set to 1.

Figure 11-14. Using a guide curve to constrain the shape of a patch surface. A—The original model. The middle post has been added. The spline is drawn for use with the Guides option. B—The patch surface created after selecting the spline as the guide curve. Notice the resulting shape.

In Figure 11-15, 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. See Figure 11-15B.

Using the Curves option of the SURFPATCH command is shown in Figure 11-16. In Figure 11-16A, multiple curves have been created to design a sophisticated freeform shape. First, the LOFT command is used to create six loft surfaces defining the sides. Refer to the shaded surfaces shown in Figure 11-16B. Then, the Curves option of the SURFPATCH command is used to create three surface patches forming the top of the model. Refer to Figure 11-16C. To use the Curves option, enter it after selecting the SURFPATCH command. Then, select the curves defining the patch surface.

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 PREVIEWCREATIONTRANSPERANCY 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.

Exercise 11-4

Complete the exercise on the companion website.
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Figure 11-15. Using the Chain option to select multiple surface edges to define the patch surface. A—The original model is a surface converted from a mesh model. The top opening includes eight surface edges. B—The patch surface created after using the Chain option to select the edges. The continuity is set to G2 and the bulge magnitude is set to 0.7.

Figure 11-16. Using the Curves option of the SURFPATCH command to create surface patches from multiple curves. A—Curves used with the LOFT command to create loft surfaces defining the sides. B—Surface patches using the Curves option to form the top of the model.

PROFESSIONAL TIP

The PREVIEWCREATIONTRANSPERANCY 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.
Offsetting Surfaces

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.

Select the SURFOFFSET command and select the surface or region to offset. Then, press [Enter]. A preview of the offset surface appears with offset arrows indicating the direction of the offset. See Figure 11-17A. Next, specify the offset distance. If the design calls for the offset surface to be located on the opposite side, select the Flip direction option. You can offset the surface to both sides by selecting the Both sides option. After specifying the offset distance, press [Enter] to create the offset surface. In Figure 11-17B, the hair dryer housing is offset to the inside of the existing surface at a distance of .125.

The Solid option allows you to create a new solid based on the specified offset distance. In Figure 11-17C, the Solid option is used to create a solid. This is similar to using the THICKEN command, as discussed later in this chapter.

The Connect option can be used when you have more than one surface to offset and you need to maintain connection between the surfaces. When using this option, the original surfaces must be connected. See Figure 11-18.

The Expression option allows you to enter an expression to constrain the offset distance. Surface associativity must be enabled in order to use this option.

Exercise 11-5

Complete the exercise on the companion website. www.g-wlearning.com/CAD

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. Commands used to fillet solids are introduced in Chapter 10.

To create a fillet surface, select the SURFFILLET command. First, set a radius using the Radius option. Then, select two surfaces. See Figure 11-19A. 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 FILLETRAP3D system variable. If you do not specify a radius, the current FILLETRAP3D system variable setting is used.
Creating a fillet surface between surfaces that do not meet is shown in Figure 11-19A. You can also create a fillet surface between two surfaces meeting at an edge or intersect, as shown in Figure 11-19B and Figure 11-19C.

After selecting the two surfaces, a preview appears and you can drag the fillet grip to adjust the radius dynamically. If dynamic input is turned on, you can type a value. If the fillet radius is too large, AutoCAD displays a message stating that the fillet surface cannot be created. When using the Radius option to set the radius, you can select the Expression option to enter a mathematical expression for the radius value.

After creating the fillet surface, you can use the Properties palette to edit the fillet surface radius. A radius of zero is not permissible.

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.

Exercise 11-6
Complete the exercise on the companion website.
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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. The Expression option allows you to enter a mathematical expression for the extension distance. If you specify a distance and press [Enter], the surface is extended using the default settings. See Figure 11-20.

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.

Exercise 11-7
Complete the exercise on the companion website.
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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. When you are finished trimming, press [Enter] to end the command. See Figure 11-21.

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, as discussed in the next section.

Exercise 11-8

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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. See Figure 11-22. 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. See Figure 11-22B. The areas to be trimmed are then selected to complete the sides. See Figure 11-22C.

To use a curve instead of an existing surface for trimming, select the curve after selecting the object to trim. 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.

PROFESSIONAL TIP

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

Exercise 11-9

Complete the exercise on the companion website.
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Untrimming Surfaces

If you need to restore a trimmed surface back to its original shape, use the \texttt{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 \texttt{Surface} option. The \texttt{SURFUNTRIM} command untrims surfaces trimmed by the \texttt{SURFTRIM} command. It does not untrim surfaces trimmed using the \texttt{PROJECTGEOMETRY} command.

