

# Introduction to Surfaces – SolidWorks 2009

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TECHNOLOGY  
SUBJECTS  
SUPPORT  
SERVICE

## Keyboard Button.



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### Surface Modelling

In surface modelling a model is built face by face. Faces created by surface features may knit together to enclose a volume, which may be turned into a solid model.

Surface modelling is used to create faces and features which may not be conveniently produced using solid modelling techniques. Surface tools are employed in situations where they make it easier, more efficient or possible to complete the task at hand.

The focus of this exercise is to give a basic introduction to surfaces and explore the functionality of some surfacing tools.

As we work through the exercise we will explain the terminology associated with surfaces.

### Prerequisite knowledge

To complete this model you should have a working knowledge of Solidworks 2006/2009.

### Focus of lesson

This lesson focuses on using the following surface tools; *Surface Loft, Planar Surface, Filled Surface, Surface Knit, Surface Offset, Replace Face*, as well as *Shell and Extrude* feature tools.

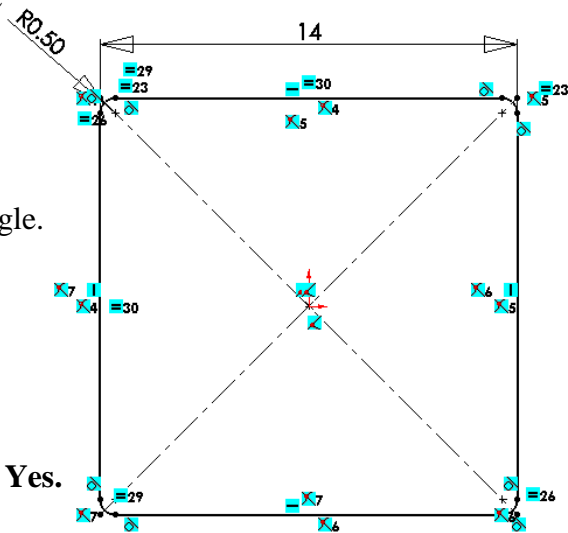
## Getting started. New File

Create a new part file and save it as **Keyboard Button** in the desired location.

## New Sketch

Using **Centre Rectangle** , create the sketch shown on the **Top plane**.

This sketch will represent the profile of the top of the button



## Steps required

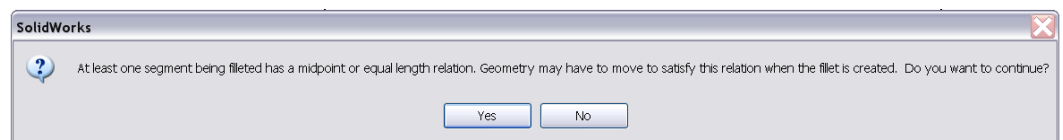
Apply an **Equals Relation** between the horizontal and vertical sides of the rectangle.

Smart dimension one of the sides **14mm**

Add **sketch fillets** of **0.5mm** to the four corners.

Should the warning below appear choose **Yes**.

**Exit** the sketch.



## Inserting a plane

We want to insert a plane parallel to the top plane at a distance of 3mm below it. We will sketch the profile of the base of the button on this plane.


Choose **Insert, Reference Geometry, Plane...** from the drop down menu or choose **Plane** from the pop-up menu.

*Note: The pop-up menu is accessed by pressing 's' on the keyboard.*

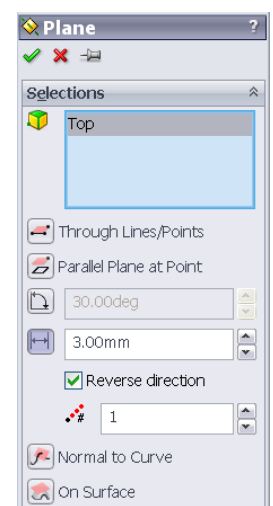
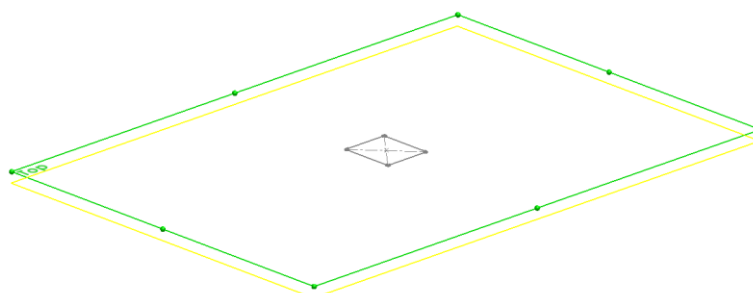


Choose the **Top plane** as the **Reference Entity**

Insert a **distance** of **3mm**

Toggle **Reverse direction** to create the new plane below the Top plane if required. Choose **OK** 

*Note – You may need to zoom out to see the preview of the parallel plane, as shown below.*

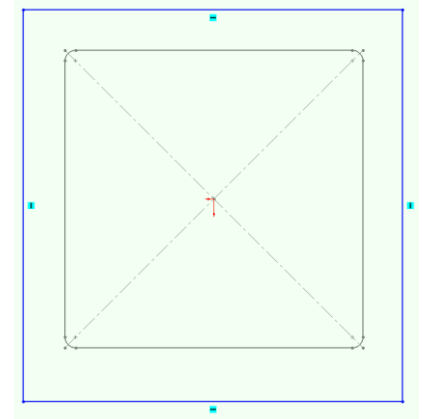


## New Sketch

Create a sketch on the offset plane.

Choose **Top view**.

