Mastering Surface Modeling with SOLIDWORKS® 2026

Basic through Advanced Techniques

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Chapter 1: Introduction to Surfaces
Solids or Surfaces – What is the difference

1. Solids vs. Surfaces:

In SOLIDWORKS, both solids and surfaces are known as Solid Bodies and Surface Bodies. They are advanced tools used to easily create complex geometry. These two entities are very similar and may even be considered the same.

Solid Bodies and Surface Bodies are made up of two types of entities: Geometry and Topology. Geometry represents shapes such as a surface, curve, line, arc, spline, or a point. Topology describes relationships such as a face, an edge, or a vertex, which connects or forms an opened or closed boundary.

The images below display two different objects that appear to be different at first glance. Geometrically they are not quite the same. However, topologically, they are alike. Both objects have eight vertices, six faces, and twelve edges.
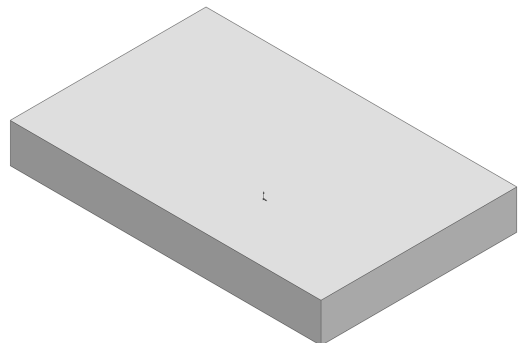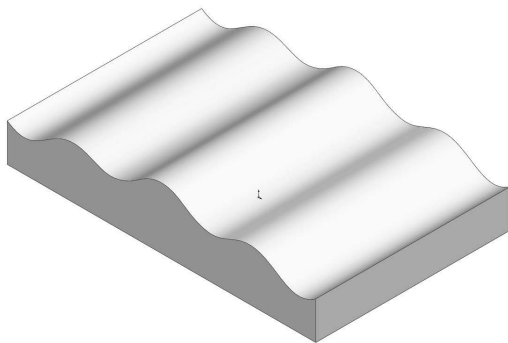
SOLIDWORKS creates surface bodies by forming a mesh of curves called U-V curves. The U curves run along a 4-sided surface and are shown in the magenta color. The curves that run perpendicular to the U curves are called the V Curves, and they are shown in the green color.

Certain commands may offer the preview mesh where these curves are also displayed.
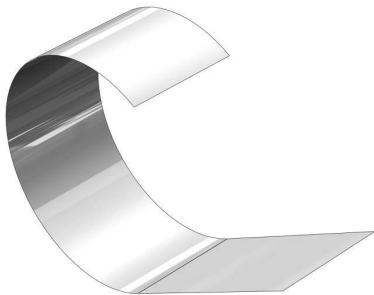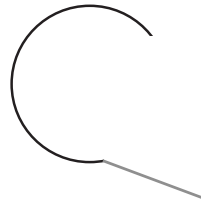
U Curves

V Curves
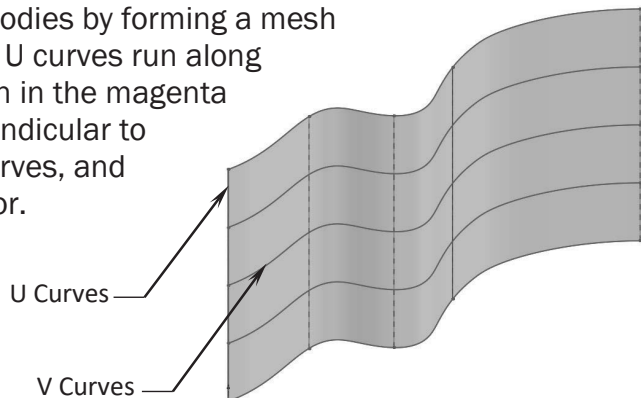
2. Patching with continuity:

Blending issues may often arise when creating or patching with surfaces. The three examples below showcase the common examples of patches frequently found in design work.
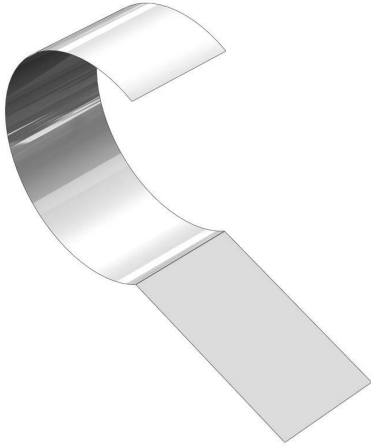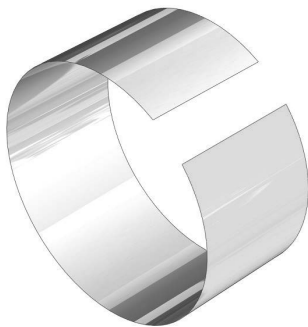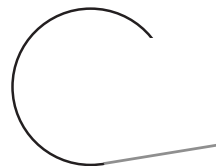
<u>Contact:</u> (also called **C0 Continuity**)

 The yellow surface is in contact with the gray surface. They both share a common edge.
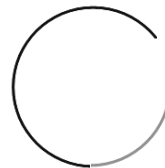
<u>Tangent:</u> (also called **C1 Continuity**)

The yellow surface is in contact with the gray surface. They share a common edge and are also tangent to each other.

<u>Curvature Continuous:</u> (also called **C2 Curvature**)

The yellow surface is in contact with the gray surface. They are tangent and share the same curvature.