\textbf{PROFESSIONAL TIP}

Open the Properties palette if you are unsure if a surface has been trimmed. The \textit{Trimmed surface} property setting indicates the surface trim status.

Exercise caution if you are trimming an associative surface using another surface as a cutting object. When using another surface as the cutting object, turn off surface associativity (set the \texttt{SURFACEASSOCIATIVITY} system variable to 0) before trimming to avoid potential problems in future edits.

\section*{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.

As discussed earlier in this chapter, a NURBS surface is created when the \texttt{SURFACEMODELINGMODE} system variable is set to 1. In addition, NURBS surfaces are non-associative. The setting of the \texttt{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. For example, in computer animation work, NURBS surface modeling techniques are commonly used for modeling characters to produce the organic shape desired. In AutoCAD, you can create highly sophisticated, freeform shapes using NURBS surface models. This chapter introduces the NURBS surface modeling tools available in AutoCAD.

\section*{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:

- \textbullet{} Create other surfaces, such as blend surfaces, patches, fillets, and offset surfaces.
- \textbullet{} Use the commands presented in this chapter.
- \textbullet{} Convert the surfaces into NURBS surfaces.
- \textbullet{} 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:

- \textbullet{} Set the \texttt{SURFACEMODELINGMODE} system variable to 1 (on). With this setting, NURBS surfaces are created.
- \textbullet{} 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 \texttt{SPLINE} command. Splines used for NURBS surface models are typically created with the \texttt{Method} option of the \texttt{SPLINE} command set to \texttt{CV}. This creates splines with control vertices (CVs), also known as \textit{CV splines}. See Figure 11-23. Control vertices play a major role in editing NURBS surfaces.
- \textbullet{} 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 \textit{cannot} be converted back to a procedural surface. In addition, once a NURBS surface is created, it \textit{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.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{Figure_11-23.png}
\caption{Splines used to create NURBS surfaces are typically created as CV splines. A—A CV spline used for creating an extruded NURBS surface. B—When you select a CV spline, the control vertices are displayed for editing. C—A NURBS surface model after using the \texttt{EXTRUDE} command to extrude the spline.}
\end{figure}
Creating and Editing NURBS Surfaces

If the surface model you are creating will have only slight modifications, a simple approach is to create a procedural surface and then convert it to a NURBS surface. The \texttt{CONVTONURBS} command is used to convert a procedural surface to a NURBS surface. After entering this command, select the surface to convert. When you press [Enter], the surface is converted.

If the intention is to create more complex, freeform shapes, you can create NURBS surfaces without first converting procedural surfaces. In this case, set the \texttt{SURFACEMODELINGMODE} system variable to 1. Create profile geometry using splines and use the appropriate surface modeling commands.

The model in Figure 11-24 shows an example of editing a NURBS surface converted from a procedural surface. The original model shown in Figure 11-24A is a loft surface created from open profiles. In Figure 11-24B, the model is shown after using the \texttt{CONVTONURBS} command. The model is shown selected with control vertices displayed (the wireframe view is shown for reference only). When you select a NURBS surface, control vertices do not appear by default. The display of control vertices is controlled by the \texttt{CVSHOW} command. To display control vertices, select the \texttt{CVSHOW} command and then select the NURBS surface. Then, press [Enter]. You can also select the command after initially selecting the surface. To remove the display of control vertices, select the \texttt{CVHIDE} command. Using this command removes the display of control vertices from all objects in the drawing.

Editing control vertices is similar to editing grips. Pick on the control vertex grip and pull or drag. You can press and hold the [Shift] key to select multiple control vertices. You can also use the gizmo that appears. As you pull or drag, the shape of the surface is modified. In Figure 11-24C, the model is shown after editing a control vertex on the back end of the surface. A shaded view of the model after editing and using the \texttt{CVHIDE} command is shown in Figure 11-24D.

For greater control when editing the control vertices of a NURBS surface, you can use the \texttt{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, Figure 11-25A. This gizmo is similar to the move gizmo that appears when working with 3D objects, as discussed in Chapter 10. However, it contains additional grips for setting the tool options and modifying the tangencies of the surface. See Figure 11-25B. 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 \texttt{Move Point} and \texttt{Tangent Direction} options. The \texttt{Move Point} option is used to reshape the surface by moving the base point. The \texttt{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.

Use the 3D edit bar gizmo in the same manner as other gizmo tools. Pull or drag on a grip and then pick to set a distance or use direct distance entry. The movement can be restricted to the X, Y, or Z axis or the XY, XZ, or YZ plane by picking the appropriate axis or plane on the tool. The surface updates dynamically as you drag a grip.

