



Professional Development | An tSeirbhís um Fhorbairt
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Intermediate Solidworks



Spatula



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Introduction

This workshop aims to examine some of the intermediate level sketching tools and features used on SolidWorks 2017.

The workshop will work through the modelling of a series of part and their assembly into the completed spatula.



Learning Intentions

At the end of this workshop you should be able to:

- Understand best practice in the use the **sketching tools** – spline style spline, slots, sketch fillet and ellipse.
- Appreciate the use of **features** such as: mirror, boundary boss/base, variable fillet, sweep and indent.
- Explorer the use of **in-context modelling** in the creating of a part.
- Examine the use of **split line** in the creation of coplanar surfaces.
- **Assembly** a series of parts together to form an assembly of a spatula.
- Develop a better understanding of the creation of photorealistic images when using **Photoview 360**.



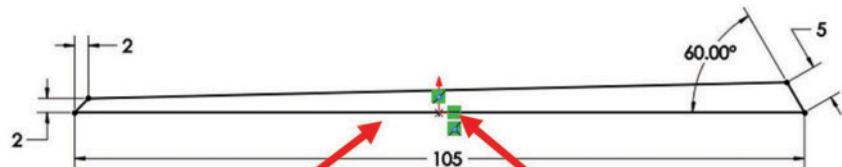
Spatula Blade

Create a New Part and save this part as **Base**.



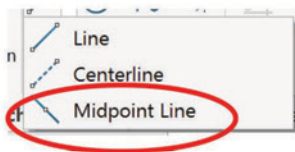
Create a sketch on the **Front Plane**

Sketch the given blade outline and dimension as shown.

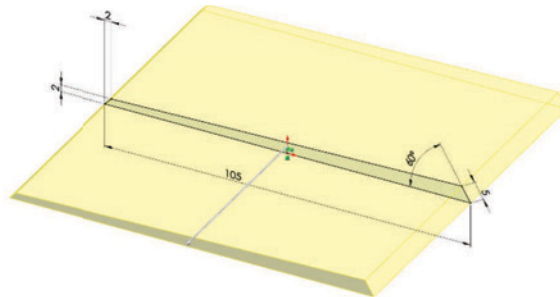


Use Midpoint Line to construct this line

Origin



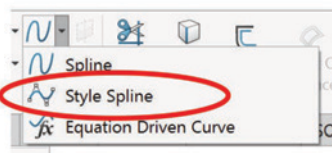
Extrude Boss/Base **105mm** using **midplane** end condition.



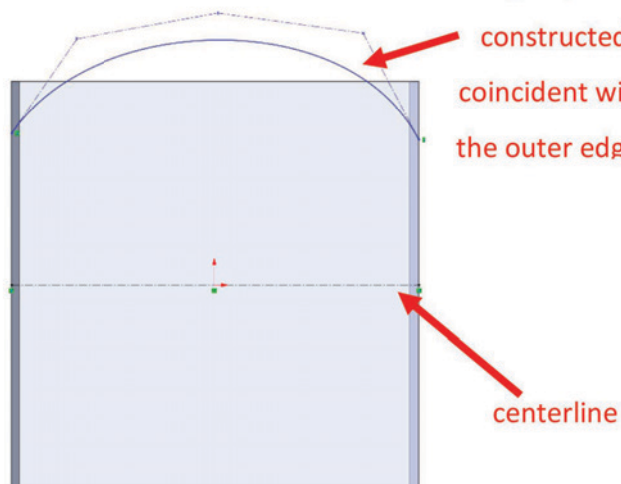
New sketch on **Top Plane**

Draw a **centerline** across the centre of the blade

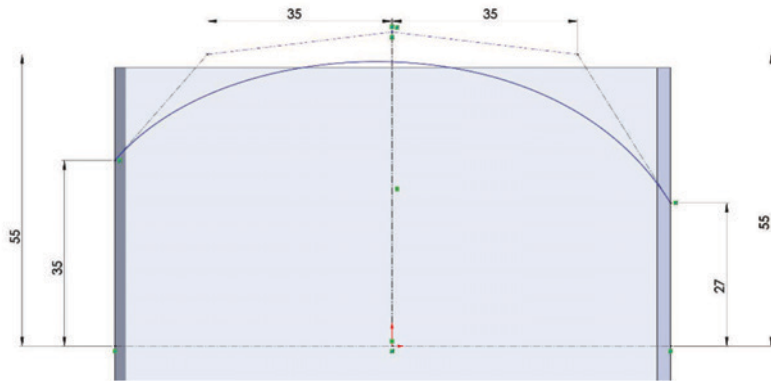
Style Spline constructed coincident with the outer edges



Select **Style Spline** and draw the curve using five nodes as shown.



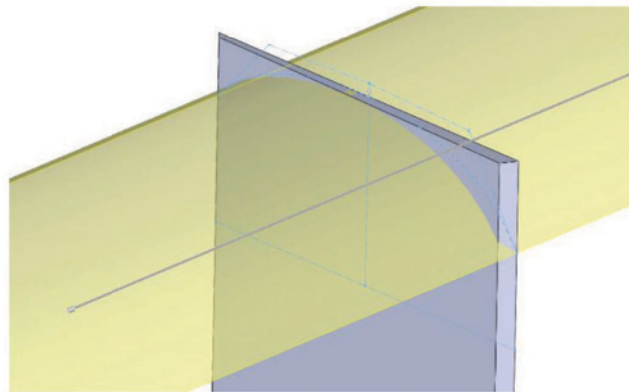
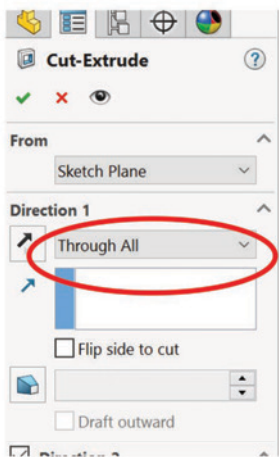
Using **Smart Dimension**, add the dimensions and centerline as shown. Add a **vertical** relation to the centerline.