Each of the blend types has its own features to assist designers in creating the right patch for a surface model. Though every model may have unique needs, these options will provide a smooth transition between two surfaces.

3. When to use surfaces:

Although surface models can be more difficult to create than solid models, solid models may not be enough for all designs. This is where surfacing becomes useful with the important tools to fill in those gaps.

When determining whether to use solid features or surface features, it is helpful to remember that shapes which can be easily made with the Extrude, Sweep, or Loft commands should be made as solid features. These shapes will frequently require more than one side to be created at the same time while residing within the same feature.

A surface model, on the other hand, is created by constructing one surface at a time. This way, different methods and techniques may be used to create the surfaces, one side at at a time. The surface model shown on the right is a primary example of an instance where surfaces should be used.

Surfaces can be used as reference geometry in order to assist with the construction of a model. The surfaces will then be moved into a Surface Bodies folder and become hidden when the model is complete.

4. When not to use surfaces:

Shapes that can be created as one feature along one direction, such as blocks and cylinders, should be made as solid features.

Solid features take less time to make. Generally, the final design of a model should be a solid model. This is because a solid model can have a material assigned to it, can be analyzed to simulate future issues, and can be used to create production drawings.

It is the best practice to combine both solid and surfacing tools (Hybrid Modeling) in order to take advantage of all the tools available and use SOLIDWORKS' powerful features to their fullest extent.
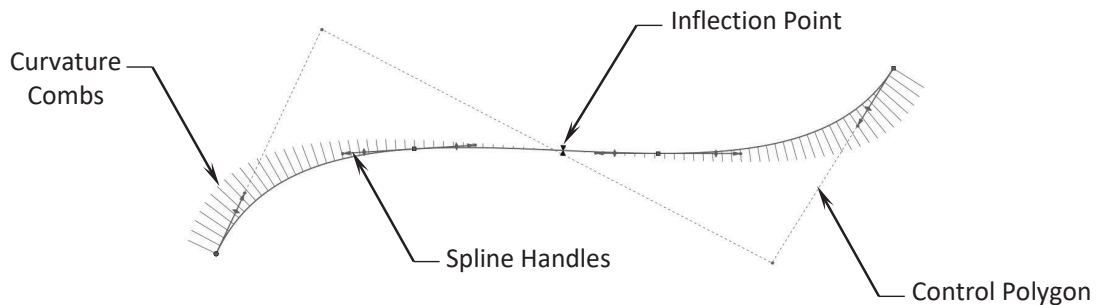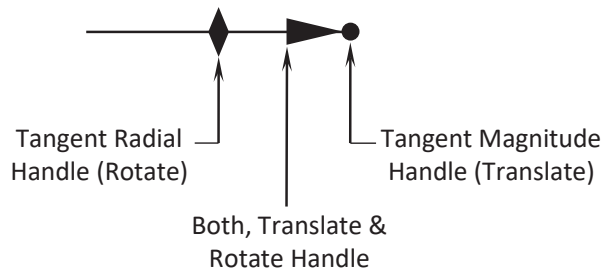
5. The Spline handles:

When creating the surfaces, the Spline tool is often used to create complex curves. This is because it can produce a higher quality curve than other available tools.

Spline can be modified using several controls, including spline points, spline handles, and control polygons. A single spline can have multiple through points and spans (the region between through points). Further, curvature constraints can be applied at each endpoint. At each through point, the tangency vector may be weighed and the tangency direction controlled.
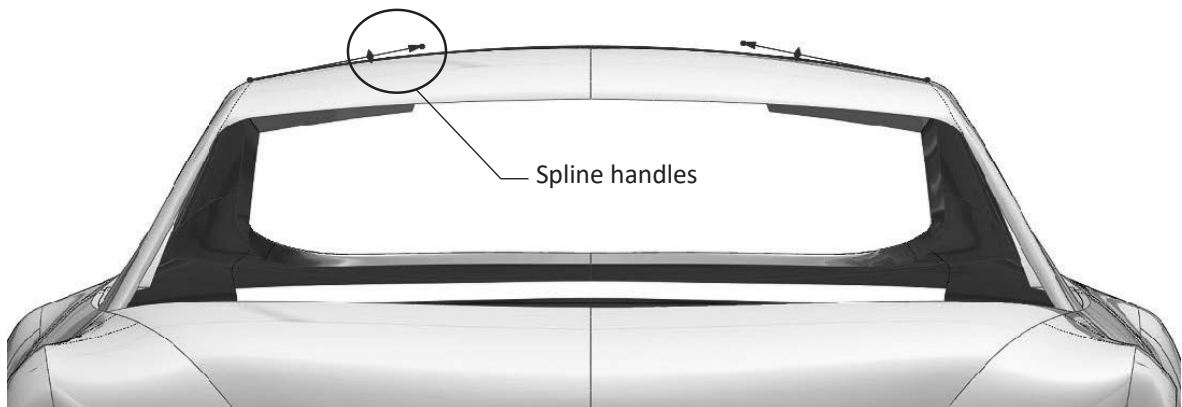
Spline handles

Each Spline includes a set of tools to assist with the manipulation of its curvature. These tools are:

* Curvature Combs
* Spline Handles
* Inflection Points
* Control Polygons

Tangent Radial Handle (Rotate)

Tangent Magnitude Handle (Translate)

Both, Translate & Rotate Handle

Inflection Point

Curvature Combs

Spline Handles

Control Polygon

6. Check your surface model frequently:

Unlike solid features in which one feature may have multiple sides (i.e. an extruded block has six sides, extruded in one direction), surface models are created one face at a time in order to complete each side of the model. Further, surface models are not frequently verified as solid features.

