

## Plastic Medicine Spoon



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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 even 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; *Filled Surface*, *Surface Thicken* and *Cut with Surface* as well as *Shell* and *Extrude* feature tools.

## Getting started.

### New File

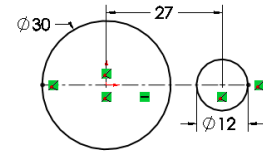
Create a new part file and save it as **Plastic Medicine Spoon** in the desired location.

### New Sketch

We are going to begin by creating a sketch to represent the top profile of the spoon. Create a sketch on the Top plane.

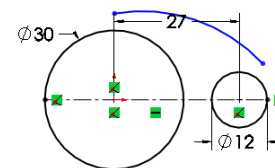
### Steps required.

Sketch the centreline as shown.



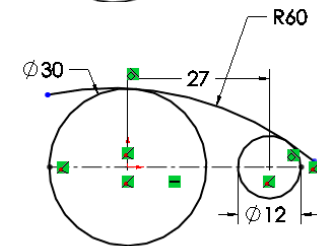
Sketch a circle of diameter 30mm with its centre coincident with the origin. Sketch a diameter 12mm with its centre Coincident with the centreline and dimension as shown.

Create the **centrepoint** arc shown.

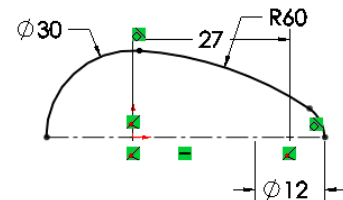


Smart dimension 60mm

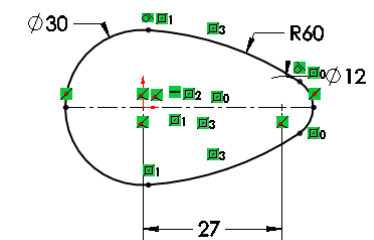
Apply a **tangent relation** between the arc and the two circles.



**Trim** the sketch using **Power Trim**.



Using **Mirror** copy the sketch across the centreline.



We have now created the profile for the outline of the spoon.

Confirm the sketch.  Rename the sketch **top profile**

### Sketch 2

Having created the top profile we will now sketch the front profile of the spoon. Create a new sketch on the front plane.

### Steps required.

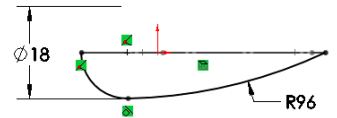
We need to use the centreline from sketch 1 in sketch 2. To do this we use the **convert entities** command. Select the centreline, choose **convert entities**. The centreline becomes an entity within the current sketch and appears as a line. Use line properties to convert it back to 'for construction'. It will now appear as a chain line within the current sketch



The centreline can now be used to create the relations required to define the sketch.

Using circle and centrepoint arc create the following sketch.

Smart dimension the sketch and apply the relations shown.



Now we have the profile for the curvature of the spoon when viewed from the front

Confirm the sketch.  Rename the sketch **front profile**

## Filled Surface

Filled surface enables you to create a surface or 'Patch' defined by edges, lines or curves.

### Steps required.

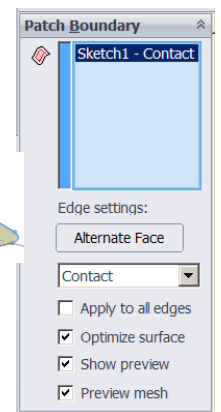
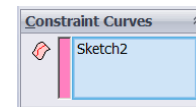
Select the filled surface tool from the surfaces toolbar.



Select **top profile** as the patch boundary. This is the outline of the spoon.

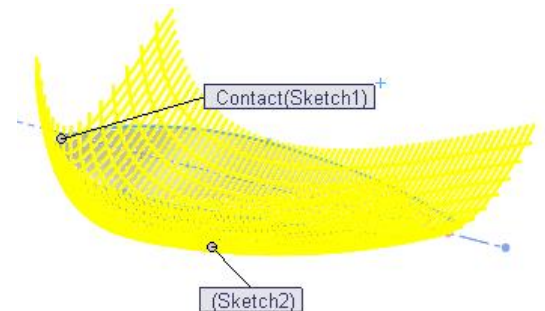



Check **Optimize surface** to ensure that the patch or surface will finish at this outline.



A planar surface is now created within the boundary of this sketch.

Select **front profile** as the constraint curve. This curve will direct the surface to 'bend' to give the curvature required to create the surface.

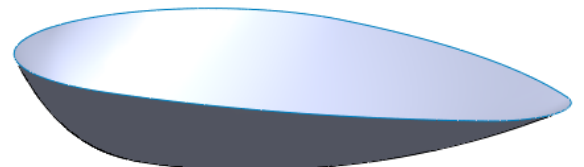


Select OK 

The surface is now created!

## Rename the feature


Rename the feature as **Spoon**.



## From Surface To Solid

When a surface is created in SolidWorks it has no thickness. If we take a section view through the surface we can see this.



To give the spoon wall a thickness we use the **Thicken** command  from the surfaces

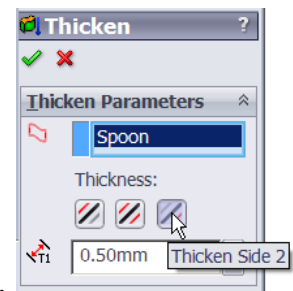
toolbar.

Select **Spoon** as the surface to thicken.


Set the thickness to 0.5mm.

We have three options to choose from in order to thicken the surface.

Selecting 'Thicken side 1' will thicken to the outside, 'Thicken both sides' will thicken either side of the surface, 'Thicken side 2' will thicken to the inside.



Choose **Thicken Side 2**

Select OK. 