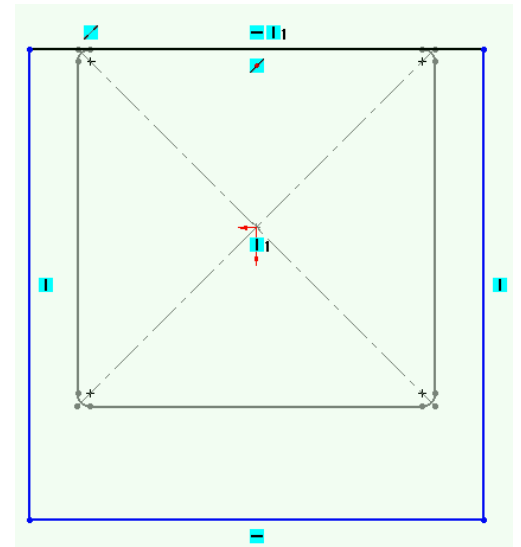
Using **Corner Rectangle**,  create the rectangle shown.



## Add Relations

Add a **Vertical relation** between the midpoint of the top horizontal line of the rectangle and the origin.

Apply a **Collinear relation** between the top horizontal lines of both sketches.

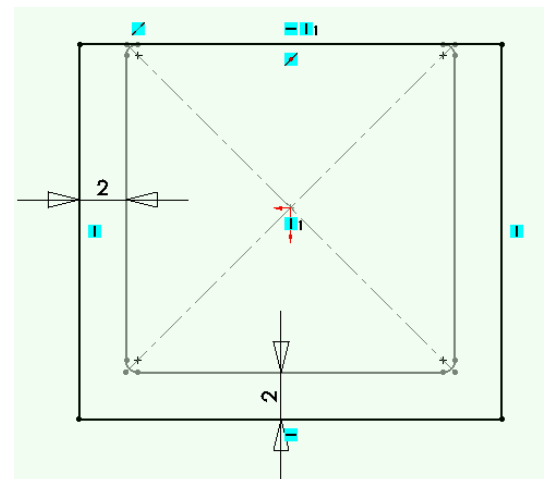
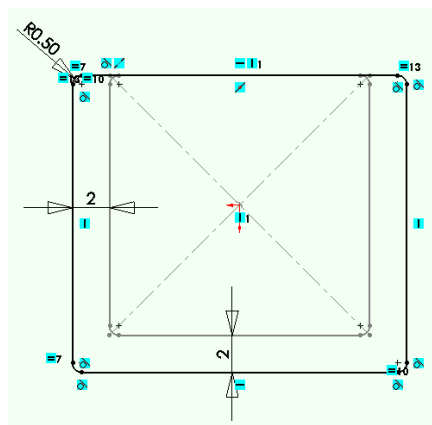


## Smart Dimension

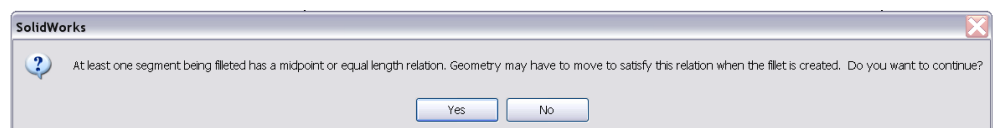
Smart Dimension the sketch as shown below right.

## Sketch Fillet

Create a **0.5mm** sketch fillet on the four corners of the rectangle.

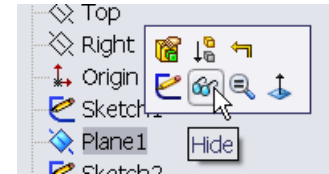


Choose **Yes** for the warning message as before. Exit the sketch.



## Hide Plane1

To hide plane 1; left click on **plane1** on the featuremanager design tree and choose **Hide**

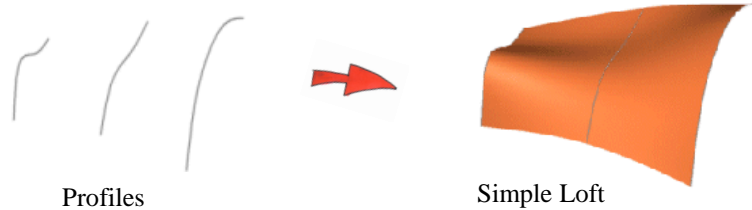


## Orientation

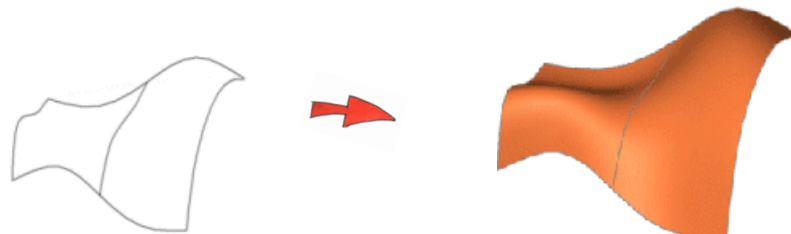
Choose **Isometric view** 

## About Lofted Surface

The difference between lofted surfaces and lofted solids is that surfaces can use edges and curved features between which to loft rather than simply sketches and faces as is the case with solids.



Guide Curves may be added, if necessary, to influence the resultant surface.



Profiles with guide curves

Loft using guide curves

## Creating the loft

Choose **Lofted Surface**  from the Surfaces toolbar

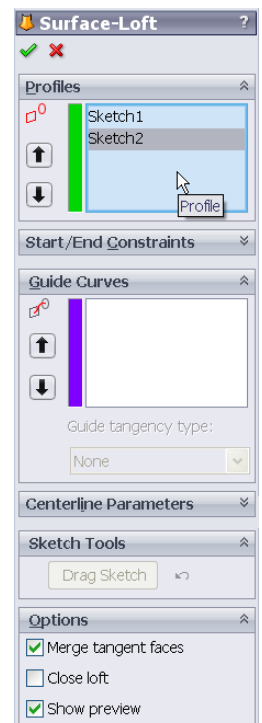
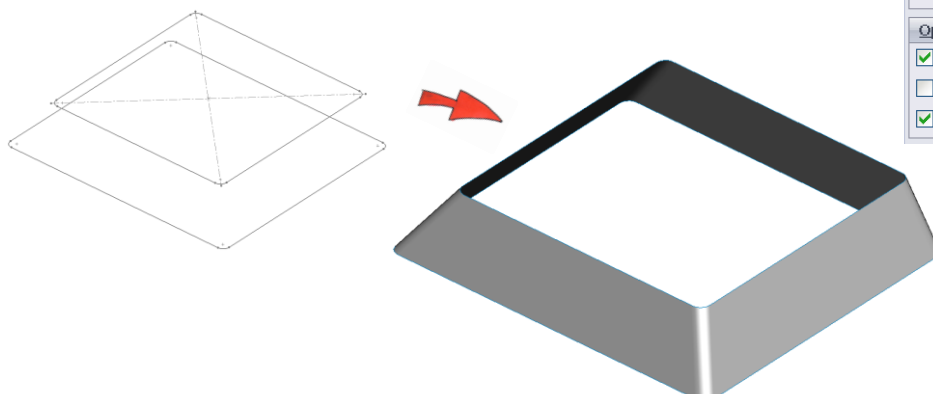
Select the two sketches as **Profiles**

We will not be using any guide curves in this example.