These surfaces may have small gaps or overlapping elements that are difficult to spot. Therefore, it is important to check your surface model for errors after each feature is built.

The Check (or Check Entity) command, under the Evaluate tab, functions as a tool to check surface models and identify geometry issues. If gaps or overlaps exist in the model, the system will prevent the surfaces from knitting into a solid model. It is the best practice to check your model often to reveal any errors.
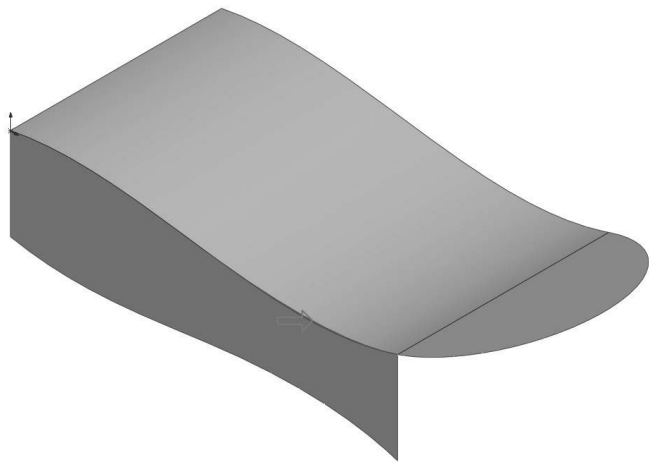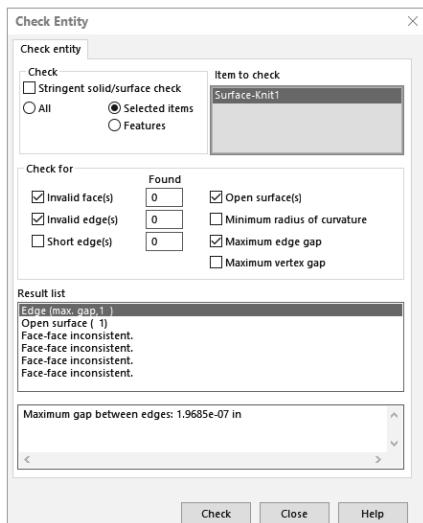
Shortcuts (or hotkeys) are often used to regenerate the surface models.

The hotkeys **Control+B** will rebuild only the new features and their related children, while **Control+Q** will force a complete rebuild of all features in the model.
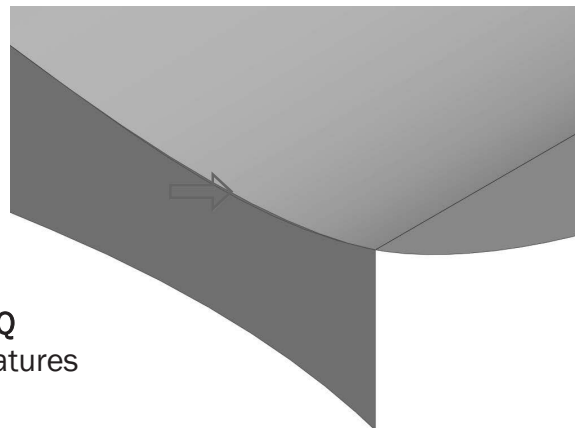
7. The best approach:

Combining solid and surface tools is the best approach to creating your model. Fully utilizing the collaboration of these tools allows for complex geometry and the optimization of different techniques. Techniques such as using a surface to trim a solid, replacing a surface with a different surface, or splitting a solid body into multiple solid bodies, can be necessary to finalize a model.

This unique approach is called Hybrid Modeling, which is a preferred method for most advanced SOLIDWORKS users. The approach, along with other techniques, will be discussed throughout this book when creating the surface models covered in these lessons.
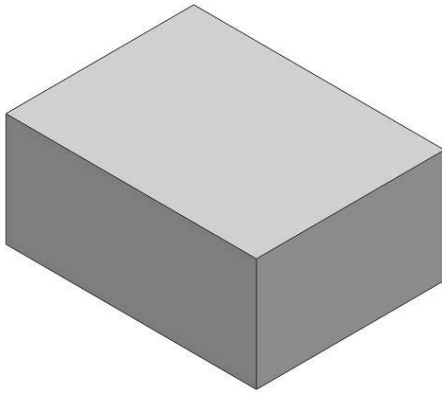
A few important factors to keep in mind from this chapter as we progress through the proceeding lessons:

Solid features are more efficient to use on simple shapes and models. Surface features should be used for complex shapes and models.

Surface features take longer to construct and should be checked frequently for errors.

After all surfaces are created, they should be knitted into one enclosed volume then thickened into a solid model. By turning the surface model into a solid model, we can now analyze for potential issues and create drawings.
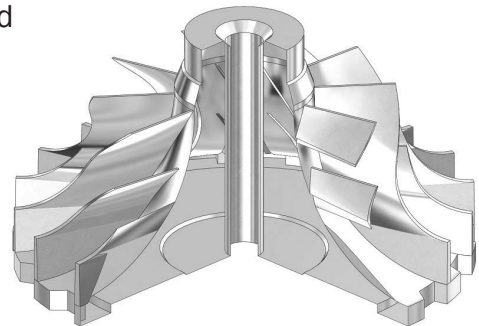
Introduction to Basic Surface Creation

This lesson will walk you through the basic surface creation using the tools that are commonly used to develop the geometry of a surface model such as Extruded Surface, Revolved Surface, Lofted Surface, Boundary Surface, Filled Surface, etc.

