Additional Surface Tools

Inventor includes several additional surface tools, techniques, and related functions. Surface tools are appropriate for some basic applications, and are commonly used when other tools cannot produce the desired results. Some surface functions are primarily for working with imported, converted, and exported surface models.

Replacing Faces

Replace faces to develop or finalize a model or to explore design alternatives or product changes without removing or editing original features. See Figure 13B-1. Access the Replace Face tool to replace the shape of a face with the shape a surface, quilt, solid, or work plane, using the Replace Face dialog box. See Figure 13B-2. The surface or quilt you use to replace a face must be the same size as, or larger than, the existing face. See Figure 13B-3. This requirement is common for creating or modifying solids using surfaces. If the shape is not large enough, or if faces are perpendicular to the new shape, replacement will not be made.

The Select Existing Faces button is active by default, allowing you to choose faces to replace. Next, pick the Select New Faces button and pick surfaces or quilts to define the shape of the new face. If the desired faces include multiple tangent faces, pick the Automatic Face Chain check box so that you do not have to select each face individually. Pick the OK button to generate the new face.

**quilt:** A set of combined surfaces. (Ch. 13 supplement)
**Figure 13B-1.**
An example of exploring an alternative design by replacing a feature face. This example shows a replaced face removing material. A replaced face can also add material to a model by filling the space between the original face and new face.

**Figure 13B-2.**
The Replace Face dialog box.

**Figure 13B-3.**
The new face reference must completely cover the original face.
NOTE

The separate Replace Face feature in the browser does not consume the referenced new faces. Suppress the Replace Face feature to return to the previous design without removing the replaced face.

Sculpting

To *sculpt* a solid, first create intersecting surfaces that define the size and shape, or boundaries, of the solid. See Figure 13B-4A. Then, access the Sculpt tool to create the sculpted feature using the Sculpt dialog box. See Figure 13B-4B. When creating a base feature, the only possible operation is to form a new solid body, as evident by the selected New Solid button. The Add and Remove buttons are available to add or remove material during a secondary sculpt operation. Use the Surfaces button to choose the surfaces to form the feature. A preview appears if Inventor finds appropriate boundaries.

The direction arrows displayed in the model allow you to define the direction from the surfaces used to produce the feature. The arrows should point toward the area where the solid will form. Pick the sphere to specify both directions. You can also set direction by picking the More button in the Sculpt dialog box and selecting the appropriate direction from the flyout corresponding to the surface listed in the Side Selection list. Pick the OK button to generate the solid. See Figure 13B-4C.

*sculpt:* The process of using intersecting surfaces to add or remove solid mass. (Ch. 13 supplement)
Figure 13B-4.
A—An example of surfaces forming sculpt boundaries. 
B—Using the **Sculpt** dialog box to select surfaces and define parameters. 
C—The sculpted solid.
Figure 13B-5 shows an example of a sculpt created using the remove operation. The remove operation does not require a surface on all sides of the removed area. Figure 13B-5B shows selecting the incorrect side from the work plane for removal. To solve this problem, pick the direction arrow in the model or use the Sculpt dialog box. See Figure 13B-5C.

PROFESSIONAL TIP

An unnecessary surface selection can cause an error when sculpting. Delete a selection from the Side Selection list by picking the surface and pressing [Delete].

Figure 13B-4. (continued)
Figure 13B-5.
A—An example of surfaces used to remove material during a sculpt operation.
Figure 13B-5. (continued)
B—Using the **Sculpt** dialog box to remove the solid between boundaries.
Figure 13B-5. (continued)
C—Redefining the side to sculpt. D—The completed sculpt.
Deleting Faces, Lumps, and Voids

Often, especially when you are developing complex feature geometry, unnecessary faces, *lumps*, and *voids* appear. In certain situations, deleting a face, lump, or void is useful and may even be required. You should delete these items as necessary to create a “clean” model and to ensure that mass properties are calculated correctly. Access the **Delete Face** tool to remove faces, lumps, and voids using the **Delete Face** dialog box. See **Figure 13B-6**.