Check - **Merge tangent faces** – This will create one surface between the two profiles as opposed to a series of individual surfaces.

**Show preview** – will provide a preview of the loft.

Choose **OK** 



### Lofted Surface

Examine the model. You will notice that the model is made completely of surfaces, which have zero thickness.

### Image Quality

Zoom into the filleted corners of the model. Circular edges may be appearing as straight line segments. This may be corrected by increasing the image quality.

Choose **Tools, Options, Document properties, Image Quality**

Drag the **Shaded and Draft Quality** slider to a higher setting. Choose **OK**

### Next Step

We now want to close off the base of the button by adding another surface. This surface is going to be defined by the edge of the base of the button. To complete this task we are going to use a **Planar Surface**.

### About Planar Surface

**Planar surfaces** are by definition planar. They may be sketched upon or used as a plane for mirroring. Planar surfaces may be defined in SolidWorks by the same means as they are defined in plane geometry, including two parallel lines.

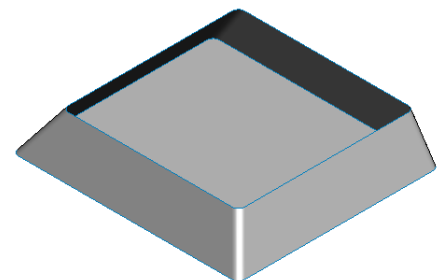
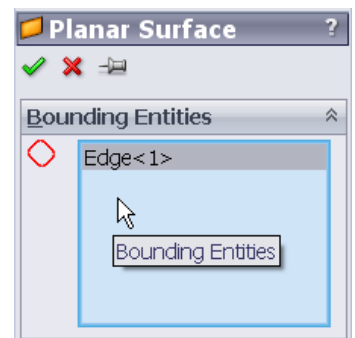
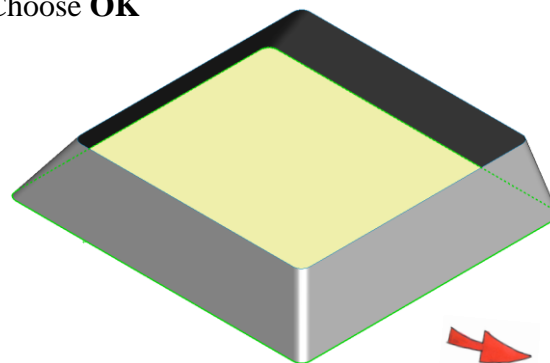
More commonly planar surfaces are defined using a closed sketch e.g. rectangle.

### Planar surface

Choose **Planar Surface**  from the Surfaces toolbar.

Select the base edge as **Bounding Entities**

Choose **OK**



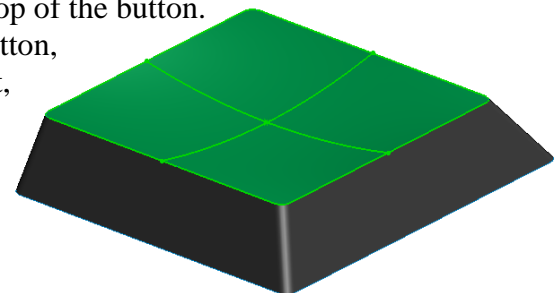
A planar surface has now been added to the base of the model.

### Top Surface

The next step is to create a surface at the top of the button. You will notice from the graphic of the button, that the top surface of the button is not flat, but concave.

For this reason planar surface will not create this surface for us.

The surface is defined by the top edge and two constraint curves, which direct the curvature of the surface in both directions. This surface is created using **Filled Surface**.



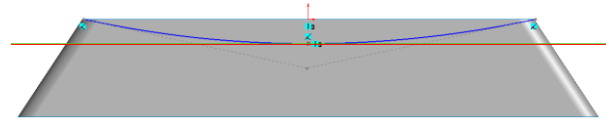
## About Filled Surface

The **Filled Surface** is intended to be used to fill gaps in surface bodies. Constraint curves may be used to drive the shape of the fill between existing boundaries.

## Constraint Curves

We must first create the two constraint curves which will be used to define the filled surface.

The constraint curve is defined by the existing surfaces along with a plane which is placed at the lowest point of its curvature.



We will begin by creating this plane.

## Insert plane

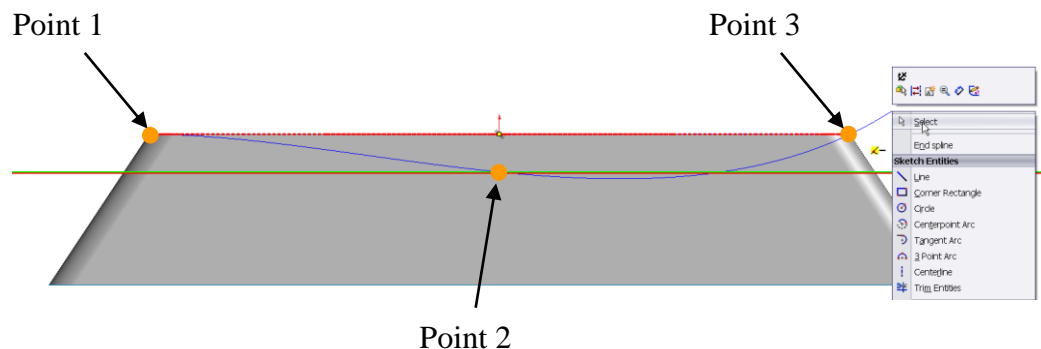
Insert a parallel plane at a distance of **0.75mm** below the **Top plane**, as described in page 2 of this document.