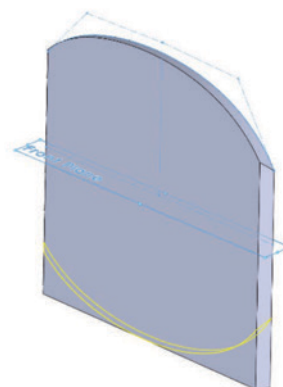
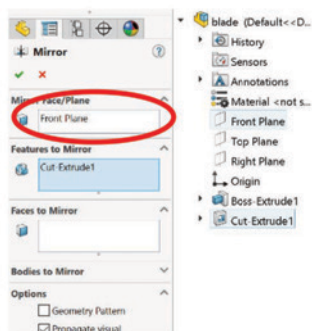
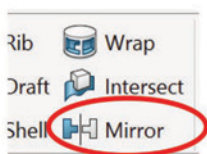
Add a **Tangent** relation between the top edge and spline curve.



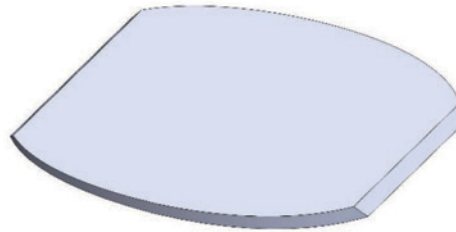
Extrude Cut, select **Through All**



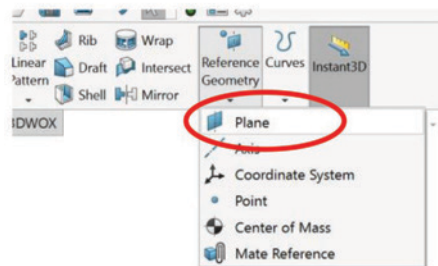
Mirror the extrude cut about the **Front plane**



Save Part

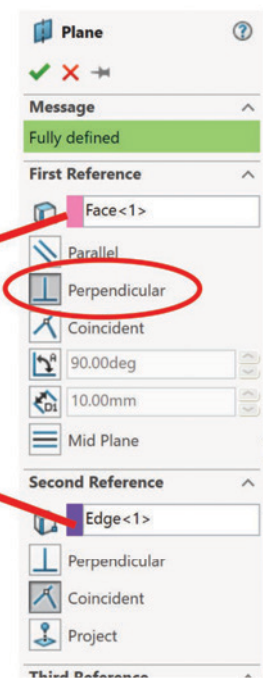
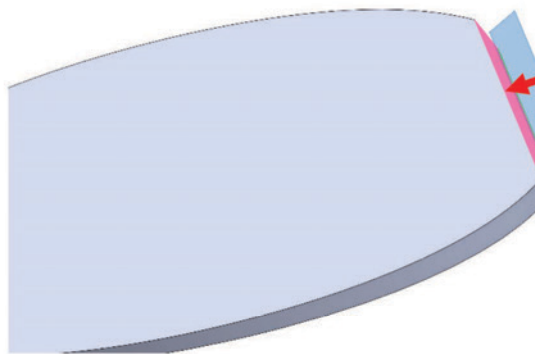


Insert a new Plane, which will be constructed perpendicular to the inclined surface at the rear of the blade.



Select **Reference Geometry** and **Plane**

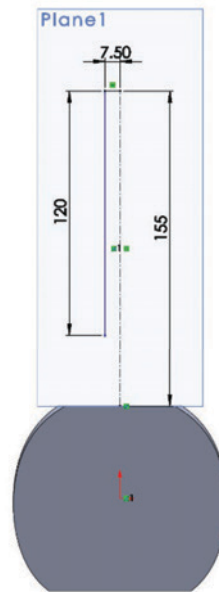
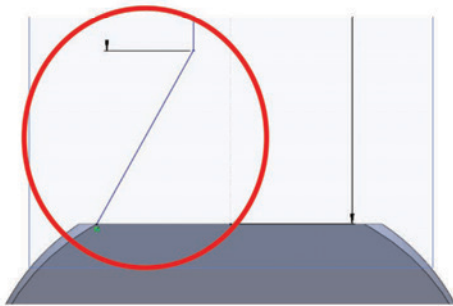
Select the face of the surface at the **first reference** and select **perpendicular**.



Select the **base edge** as the second reference to create the plane.

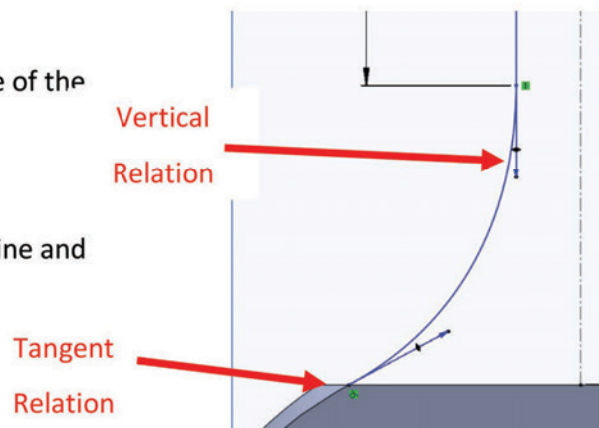
Construct a centerline, sketch the given sketch and smart dimension.