First, let us take a look at an example of an **Extruded Surface**.
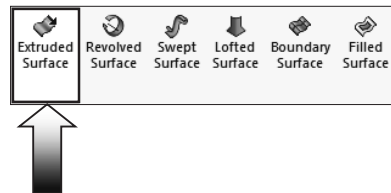
1. Opening a part document:

Select **File, Open**.

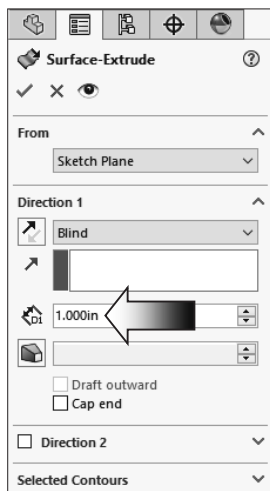Navigate to the Training Folder and open a part document named: **Basic Surface Extrude**.
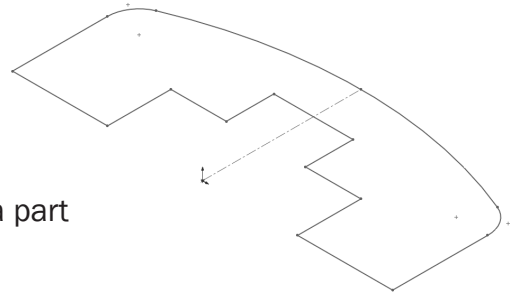
2. Creating an Extruded Surface:

Switch to the **Surfaces** tab and click: **Extruded Surface**.

Select **Sketch1** either from the graphics area or from the FeatureManager tree.

For Direction 1, select: **Blind** and enter **1.00in**. for Depth.

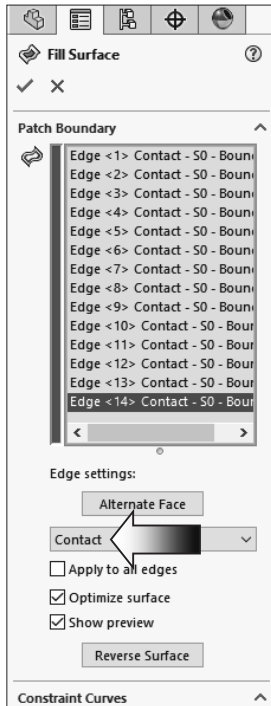The extrude direction is upward.

Click **OK**.
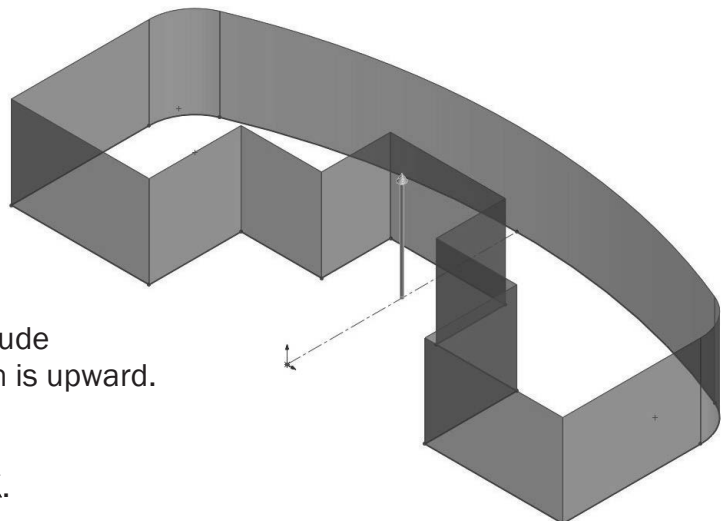
3. Creating a Filled Surface:

Click the **Filled Surface** tool on the **Surfaces** tab.

For Patch Boundary, select all **14 edges** on the bottom of the surface body, as indicated.
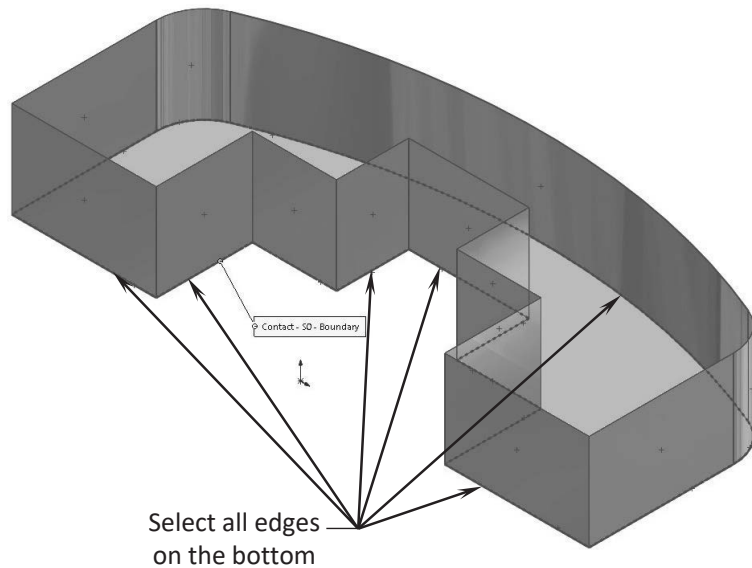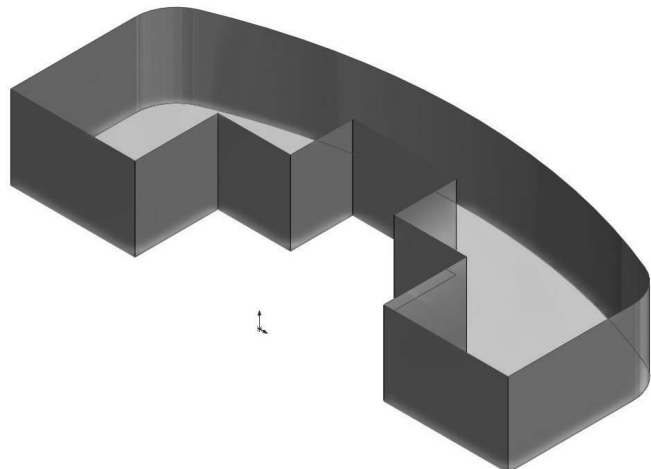
Select all edges
on the bottom