Right-clicking on the tool displays a shortcut menu with additional options for relocating the base point, setting the tangency direction, and aligning the gizmo. The \texttt{Move Point Location} and \texttt{Move Tangent Direction} options serve the same functions as the \texttt{Move Point} and \texttt{Tangent Direction} options previously discussed.

Exercise 11-10

Complete the exercise on the companion website.

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Additional Surface Modeling Methods

Additional methods are available for working with surface models. These methods include converting existing objects to surfaces and using the \texttt{THICKEN} and \texttt{SURFSCULPT} commands to create solid models from surfaces. These methods are discussed in the following sections.

For greater control when editing the control vertices of a NURBS surface, you can use the \texttt{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, Figure 11-25A. This gizmo is similar to the move gizmo that appears when working with 3D objects, as discussed in Chapter 10. However, it contains additional grips for setting the tool options and modifying the tangencies of the surface. See Figure 11-25B. 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 \texttt{Move Point} and \texttt{Tangent Direction} options. The \texttt{Move Point} option is used to reshape the surface by moving the base point. The \texttt{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.

Use the 3D edit bar gizmo in the same manner as other gizmo tools. Pull or drag on a grip and then pick to set a distance or use direct distance entry. The movement can be restricted to the X, Y, or Z axis or the XY, XZ, or YZ plane by picking the appropriate axis or plane on the tool. The surface updates dynamically as you drag a grip.

Right-clicking on the tool displays a shortcut menu with additional options for relocating the base point, setting the tangency direction, and aligning the gizmo. The \texttt{Move Point Location} and \texttt{Move Tangent Direction} options serve the same functions as the \texttt{Move Point} and \texttt{Tangent Direction} options previously discussed.
Converting Objects to Surfaces

AutoCAD provides a great deal of flexibility in converting and transforming objects. For example, a simple line can be quickly turned into a surface. Refer to Figure 11-26.

1. Use the Thickness property in the Properties palette to give the line a thickness. Notice that the object is still a line object, as indicated in the drop-down list at the top of the Properties palette.
2. Select the CONVTOSURFACE command.
3. Pick the line. Its property type is now listed in the Properties palette as a surface extrusion.

Other objects that can be converted to surfaces using the CONVTOSURFACE command are 2D solids, 3D solids, arcs with thickness, open polylines with a thickness and no width, regions, and planar 3D faces.

The THICKNESS system variable can be used to assign a default Thickness property to new 2D objects that you create, such as lines, polylines, polygons, and circles. The value of the THICKNESS system variable does not affect the thickness of a planar surface or 3D surfaces.

Thickening a Surface into a Solid

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. See Figure 11-27.

The THICKEN command can be used in conjunction with the CONVTOSURFACE command to quickly convert a 2D line into a solid. For example, create the line and then use the Properties palette to give the line a thickness. Convert the line into a surface using the CONVTOSURFACE command, and then use the THICKEN command to create a solid from the surface.

By default, the original surface object is deleted when the 3D solid is created with THICKEN. This is controlled by the DELOBJ system variable. To preserve the original surface, change the DELOBJ value to 0.

Sculpting: 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 as discussed in Chapter 10. 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.

Notice the surfaces in Figure 11-28 create a watertight volume where trimming of the surface can occur. A single-surface enclosed area or multiple surfaces that create an enclosed area are considered to be watertight objects. Watertight objects can be converted to a solid.

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). This is shown in Figure 11-28.

As a best practice, use CONVTOSOLID for converting watertight meshes to solids. Use SURFSCULPT for converting surfaces or solids.

The watertight surface area must have a G0 continuity (positional continuity) for the SURFSCULPT command to work properly.
Chapter Review

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter review on the companion website. www.g-wlearning.com/CAD

1. Name the two basic types of surface models in AutoCAD.
2. Which type of surface is created when the SURFACEMODELINGMODE system variable is set to 0?
3. What is an associative surface?
4. Which system variable determines whether a surface model is associative when created?
5. When editing the shape of an associative surface, what should be selected to maintain the surface associativity?
6. What is a network surface?
7. What two system variables set the number of isolines displayed in the U and V directions of a surface model?
8. What is the purpose of the SURFBLEND command?
9. What are the three options used to define surface continuity? What is the result of using each option?
10. Define bulge magnitude.
11. What is the purpose of the SURFPATCH command?
12. What is the purpose of the SURFFLAG command?
13. How do you create a new solid when using the SURFFLAG command?
14. What are the two creation type options available when using the SURFEXTEND command? What is the purpose of each option?
15. What are the three object types that can be used as cutting objects when trimming a surface?
16. What are the two basic ways to create a NURBS surface?
17. What command is used to display control vertices on a NURBS surface?
18. What command can be used to convert a 2D line or polyline into a surface model?
19. What is the purpose of the THICKEN command and which type of object does it create?
20. What is the preferred command to convert a watertight series of surfaces into a solid?