Draw **2 point spline** as shown below

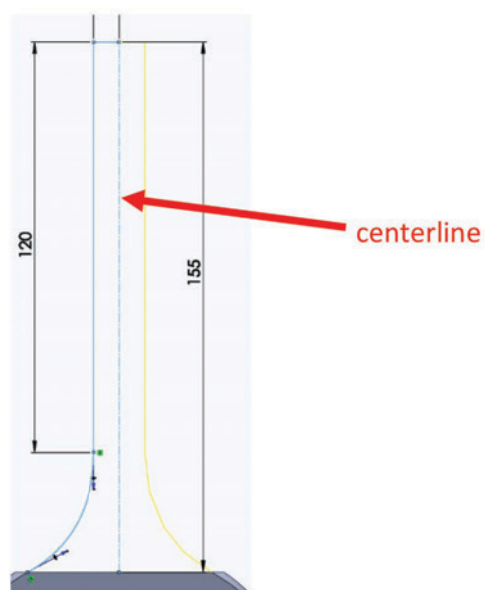


Add a **Vertical relation** to the top handle of the 2 point Spline.

Add a **Tangent relation** between the spline and the curve of the base.

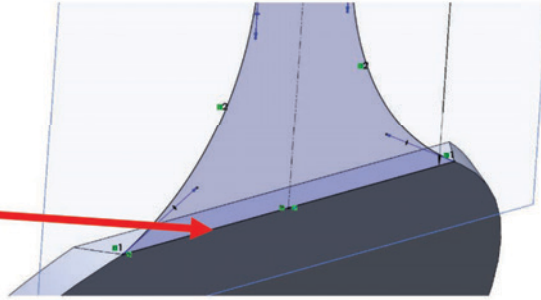


Mirror the sketch about the centerline.



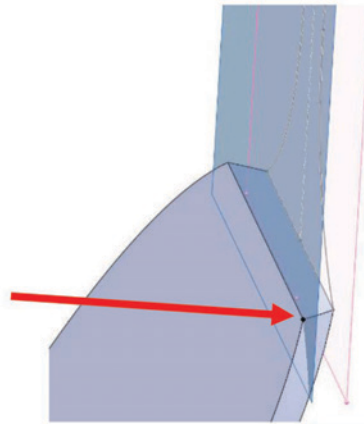
Convert Entities on the edge of the base to close the sketch

Convert this edge



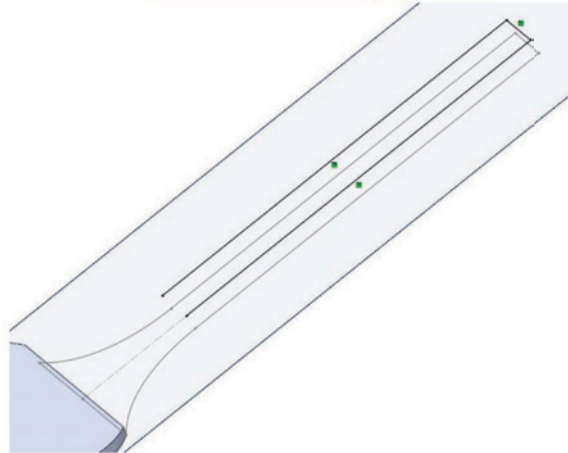
Insert a **Plane**, parallel to the initial plane created and through a point on the top edge.

Select point on top edge

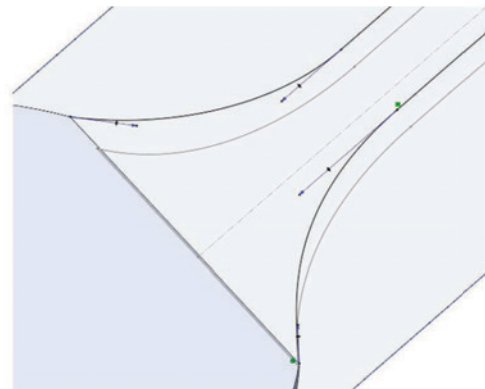
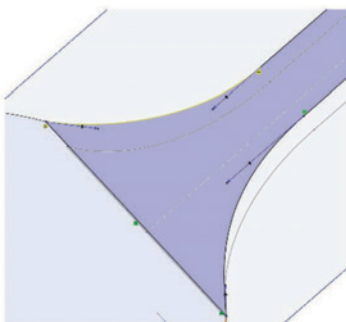


Create a **sketch** on this new plane

Use **Convert Entities**, to convert the straight lines of the handle from the previous sketch onto this.

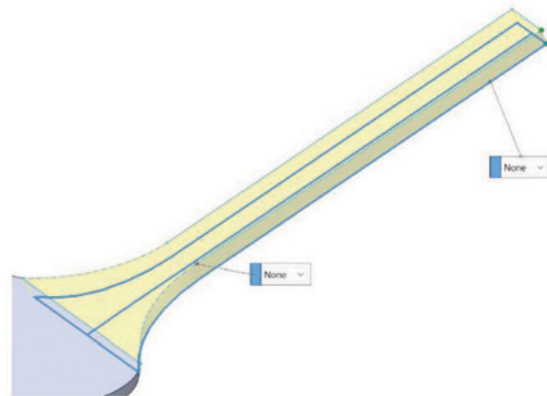


Construct the 2 point **Splines** on each side using the same construction as the previous sketch