## Create sketch

Create a sketch on the **Front plane**. Choose **Front view**.

## Spline

Choose **Spline**  from the sketch toolbar. Three points will be used to define the spline. Ensure to capture the co-incidental relations. 



## End Spline

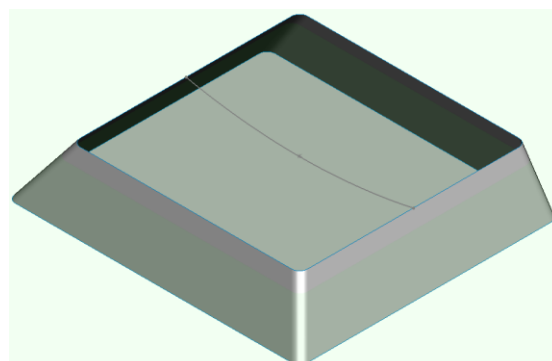
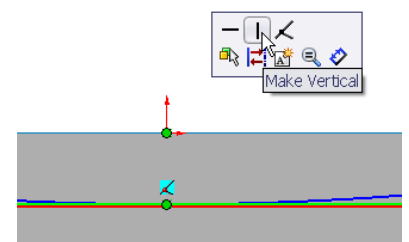
To end the spline; **Right Click** and choose **Select**.

## Add Relation

Add a **Vertical relation** between the lowest point of curvature of the spline and the origin.

Exit the sketch and choose an Isometric View.

The sketch will appear as shown.

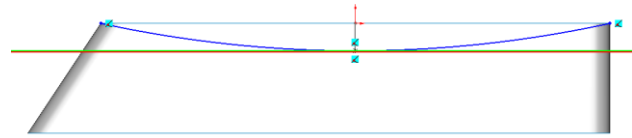




## Creating the other Constraint Curve

Create a sketch on the **Right plane**  
Choose **Right View**

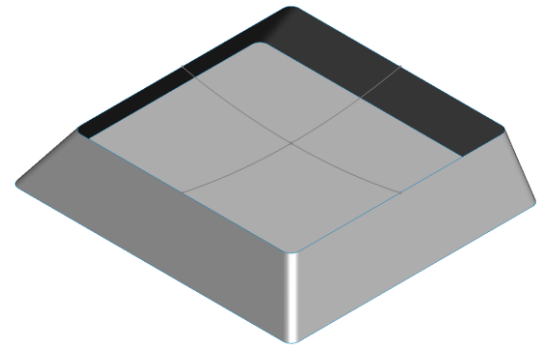
Using a similar procedure as before,  
sketch the spline shown opposite.



Exit the sketch and hide the plane shown.

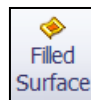
Choose Isometric View.

We now have the necessary geometry  
to create the filled surface.



## Filled Surface

Choose **Filled Surface** from the surfaces toolbar.



Select the top edge of the model as the **Patch Boundaries**

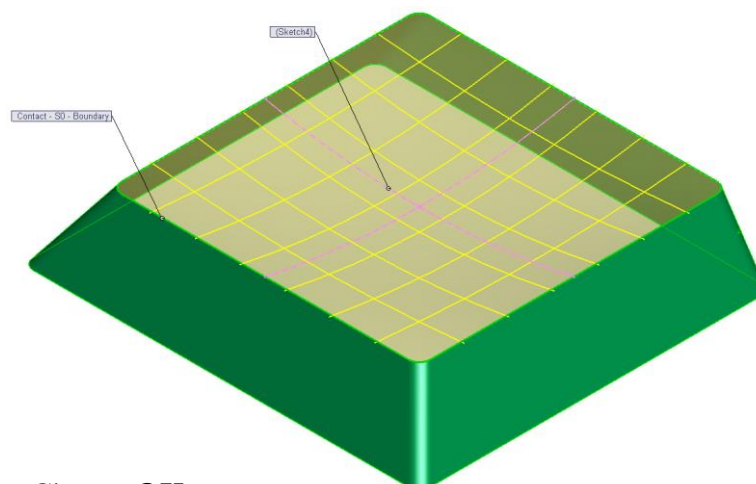
Choosing **Optimize surface** will ensure that SolidWorks  
tries to fit the patch surface within the limits of the  
boundary.

Select **Show Preview** and **Preview mesh**

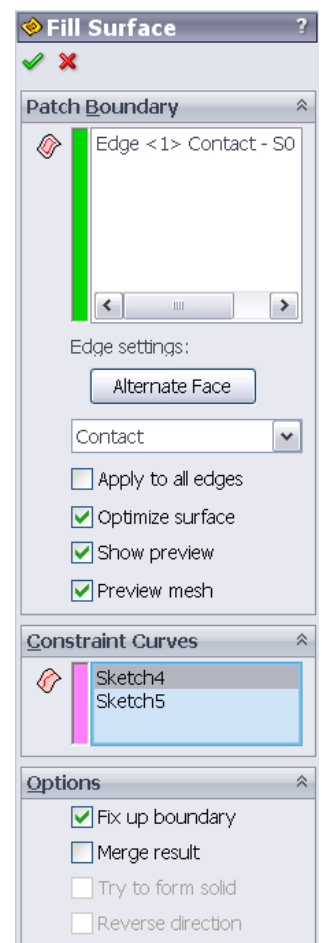
Select the two sketches as **Constraint Curves**

We will choose not to **merge result** in this case.  
Therefore the surface produced may be chosen  
independently of the others.

A preview of the surface is displayed in the graphics area.