Deleting Faces

The **Select Individual Faces** button is active by default, allowing you to delete faces. Use the **Activate Faces** button to pick the faces to remove. **Figure 13B-6A** shows an example of multiple faces left over from a cut-sweep operation. When deleting faces, be sure you select all of the faces to remove. For example, select six faces to create the final model shown in **Figure 13B-6B**.

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**lump**: Any set of external feature or surface faces created when you develop a solid model. (Ch. 13 supplement)

**void**: Any set of internal feature faces that define a hollow area in a solid. (Ch. 13 supplement)
Figure 13B-6.
A—Unwanted faces left during a cut-sweep operation selected for deletion.
B—Faces removed.
Another application for deleting a face is to remove all volume from a feature or part, transforming the model into a surface. Apply this technique to use an existing solid to generate a surface or quilt for additional modeling purposes. An example is creating a volumeless surface copy of a part for establishing a decal. The surface icon replaces the part icon, and the deleted face operation appears in the browser. See Figure 13B-7.

Figure 13B-7.
An example of using the **Delete Face** tool to create a surface model.
Select the Heal check box to delete a face and repair the gap created by removing the face. When you heal a deleted face, the faces adjacent to the removal come together. Heal deleted faces to remove a specific face without completely removing or editing a feature. See Figure 13B-8. Pick the OK button to delete the faces.

Figure 13B-8.
An example of the effects of deleting and healing a face.

**CAUTION**
Use caution when deleting faces. Be sure the faces you remove do not make the model lose mass, unless this is the desired effect.

**Deleting Lumps**

A lump can be a single extrusion or a complex multi-feature part. In some situations, multiple unnecessary lumps form in a single part. For example, you create two lumps when you extrude a sketch of two individual circles. Later in the design process, you may want to delete one of the lump cylinders. Pick the Select lump or void button and select the lumps to delete. See Figure 13B-9. Pick the OK button to delete the lumps.
Deleting Voids

A void may occur during certain modeling operations. For example, if you extrude a sketch of two rectangles and then shell one of the extrusions, the shell creates a void in both extrusions even though you only selected one face to remove. See Figure 13B-10. A void affects the mass properties of a solid and can complicate the development of additional features that intersect the void. Pick the Select lump or void button and select the voids to delete. Voids are inside features. To select and delete a void, cycle through possible selections using the Select Other tool. Pick the OK button to delete the selected voids.
Do not confuse the process of deleting a face, lump, or void using the **Delete Face** tool with the process of removing a model item by picking the object and pressing the [Delete] key, or by right-clicking on an item and selecting **Delete**. These deleting options are very different and do not allow you to remove faces, lumps, or voids.
Boundary Patches

Use a boundary patch to generate a surface from 2D or 3D sketch geometry or feature edges. Access the Boundary Patch tool to create a boundary patch using the Boundary Patch dialog box. Figure 13B-11 shows selecting 3D sketch geometry to create a boundary patch. You must select a closed loop. Depending

**Figure 13B-11.**
A—Using the Boundary Patch dialog box to create a boundary patch from a 3D sketched silhouette curve.
B—Using the boundary patch as a split tool, and then adding a shell feature.

boundary patch: A surface formed by patching the space within a selected closed region. (Ch. 13 supplement)
on the objects, you may be able to select a loop with one pick, or you may have to pick individual curves. Pick the **OK** button to create the surface.

**Figure 13B-12** shows selecting existing feature edges to create a boundary patch. You must select a closed loop. Depending on the objects, you may be able to select a loop with one pick, or you may have to pick individual edges. **Figure 13B-13** shows an example of creating a boundary patch using the edge of an existing surface. In this example, because the boundary patch is adjacent to tangent surfaces, you have the option of applying a **Free Condition** or **Tangent Condition**. Pick the **OK** button to create the boundary patch.