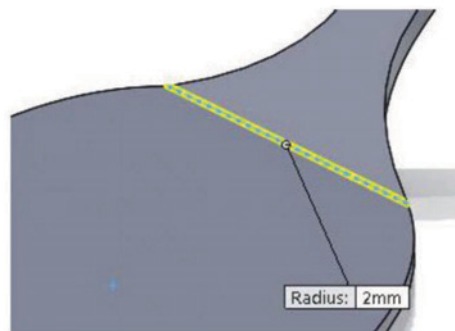


Convert Entities on the top edge to close the sketch

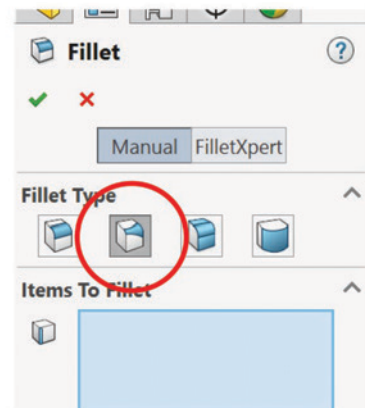
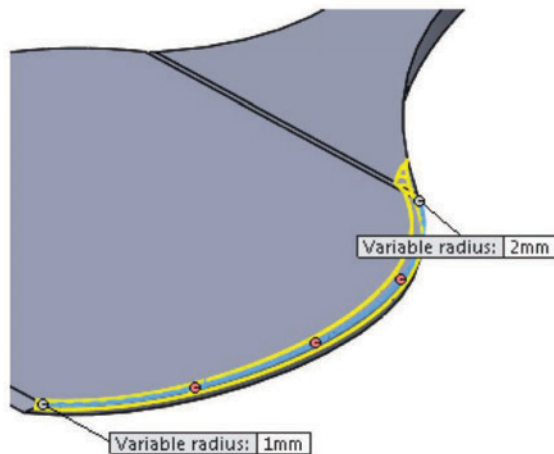
Select **Boundary Boss/base** and select the two sketches to complete the stem.



Add a **2mm** fillet as shown.

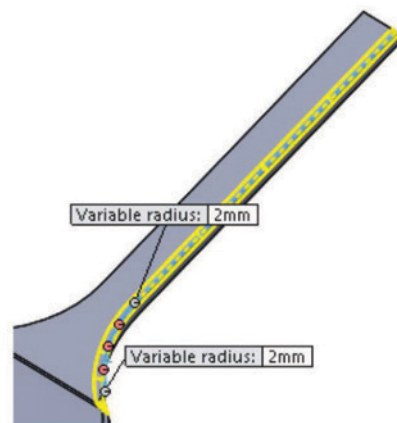


Use **Variable Fillet** command to apply fillets as shown.



Complete the exercise by applying **2mm** fillets to the stem.

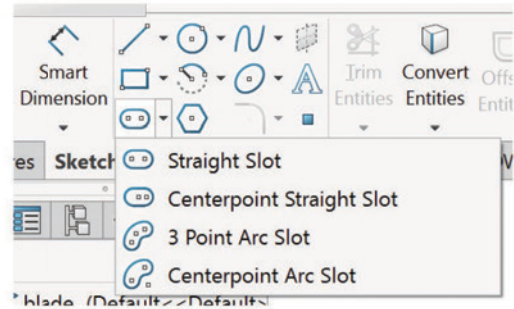
Complete the other side of the blade using the same dimensions.



Slots

The slot command saves a lot of time, as alternatively circles, lines and trim commands would need to be used to create the same shape.

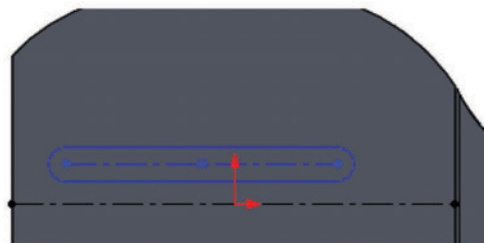
As can be seen in the diagram, various shapes of slots can be drawn. Slots have also been included in the mate commands, enabling easy mating between slots.



Create a **sketch** on the top surface of the spatula blade

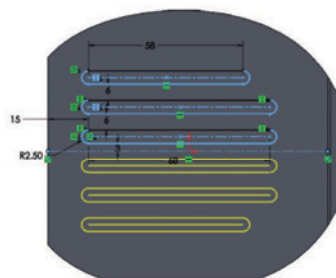
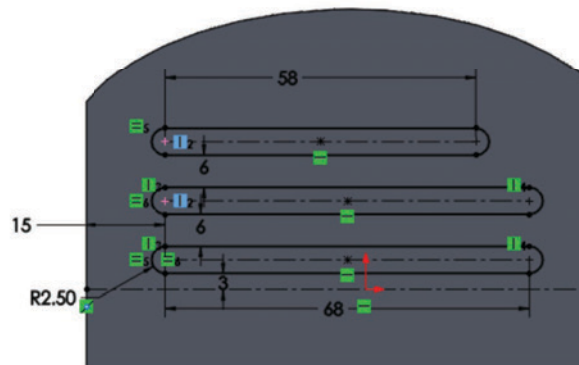
Draw **centerline** across the centre of the blade surface

Select **Straight Slot** and draw as shown



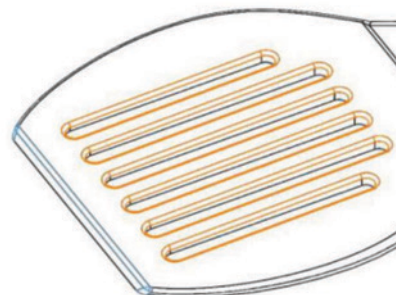
Add **Equal relations** for the radii and vertical relations for the centre point 1 of the three slots.

Add the following measurements and **mirror** about the centreline.