Choose **OK**






## Fillet Surface

Surface fillet works in a very similar manner to that of Solid Fillet.

To apply a fillet to the top edge of the button;

Choose Fillet  from the Surface toolbar.


Select the **Manual** tab. Choose **face fillet**

Insert a **Fillet radius** of **0.3mm**

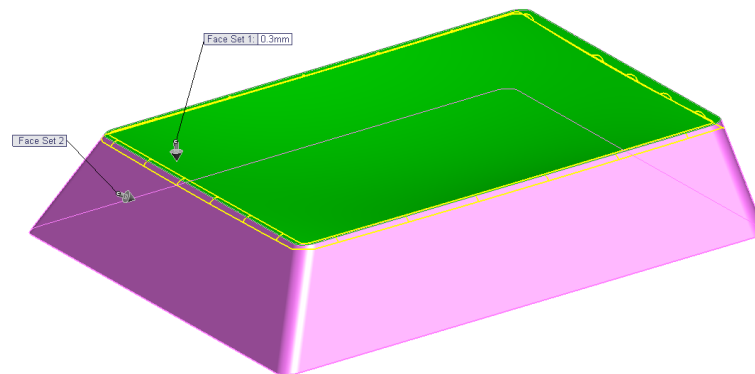
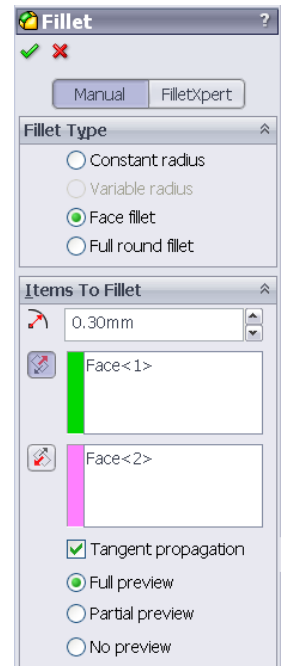
## Select surfaces

Select the top face as **face set 1** and the side as **face set 2**

Ensure that the direction arrows are pointing inward.

These may be toggled by choosing **reverse face normal** 

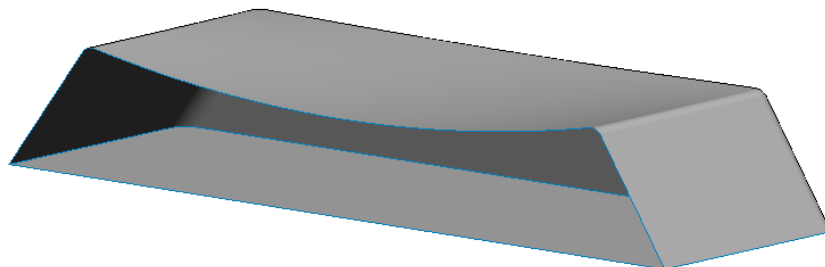
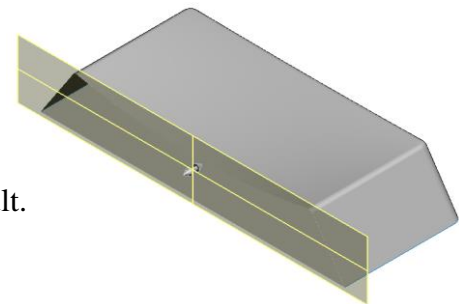
Check **Full preview**. Choose **OK** 



## Section


Choose **Section**  from the heads-up toolbar.

A section is created using the front plane by default.  
Choose **OK**.



## Examine the model

You will notice that the model is completely hollow inside and is enclosed by the surfaces created.

Deselect Section View  to return to a view of the complete model.

## Creating a Solid

As discussed previously, the aim of surface modelling is commonly to create a solid. A solid model may be generated from a surface model using **Knit Surface**

### About Knit Surface


**Knit** joins multiple surface bodies into a single surface body. It also has the option to create a solid if the resultant surface body satisfies the requirements; *a fully enclosed volume without gaps or overlaps*. Surface bodies must intersect edge to edge.

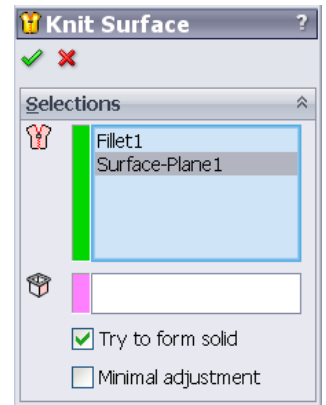
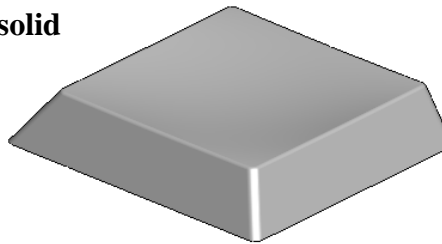
### Knit Surface

Choose **Knit Surface**  from the surfaces toolbar.


Choose the fillet and base surface as **Surfaces and Faces to Knit**

Check **Try to form solid**

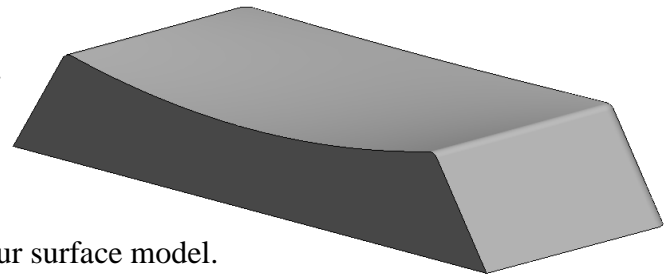
Choose **OK**. 




### Section

**Section**  the model as before.

You will notice that the model is now completely solid.



**Knit** has created a solid from our surface model.

Deselect Section View  to return to a view of the complete model.

### New surfaces

We are now going to create two new surfaces. These surfaces will be created as copies of existing surfaces. We are going to use **Offset Surface** to create these new surfaces.