Drawing Problems

1. Create a loft surface from the three cross sections shown. The spacing between cross sections is 5 units. The circle cross section is a ∅6 circle. Use your own coordinates for the circle center point and the ellipse center points. To draw the two ellipse cross sections, refer to the dimensions given. The ellipses are centered on the same center point along the Z axis (refer to the top view). The major axis of the top ellipse is parallel to the minor axis of the lower ellipse. Use your own orientation for the ellipse axes relative to the circle (the exact orientation is not important). When creating the loft, create a procedural surface with associativity. After creating the loft, edit it by changing the dimensions of the ellipse cross sections. Refer to the dimensions given. Save the drawing as P11_01.

<table>
<thead>
<tr>
<th>Cross Section</th>
<th>Dimensions (Loft A)</th>
<th>Edited Dimensions (Loft B)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ellipse A</td>
<td>Major diameter = 3.80</td>
<td>Major diameter = 3.00</td>
</tr>
<tr>
<td></td>
<td>Minor diameter = 2.70</td>
<td>Minor diameter = 2.00</td>
</tr>
<tr>
<td>Ellipse B</td>
<td>Major diameter = 2.70</td>
<td>Major diameter = 6.00</td>
</tr>
<tr>
<td></td>
<td>Minor diameter = 2.30</td>
<td>Minor diameter = 4.00</td>
</tr>
</tbody>
</table>
2. Create the cell phone case shown. Create the case as an extruded surface. Create a patch surface for the top surface. Then, use the arcs to trim the sides.
   A. Draw the base profile in the top view using the dimensions given.
   B. Draw the arcs in the top view. The arcs should extend past the perimeter of the cell phone case. Draw the first arc on the right side of the profile and then mirror it to the other side. Move the arcs along the Z axis so they are located .75 units above the bottom of the cell phone case.
   C. Extrude the base profile to a height of .5 units.
   D. Create the top of the cell phone case by creating a patch surface with C2 continuity and a bulge magnitude of .125.
   E. Trim the sides by projecting the arcs and selecting areas to trim.
   F. Save the drawing as P11_02.

3. Create the hair dryer handle shown. Create the base profile using a spline and a straight line segment. Use the profile to create a planar surface. Then, extrude the planar surface and fillet the bottom end. Finally, convert the model to a NURBS surface and edit the top surface.
   A. Draw a spline and a line using the profile shown. Use the CV option of the SPLINE command to create a CV spline. Dimensions are not important. Create the general shape by picking points to define the control vertices as indicated. Draw a line to form the straight segment at the end of the handle. Then, use the JOIN command to join the line to the spline. The resulting object should be a single, closed spline.
   B. Create a planar surface from the profile.
   C. Extrude the planar surface to a height of .85 units.
   D. Use the SURFFILLET command to fillet the bottom end of the handle. Use a radius of .375 units. Set the trimming mode to Yes.
   E. Using the CONVTONURBS command, convert the model to a NURBS surface. Edit the control vertices of the top surface to create a different shape.
   F. Save the drawing as P11_03.
4. Create the computer speaker using surface modeling commands. Use associative surfaces and edit the height from 4.5 units to 6 units, as shown. Save the drawing as P11_04.

5. Create the kitchen chair shown. In earlier chapters, you created this model as a solid model. In this problem, use surface modeling commands to create the model. Use the drawings shown to create the seat, seatback, legs, and crossbars.
   A. Create the top of the seat using the profile shown. Use the edge profile to create the curved, receding edge extending to the bottom of the seat. The bottom of the seat is 3" below the top.
   B. Create the profile geometry for the seatback. Loft the circular cross sections to create the supports. Loft the square and circular cross sections to create the bow.
   C. Create the framework for lofting the legs and crossbars as shown on the next page. The crossbars are at the midpoints of the legs. Position the lines for the double crossbar about 4.5" apart. The legs transition from ∅1.00 at the ends to ∅1.25 at the midpoints. The crossbars transition from ∅0.75 at the ends to ∅1.00 at a position 2" from each end.
   D. Save the drawing as P11_05.
Profiles for Legs and Crossbars

Completed Model