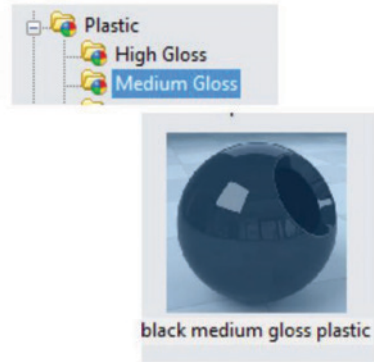


Select **Extrude Cut**, Through All.

Add a few more **2mm** fillets as shown.



Add an **Appearance** as shown



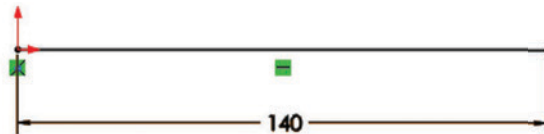
Handle

Create a New Part and save this part as **Handle**.

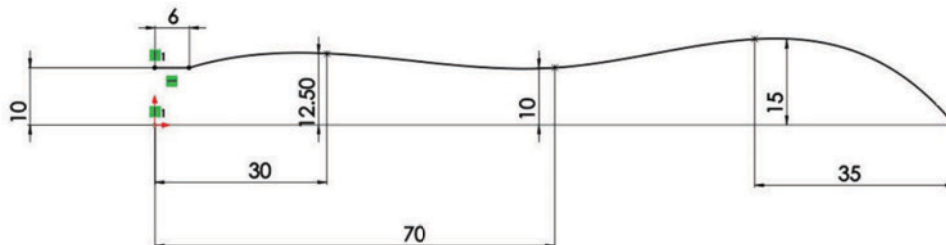
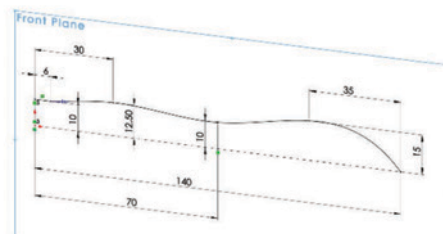


Three separate sketches are drawn to create the handle shape.

Sketch 1 - A line is drawn on the **Front Plane**.



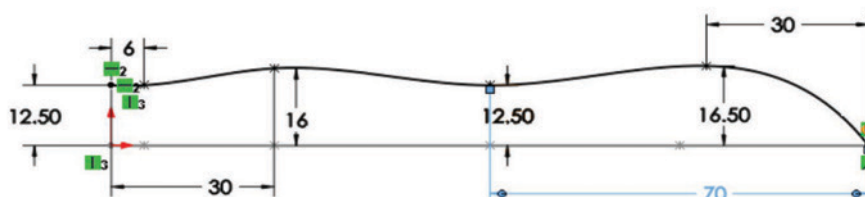
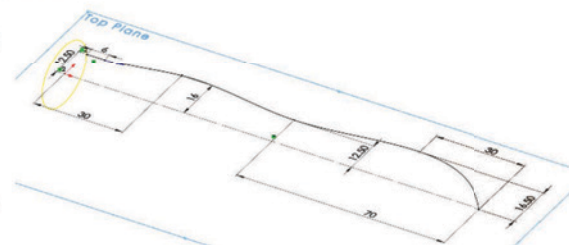
Sketch 2 - On the **Front Plane** using Line and **Spline** command, draw to the dimensions shown. A line **6mm** in length is drawn first.



Sketch 3 - On the **Top plane** draw a **Spline** with 5 points.

As above, a line **6mm** in length is drawn first with a **Vertical relation** between first point and Origin.

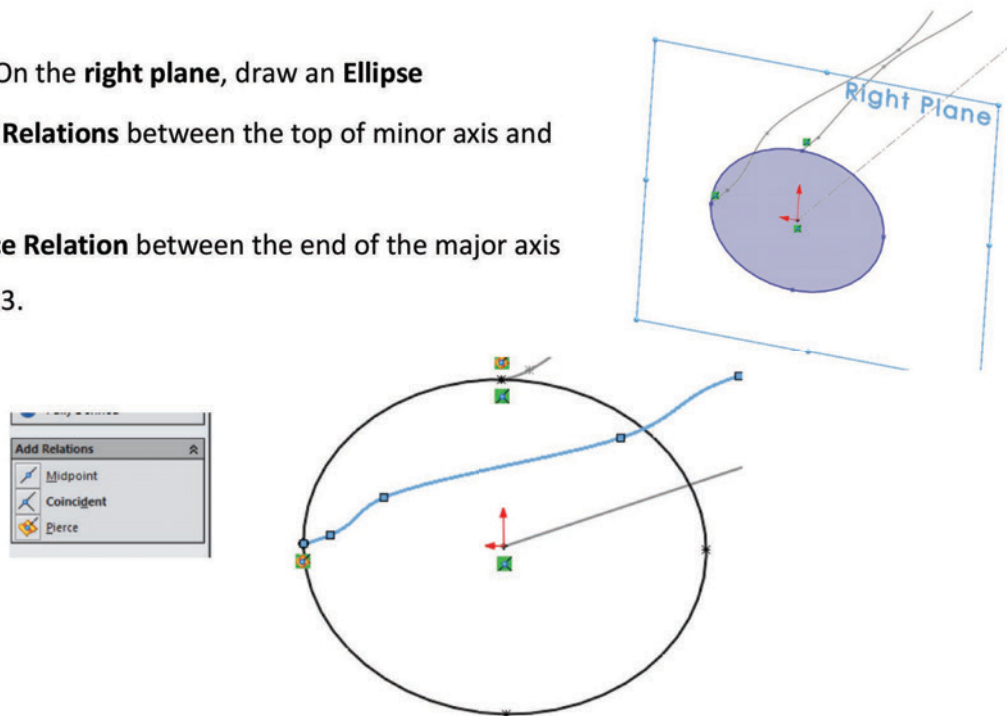
Add a **Pierce relation** between the last point of the spline and the line.