### Offset Surface

**Offset Surface** does in 3D what Offset Sketch does in 2D. When a surface is offset a new surface is generated a distance of the offset away from the original surface.

Similar to Sketch Offset, an offset surface will fail if it is offset in the direction of decreasing radius and the offset is greater than the smallest radius of curvature of the surface.

One way of troubleshooting a failing offset surface is to use **Tools, Check...** to find the minimum radius.

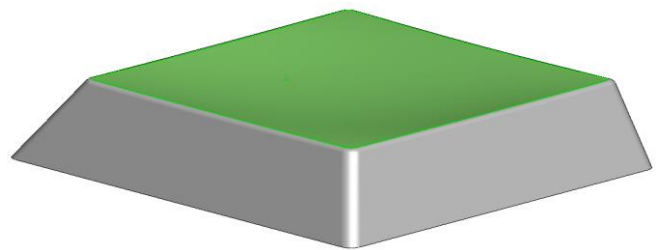
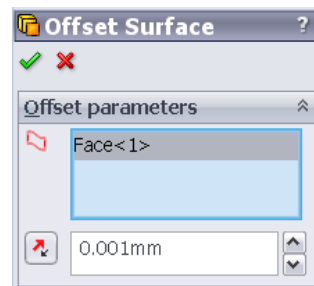
A copy of a surface will be generated if it is offset by a distance of zero.

To continue, we will now offset both the top and bottom surface of the model.

## Offset Surface

Choose **Offset Surface**  from the surfaces toolbar.

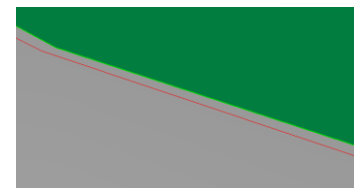
Select the top face as **Surfaces or Faces to Offset** with an offset distance of **0.001mm**.



## Zoom

Zoom in to ensure that the surface is being offset **above** the existing surface.

If below, choose **Flip Offset Direction**  to reverse it.

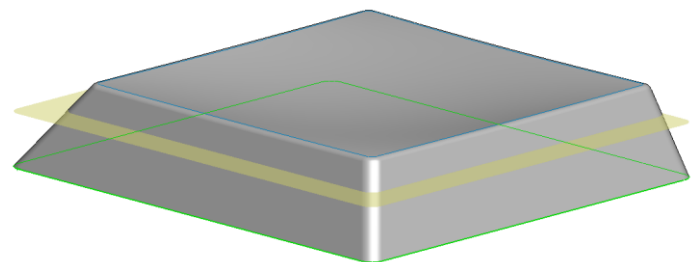
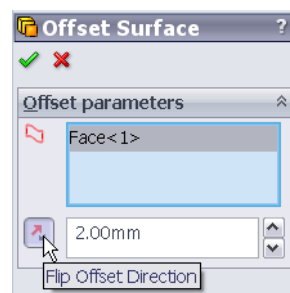


This will create a copy of the top surface, offset a distance of 0.001mm above it. This surface will be used when applying the letter 'J' to the button.

## Offset Surface 2

We will now repeat this procedure to create an offset surface from the **base surface** at a distance of **2mm**. Choose **Flip Offset Direction** if required.

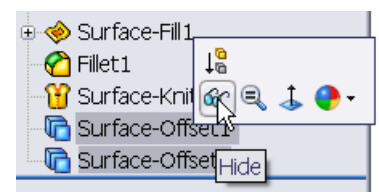
Choose **OK** .



This surface will be used later to replace a surface created when we shell the button.


## Hide Surfaces

Choose **Surface Offset1** from the feature manager design tree. Hold Ctrl and select **Surface Offset2**. Release **Ctrl** and choose **Hide**.



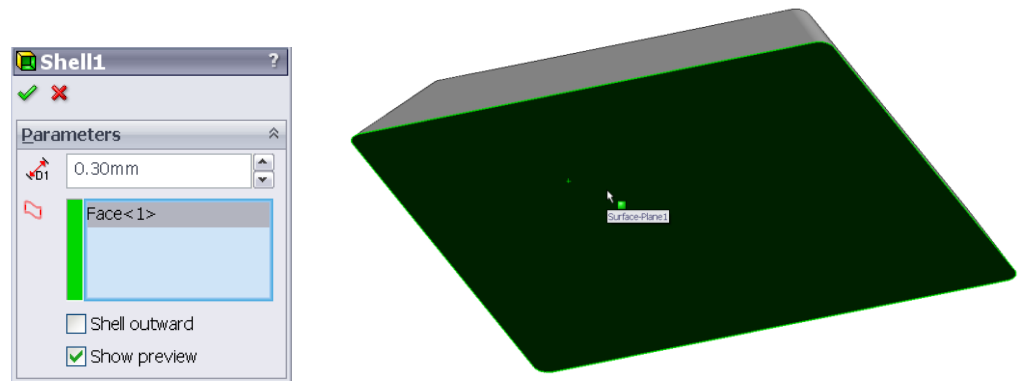
The two surfaces will be hidden.

## Shell


Choose **Shell**  from the **feature** toolbar.

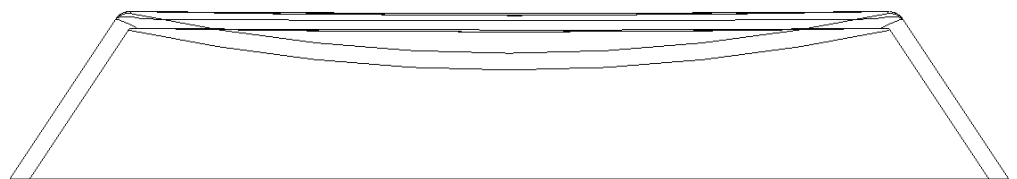
Insert a **distance** of **0.3mm**. Rotate the model and choose the base surface as **faces to remove**.

Choose **Show preview**. Select **OK**



## Orientation

Choose a **front view** orientation and select **wireframe display**  from the heads-up toolbar.



The wireframe display shows the internal walls created by the shell feature.