**Figure 13B-12.**
Forming a boundary patch between existing feature edges.
Figure 13B-13.
Trimming and Extending Surfaces

In order to trim a surface to remove unnecessary information, you must create a cutting tool. Then access the Trim Surface tool to display the Trim Surface dialog box. See Figure 13B-14. If the cutting tool is not selected automatically,

**Figure 13B-14.**
Trimming a surface using the Trim Surface tool.

**cutting tool:** A surface, quilt, 2D sketch curve, work plane or existing feature face intersecting the surface to trim that provides an edge to which the item is trimmed. (Ch. 13 supplement)
use the **Cutting Tool** button to choose the cutting tool. Then select the side of the surface to trim using the active **Remove** button. If you select the incorrect side, pick the **Invert Selection** button to reverse the operation. Pick the **OK** button to trim the surface.

Extend a surface to add material to help create other surface features. Access the **Extend Surface** tool to display the **Extend Surface** dialog box. See **Figure 13B-15**. Use the **Edges** button to choose the surface edges to lengthen. You can select multiple tangent edges automatically by picking the **Edge Chain** check box. Then specify the length of the extension by selecting **Distance** or **To** from the **Extents** drop-down list. Use the **Distance** option to extend the edge by the length specified in the text box. See **Figure 13B-15A**. Pick the **To** option to terminate the edge at a surface or plane using the **Select surface to end the feature creation** button. See **Figure 13B-15B**. If the selected face or plane does not intersect the extrusion path, select the **Check to terminate feature on extended face** check box.

**PROFESSIONAL TIP**

Use the **Extend Surface** tool to extend surfaces used to produce split features and all other features that require a specific size surface.
Figure 13B-15.
A—Using the **Distance** option of the **Extend Surface** tool to extend the edge of a surface a specified distance.  
B—Using the **To** option to extend the edge of a surface to another surface.
Stitching

You may have to *stitch* individual surfaces in order to eliminate separate surfaces that may cause modeling errors. Access the **Stitch Surface** tool to stitch these surfaces together using the **Stitch Surface** dialog box. See **Figure 13B-16**. The **Surfaces** button is active by default, allowing you to select surfaces to stitch. Pick the **OK** button to complete the operation.

**Figure 13B-16.**
Using the **Stitch Surface** dialog box to stitch surfaces side-by-side. One surface is opaque for clarity.

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**stitch**: Combine two or more surfaces to form a single surface or quilt. (Ch. 13 supplement)
Stitching and Promoting

You can import surfaces into Inventor using the **Open** tool or **Import** tool. When you import surfaces into Inventor, the surfaces appear in a nonparametric construction environment. This occurs because the data used to create surfaces in .iges and .sat files, for example, is inconsistent with the data required by Inventor. Once a surface has been imported, you may have to stitch and then **promote** the surfaces in order for the geometry to be useful in a model.

**PROFESSIONAL TIP**

You may be able to bypass the entire process of stitching and promoting surfaces by selecting the **Auto Stitch and Promote** check box of the **Import Options** dialog box. The **Import Options** dialog box is available by picking the **Options...** button in the **Import** and **Open** dialog boxes.

**promote:** Add to the part environment. (Ch. 13 supplement)
Activity

1. Launch Inventor if it is not already open.
2. Open EX12-9.ipt and save it as ACT13-1.ipt.
3. In the ACT13-1.ipt file, access the **Boundary Patch** dialog box and select the 3D sketched silhouette curve to create a boundary patch. You must pick four curves.
4. Use the **Split** tool to split the loft using the following information.
   - Method: Split Solid
   - Split tool: Boundary Patch1
5. Expand **Solid Bodies(2)** in the browser and turn off the visibility of Solid2.
6. Apply the .1” uniform thickness shell shown in **Figure 13B-17A**.
7. Turn off the visibility of Solid3 and turn on the visibility of Solid2.
8. Access the **Parameters** dialog box and change the name of the shell thickness parameter, possibly d14, to SHELL_THICKNESS.
9. Apply the shell shown in Figure 13B-17B using the List Parameters menu option from the Thickness text box to choose the SHELL_THICKNESS parameter.

10. Turn on the visibility of Solid3 and turn off the boundary path visibility.
11. Override the model color to Blue (Clear).
12. Edit Shell1 to change the thickness to .2”. Notice how the thickness of Shell2 automatically updates according to the parameter change. The final part should look like the part shown in Figure 13B-17C.