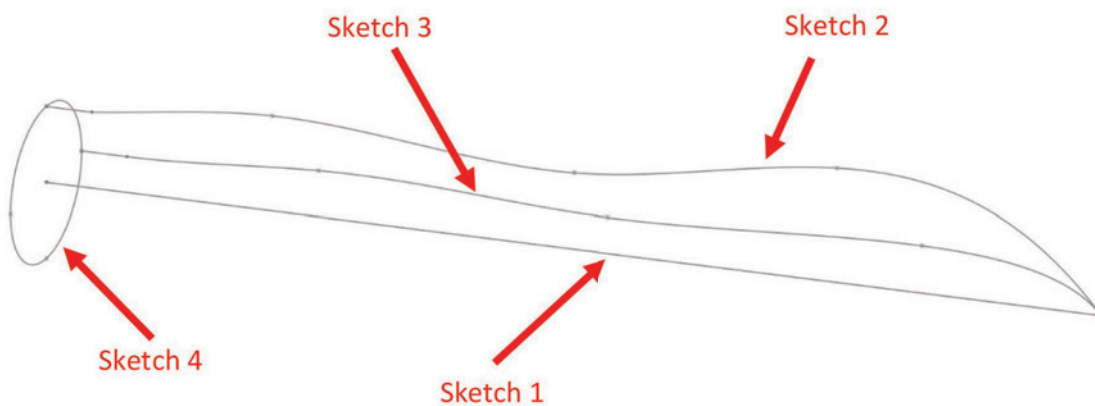
Sketch 4 - On the right plane, draw an Ellipse

Add **Pierce Relations** between the top of minor axis and sketch 2.

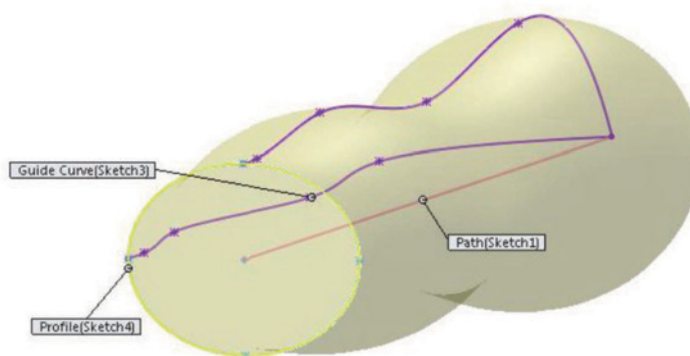
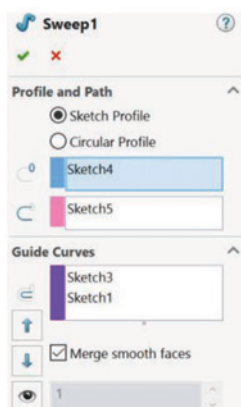
Add a **Pierce Relation** between the end of the major axis and sketch 3.



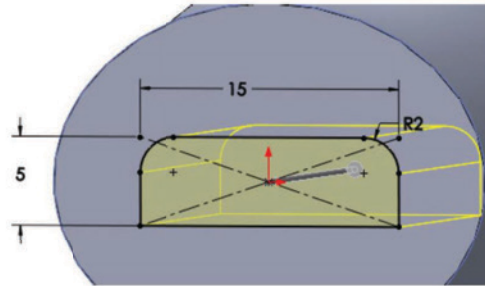
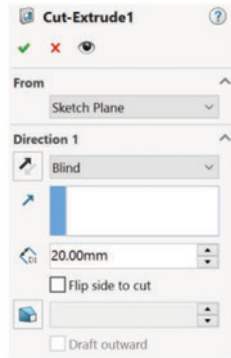
Sketches to complete the handle shape



Select **Sweep** command to form the shape.

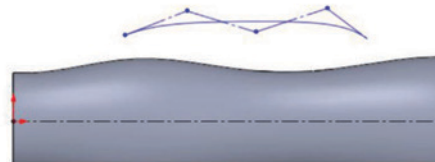


On the front face, draw the **Rectangle**, and **Extrude Cut** by 20mm. This will be the recess for the blade.

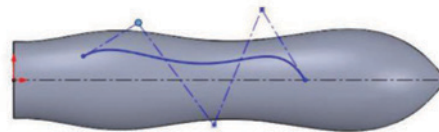


Adding decoration on the face of the handle.

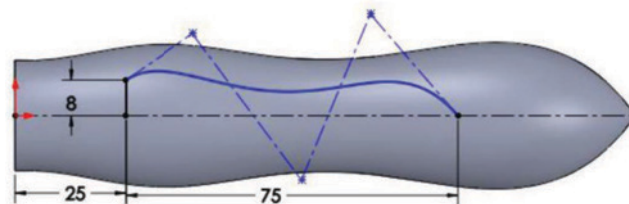
On the **Top Plane**, use **Style Spline** to draw the shape shown.



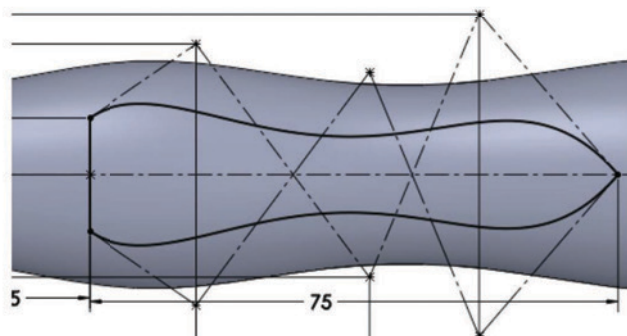
Drag the points into position and adjust the curve.