Because the top surface is curved, the internal top surface is also curved. In reality this is not the case. The internal surface is planar.

We will now replace the internal curved surface with the offset surface created from the base. To do so we will use the **Replace Face** tool.

## About Replace Face

**Replace Face** replaces selected faces of a solid or surface body with a selected surface.

If this was to be done manually it would involve **deleting** existing faces of the solid, **extend** and **trim** the new faces and then **knit** the new faces together.

## Orientation and display

Choose **Shaded with edges** as display style.

Rotate the model to an orientation which will allow you to choose the curved surface on the inside.

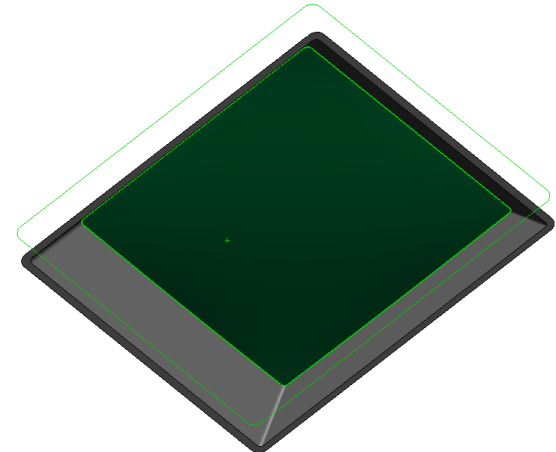
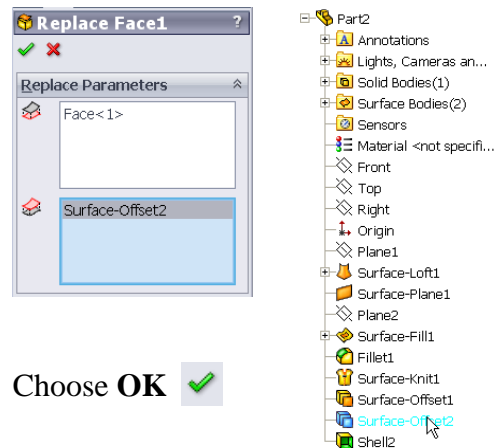


## Replace Face

Choose **Replace Face**  from the surfaces toolbar.


Select the curved surface of the model as **the target face for replacement**

Highlight the selection box for **Replacement Surfaces**. Expand the feature manager design tree in the graphics area and choose **Surface-Offset2**

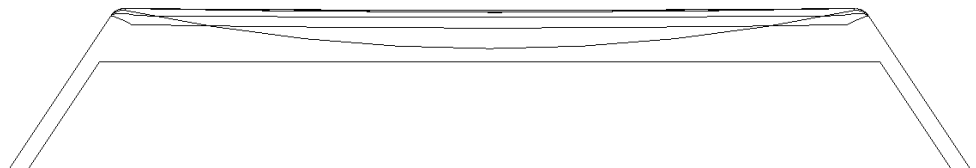


Choose **OK** 

## Orientation

Choose a **front view** orientation and select **wireframe display**  from the heads-up toolbar.

The top surface inside is no longer curved.  
It has been replaced by a flat planar surface.



## Orientation and display

Choose **Shaded with edges** as display style. Select an **Isometric View**.

## Adding text

The final stage in completing the exercise is to add the letter 'J' to the curved surface of the button.

In order to do so we must first create a sketch, of the letter, on the top plane.

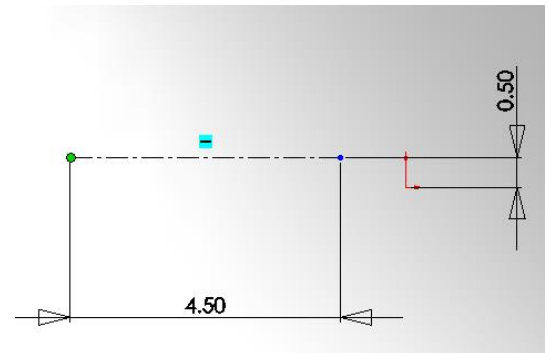
We will then extrude the letter between the offset face and the top face of the button.

## Creating the sketch

Create a sketch on the **Top plane**. Choose **Top View**.

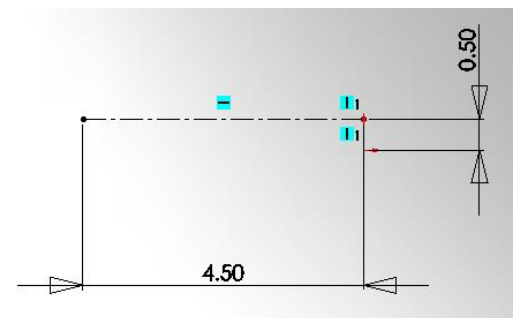
## Centreline

Sketch a horizontal **Centreline** and  
Smart dimension as shown.



## Add Relation

Add a **Vertical Relation** between the  
endpoint of the line and the origin.



## Text

Choose **Text**  from the sketch toolbar.

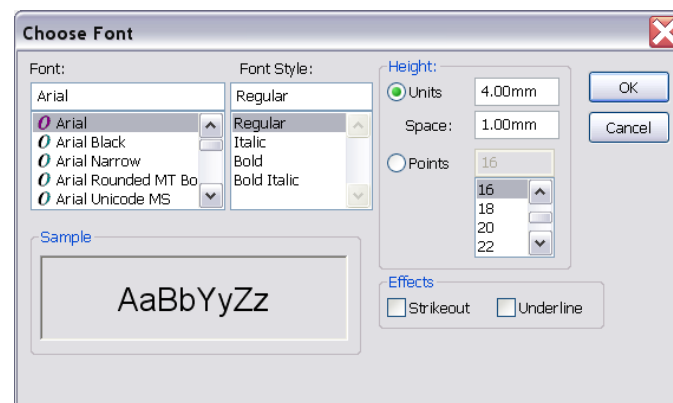
Select the centreline sketch as the guide.