For Edge Settings, select: **Contact.**

Click **OK.**

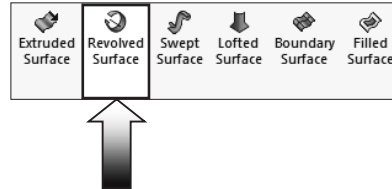<u>Save</u> your surface model as: **Extruded Surfaces Example.**

Next, let us take a look at how the **Revolved Surface** tool is used to create a cylindrical surface body.
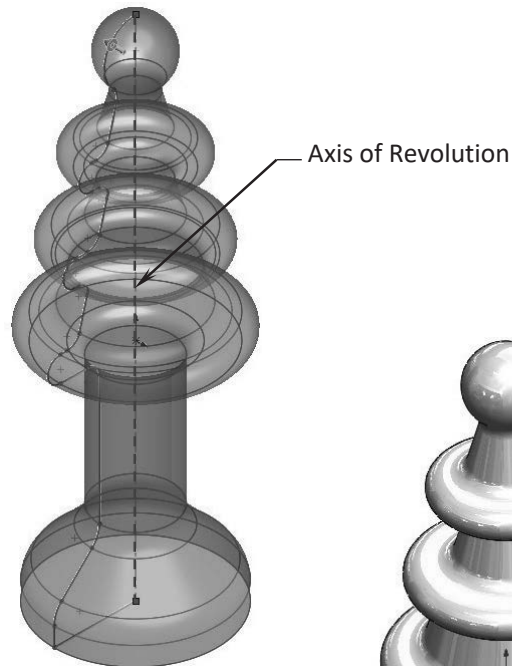
4. Creating a Revolved Surface:

Browse to the Training Folder and open the part document named: **Basic Surface Revolve**.

Switch to the **Surfaces** tab and click: **Revolved Surface**.

Select **Sketch1** from the Feature-Manager tree.

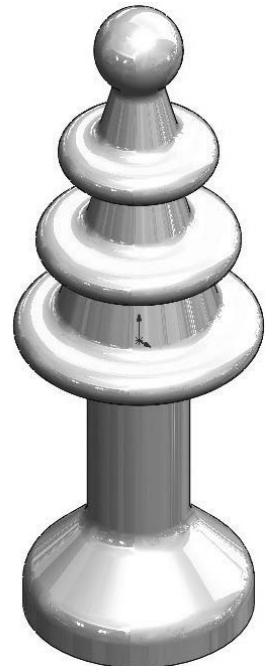The **Centerline** in the middle of the sketch is selected automatically to use as the Axis of Revolution.

Axis of Revolution

For Direction 1, select the **Blind** option from the list.

For Angle, use the default **360.00deg**.

Click **OK**.

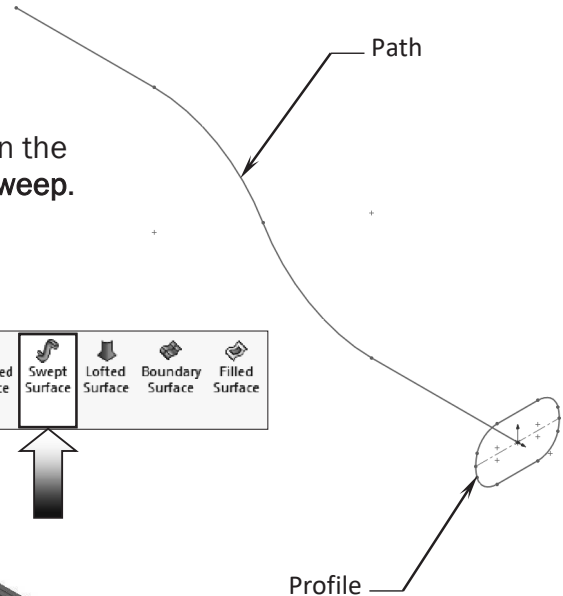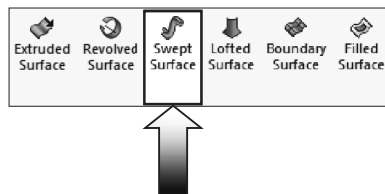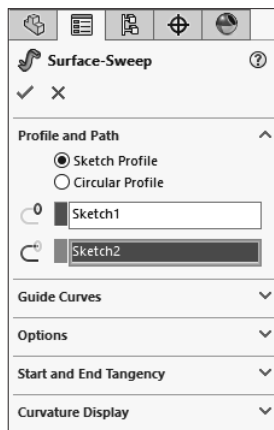Save your surface model as: **Revolved Surfaces Example**.