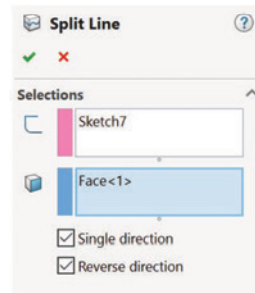
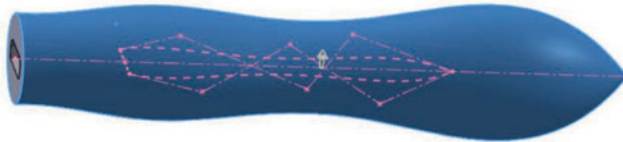
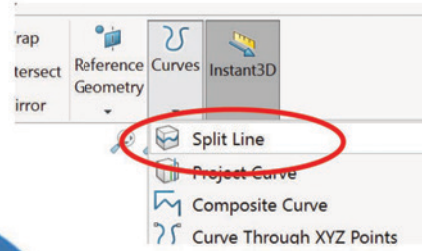
Add the shown dimensions



Mirror the sketch about the centreline and **Fully Define** the sketch.



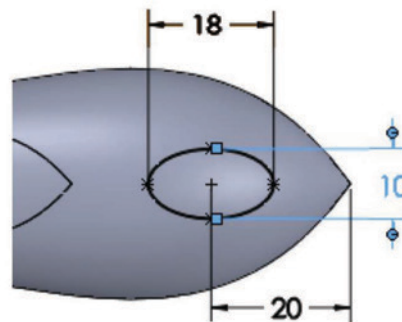
Select **Split Line**, and make sure single direction is selected.



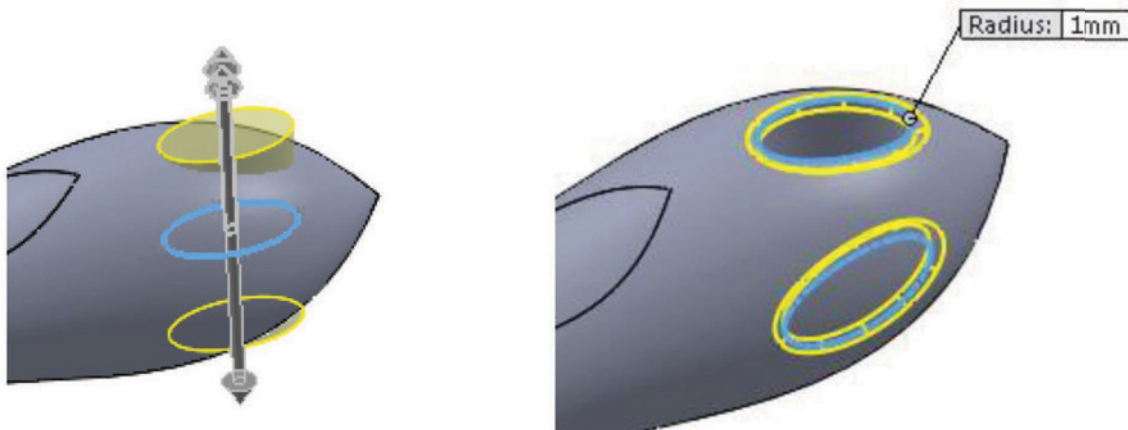
The Hole at the Back

On the **Top Plane**, draw the **Ellipse** shown.

Add relations to make the major axis **collinear** with the end point.

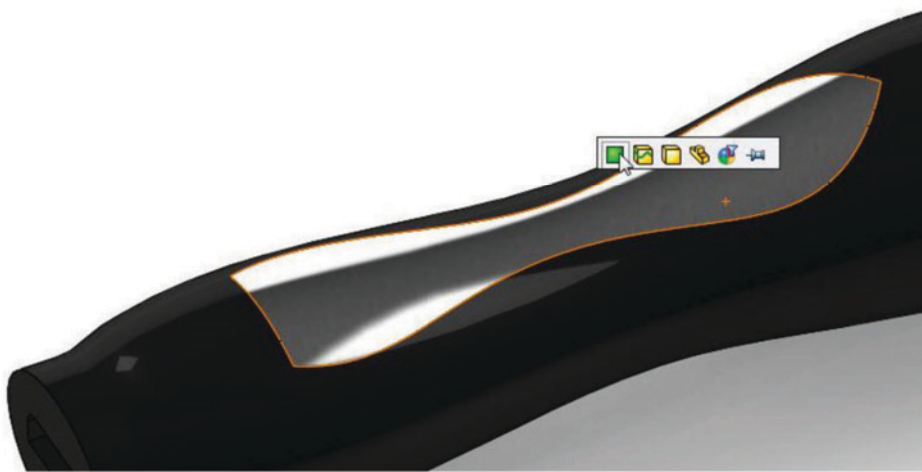
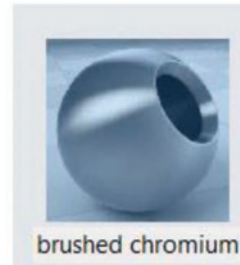
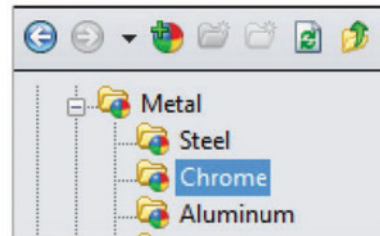


Extrude Cut in both directions, and add **1mm fillets**.



Appearance:

Apply a **Brushed Chrome** appearance to the face.