Enter 'J' as **TEXT**

Note the preview in the graphics area.  
You may have to use **Flip Vertical** or **Flip Horizontal**  
to orientate the text correctly.

Deselect **Use document Font** and select 

Choose the settings displayed below.



Choose **OK**. Choose **OK**  Exit the sketch

## Extrude

We are now going to create the extruded feature from the text sketch.

## Orientation and display

Select an **Isometric View**.

**Note:** The sketch is contained on the Top Plane. Underneath the top plane we have the top surface of the button and also Surface-Offset1 which is 0.001mm above it.

When we generate the extruded feature we want it to start on the offset surface and finish on the top surface of the part, even though the sketch is contained on the top plane. i.e. The extruded feature will be 0.001mm high

## Extrude Text

Choose **Extruded Boss/Base** from the feature toolbar.

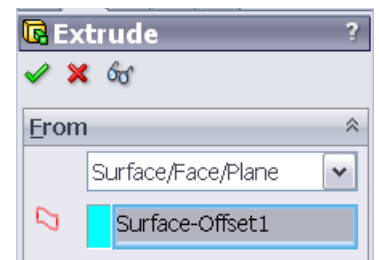
Select the sketch containing the text.

**Tip** – It may be easier to select the centreline, contained within the sketch rather than the text itself.

Ensure that the extruded boss is downward. To do so toggle **Reverse direction** 

To start the extrude on the offset surface choose **Surface/Face/Plane** as the **Starting Condition**

Select **Surface-Offset1**, as the starting surface, from the featuremanager design tree.



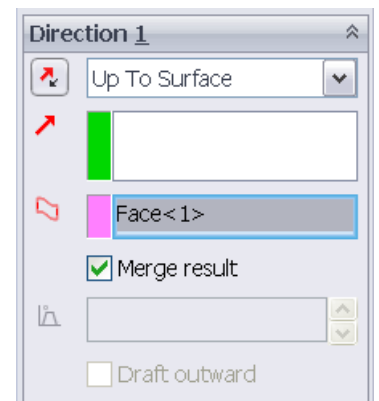
To identify the end condition;

Choose **Up to Surface** from the drop-down menu

Select the top face of the model as **Face/Plane**

**Tip** – you may pick the top surface of the model directly from the model as the offset surface, which is located above it, is hidden.

Choose **Merge result**. Choose **OK** 



The letter is now displayed as shown below.

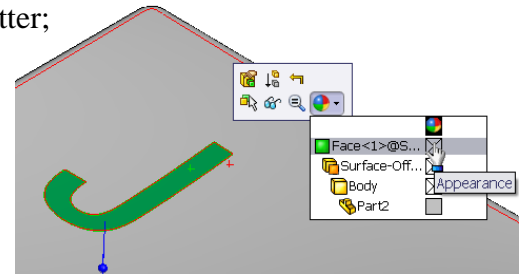




### Appearance settings

To change the appearance colour of the letter;  
left click on the face of the letter and  
choose **Appearance, Face...**

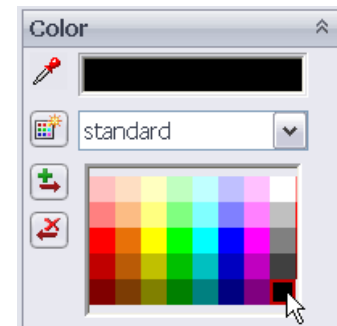
**Tip** – Choosing face will only apply  
the colour change to that face



### Face Colour

Choose Black from the colour palette on  
the left hand side.

Choose **OK** 

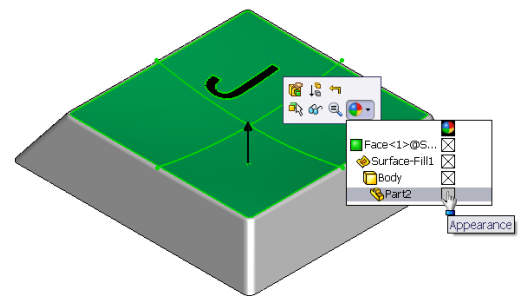


### Part Colour

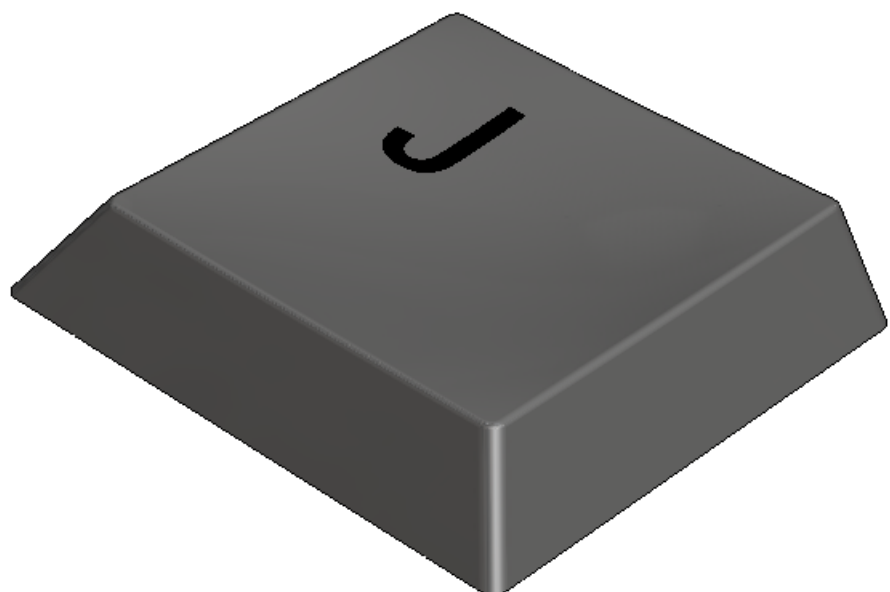
To change the part colour;  
Right click on any face and choose  
**Appearance, Part...**

Choose a colour from the palette.

Choose **OK** 



**Note** – The colour of the letter will not change because face colour overrides part colour



**Lesson Complete!**