The **Swept Surface** is next on the list. The Swept Surface moves a sketch profile along a path to define its 3D geometry. The guide curves that are used to control the twists will be discussed in another lesson later on.
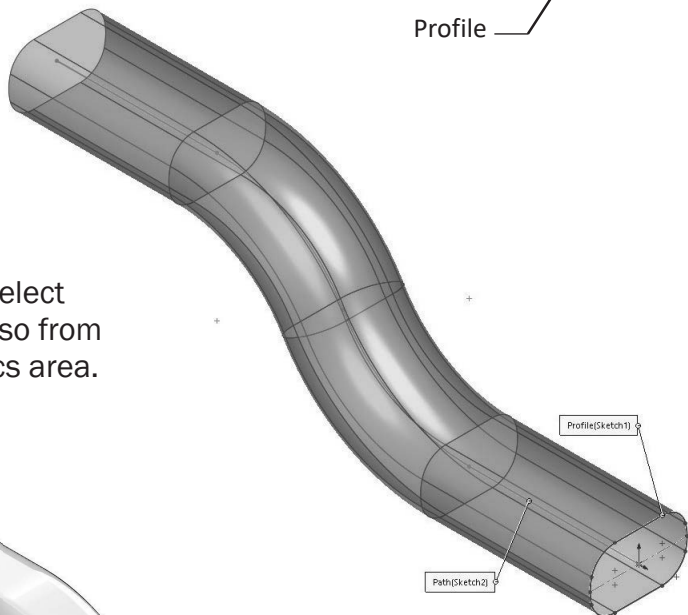
5. Creating a Swept Surface:

Navigate to the Training Folder and open the part document named: **Basic Surface Sweep**.

Switch to the **Surfaces** tab and click: **Swept Surface**.

For Profile, select **Sketch1** from the graphics area.
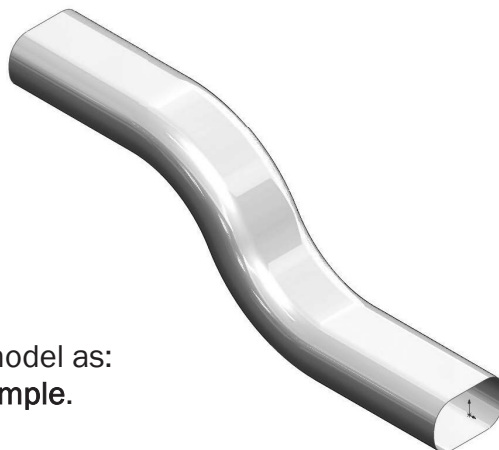
For Path, select **Sketch2** also from the graphics area.

Keep all other parameters at their defaults for this example.

Click **OK**.

Save your surface model as: **Swept Surfaces Example**.

The **Lofted Surface** tool uses 2 (or more) profiles to define the start and the end shape and size of the loft feature. The guide curves that are used to control the transition between the profiles will be discussed in another lesson later on.

6. Creating a Lofted Surface:

Navigate to the Training Folder and open the part document named: **Basic Lofted Surface**.

Switch to the **Surfaces** tab and click: **Lofted Surface**.