### Trimming back Surface thicken.

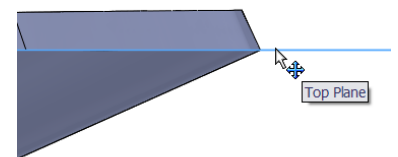
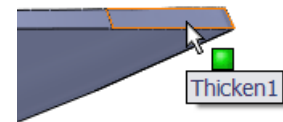
You will notice that the surface thicken gives a bevelled finish to the edge of the spoon.


To correct this we will use **Cut with surface** to trim the excess off the top edge.

**Cut with surface** removes unwanted material by cutting with a surface or a plane.

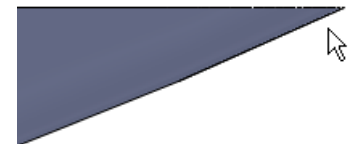
As the **Top Plane** will conveniently cut the solid in this case it will be used as the cutting plane.

Reverse direction if required.



Select OK. 

The edge is now planar.



### Creating the Handle

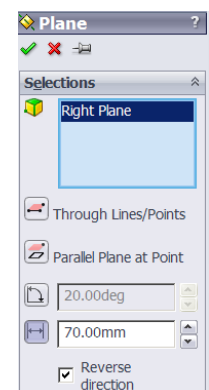
To create the handle we must first set up a plane parallel to the **Right Plane** and create a sketch on that.

From the **Surfaces** menu select **reference geometry, plane**.

Select the **Right Plane** as a reference entity.

Check **Reverse direction**.  
Set the **distance** to **70mm**.


Select OK 

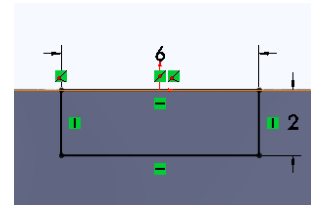


### Sketch

Sketch a **corner rectangle** on **Plane1**.


Apply a coincident relation between the midpoint of the top line of the rectangle and the origin as shown.

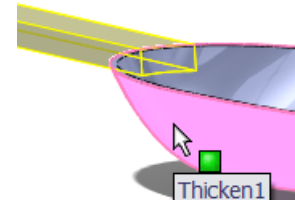
Confirm the sketch. 



### Extruding the Profile

Select **Extruded Boss/Base** from the features menu. Select **Up to Surface** as the **end condition**. Select the outer surface of the spoon as the **face/plane** to extrude to.

Select OK 



### Rename the feature

Rename the feature as **'Handle'**.


### Hide Plane 1

Click on **Plane 1** and select **hide**.

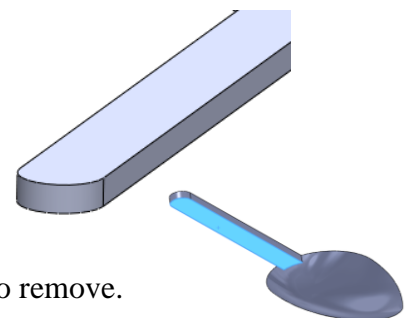
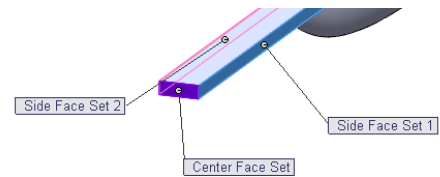


### Fillet

Apply a **Full round fillet** to the sides of the handle. Select the faces shown.

Select OK 


The end of the handle is now rounded shown.

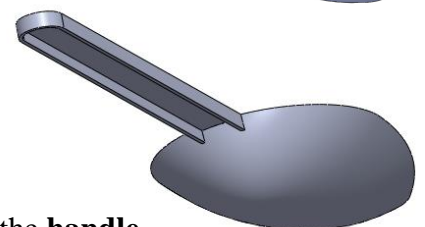


### Shell Feature

Select **Shell** from the **Features** menu. Select the underside of the **Handle** as the faces to remove.


Set the thickness to 0.5mm.

Select OK 

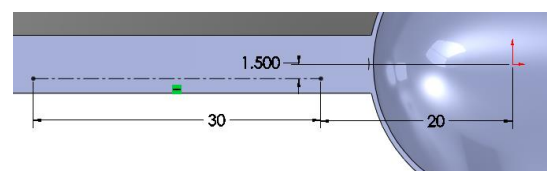


### Creating text Feature

Create the following sketch on the top surface of the **handle**.

Select **Text** from the **sketch** menu. 

Make the following selections:

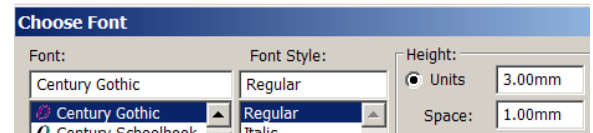
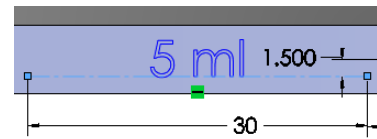


Select the **centreline** as the line for the text to follow.

Type in **5 ml** as the text.

Select **Century Gothic** as font, set the units to 3mm and centre align the text.


Select OK 



### Extrude Text

Select **Extrude** from the **Features** manager

Set the thickness to **0.25mm**.

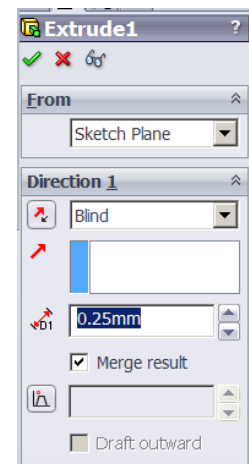
Select OK 

### Rename Feature

Rename the feature as **Text**.

### Select Material

Set the material as **PTFE**.



**Exercise complete!**