For Loft Profiles, select both **Sketch1** and **Sketch2** from the graphics area (select the 2 sketches by clicking the same endpoint in each sketch).
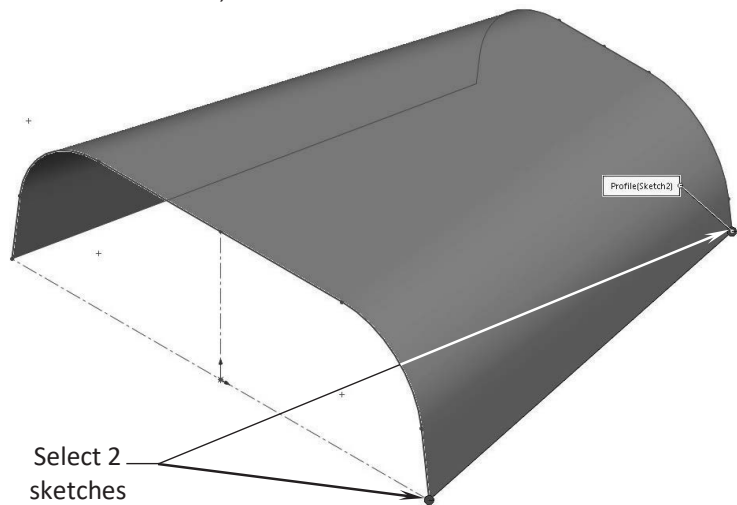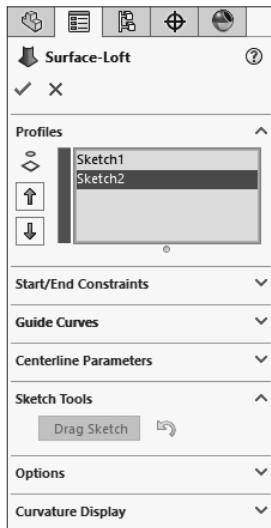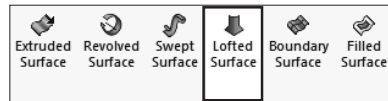
Keep all other parameters at their defaults. We will discuss those options in another lesson later on.
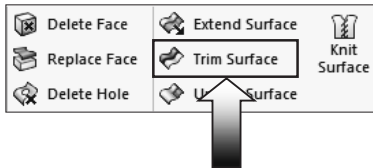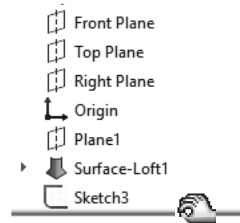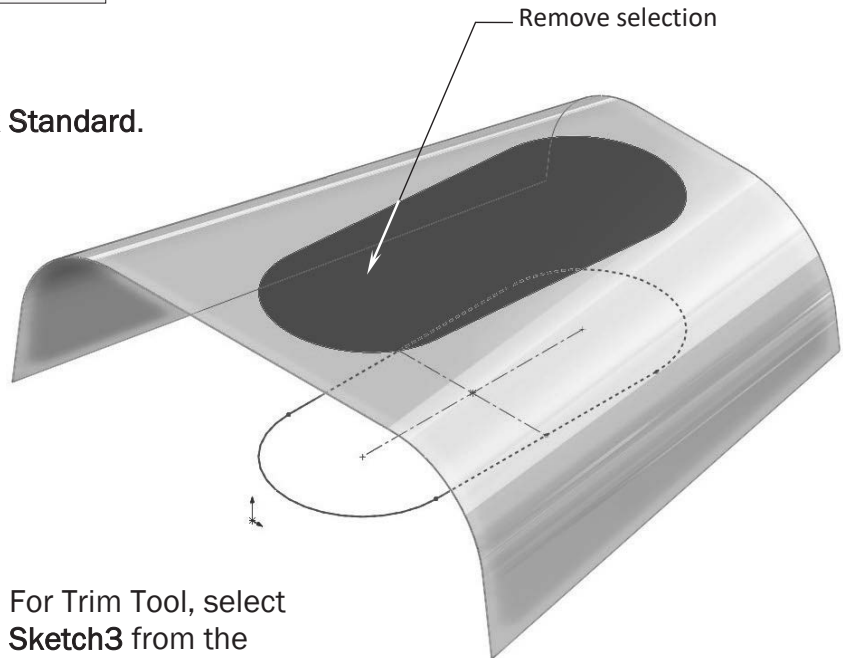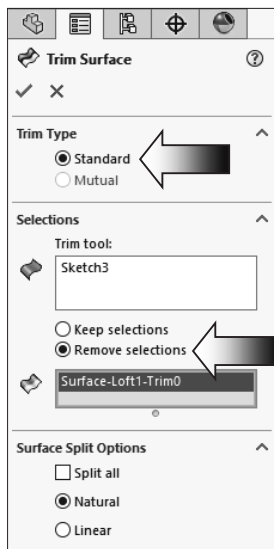
Click **OK**.
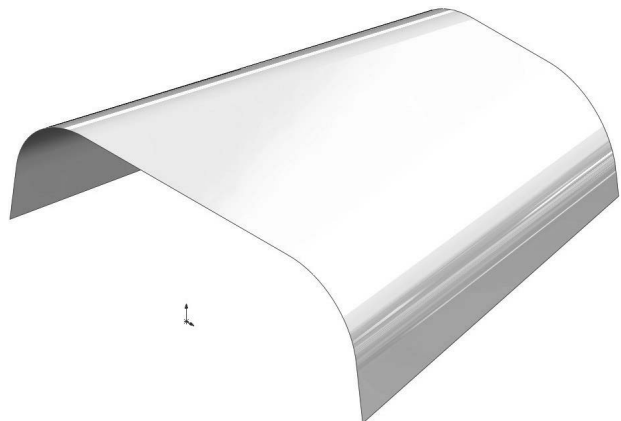
7. Trimming the surfaces:

Drag the **Rollback Bar** down below **Sketch3**.

Click **Trim Surface** on the **Surfaces** tab.

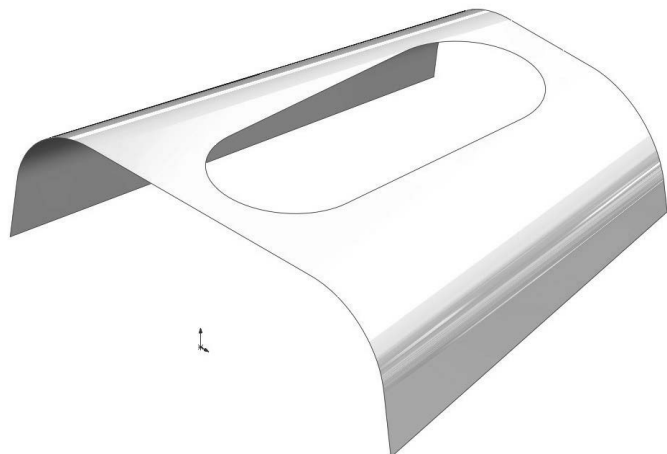For Trim Type, click **Standard**.

Remove selection

For Trim Tool, select **Sketch3** from the graphics area.

Click the **Remove Selection** button and select the area **inside** the slot as noted.

For Surface Split Option, select: **Natural** to match the existing curvature of the surface model.

Click **OK**.

Save your surface model as: **Lofted Surfaces Example**.

The **Boundary Surface** tool is similar to the loft tool; it connects two or more profiles in one or two directions and allows surfaces to be tangent or curvature continuous in both directions (all sides of the surface). The boundary tool delivers a higher quality result than the loft tool in some cases.

8. Creating a Boundary Surface:

Navigate to the Training Folder and open the part document named: **Basic Boundary Surface**.

Switch to the **Surfaces** tab and click: **Boundary Surface**.
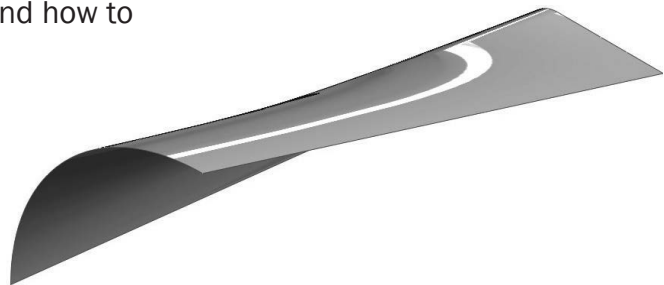
For Direction 1, select both **Sketch1** and **Sketch2** from the graphics area; be sure to select the 2 sketches from the common endpoints to prevent them from twisting.

Select 2 sketches
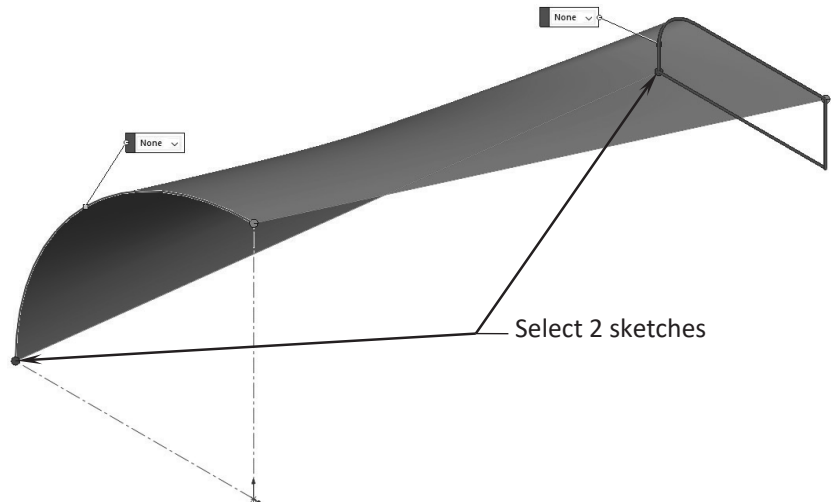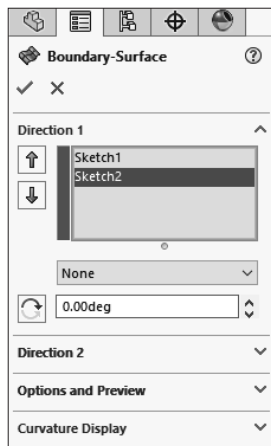
Again, keep all other parameters at their defaults. We will learn what they are for and how to use them in another lesson.
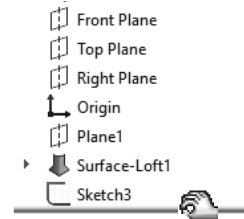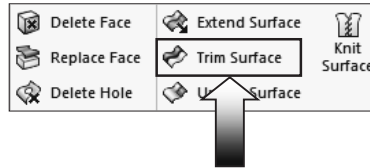
Click **OK.**

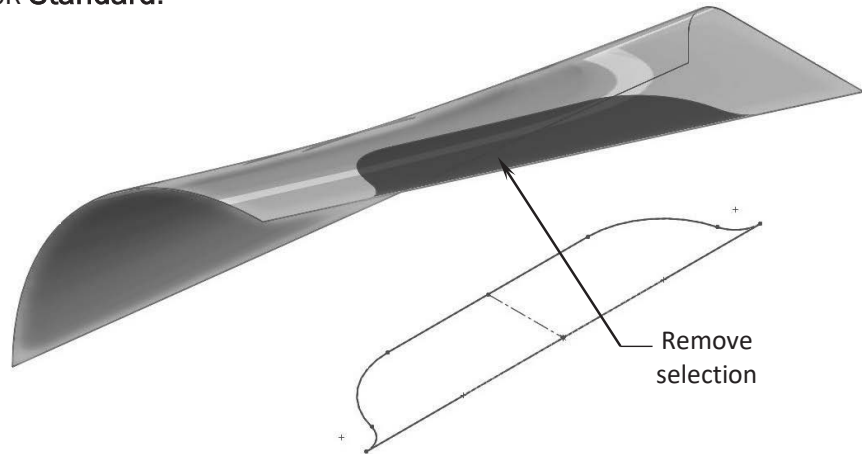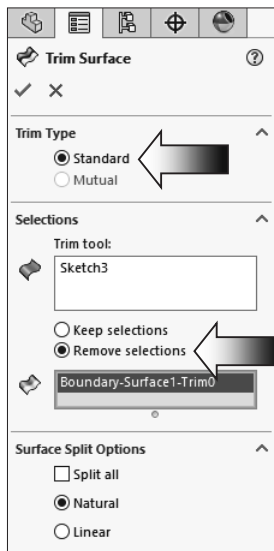Inspect your surface model against the image shown here.

9. Trimming the surfaces

Front Plane
Top Plane
Right Plane
Origin
Plane1
▸ Surface-Loft1
Sketch3

Drag the **Rollback Bar** downward to access **Sketch3**.

Delete Face	Extend Surface	
Replace Face	Trim Surface	Knit Surface
Delete Hole	Untrim Surface	

Click **Trim Surface** on the **Surfaces** tab.

For Trim Type, click **Standard**.

Trim Surface

Trim Type
● Standard
○ Mutual

Selections
Trim tool:
Sketch3

○ Keep selections
● Remove selections
Boundary-Surface1-Trim0

Surface Split Options
☐ Split all
● Natural
○ Linear

Remove selection

For Remove Selection, click <u>inside</u> the area as indicated.

For Surface Split Option, click **Natural**.

Click **OK**.

<u>Save</u> your surface model as: **Boundary Surfaces Example**.

We have examined some of the tools that are commonly used when creating the surface models. We will continue to explore and learn the use of other surfacing tools in the next lessons.