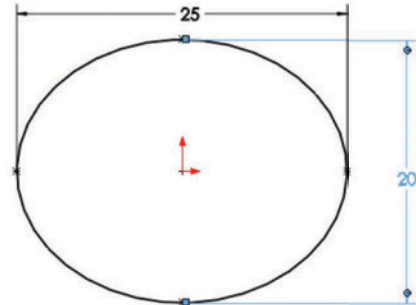


Washer

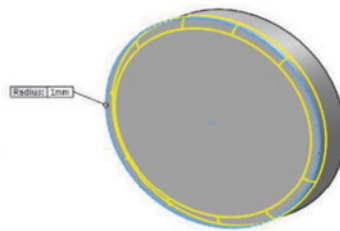
Create a New Part and save this part as **Washer**.



On **Front Plane**, draw an **Ellipse** to given dimensions.

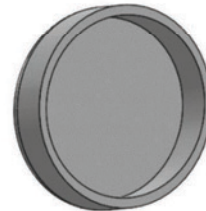


Extrude Boss/base by **4mm**.

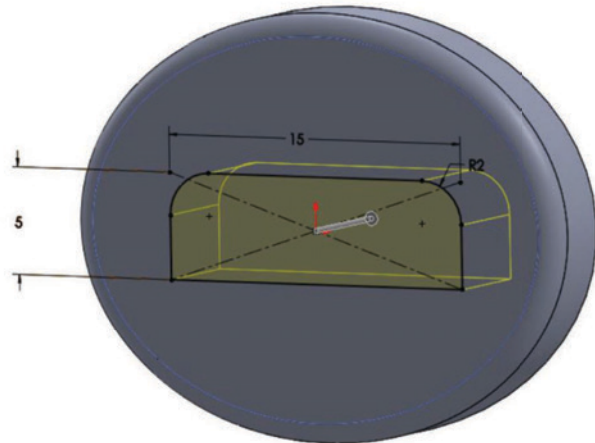


Add a **1mm Fillet** to front edge.

Shell out the back using a thickness of **1mm**.



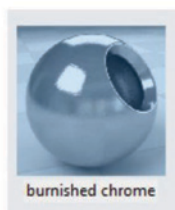
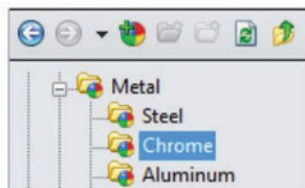
On front face, select **Centre Rectangle** and draw to the dimensions shown.




Using Fillet command, add a **2mm fillet** to the top corners of the rectangle.

Select **Extrude Cut** and through all.

Apply a **Brushed Chrome** appearance to the washer.



Fixing Washer to Handle

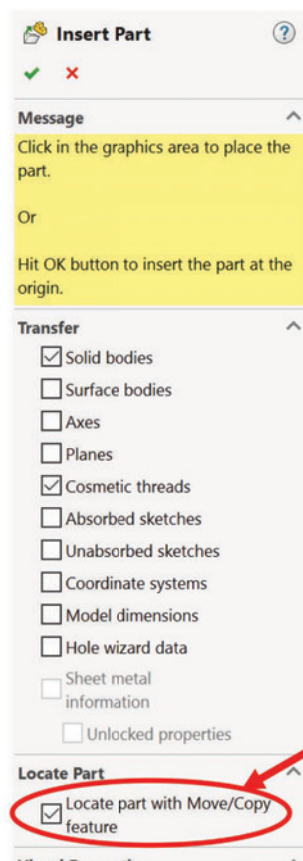
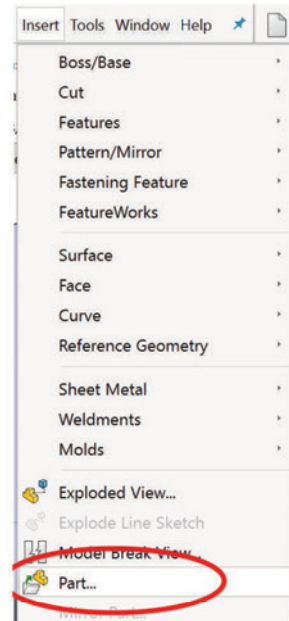
The **indent** feature  is a useful tool to ensure that the washer fits snugly onto the handle.

Open the **“Handle Part”**



Insert the **Washer** part into the Handle part by selecting **Insert** and **Part**.

Locate the washer part in the explorer window and insert.



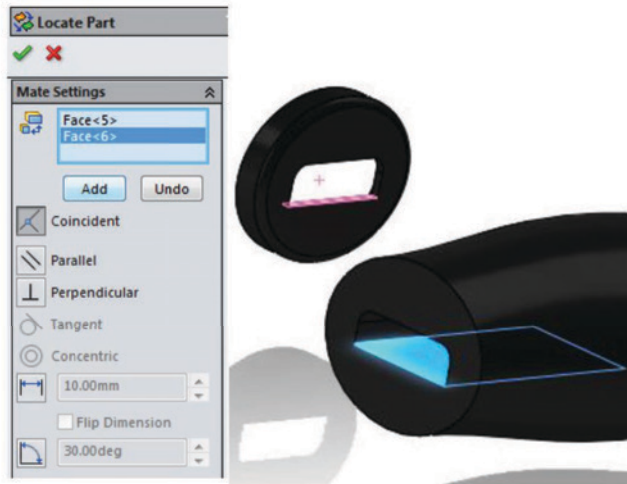
Ensure that that the **“Locate part with Move/copy feature”** option is selected so the washer can be located into position

Mates are required to position the washer correctly in position.

Select the bottom face of the hole of the handle, as shown in blue, and the bottom face of the hole in the washer, as shown in pink.

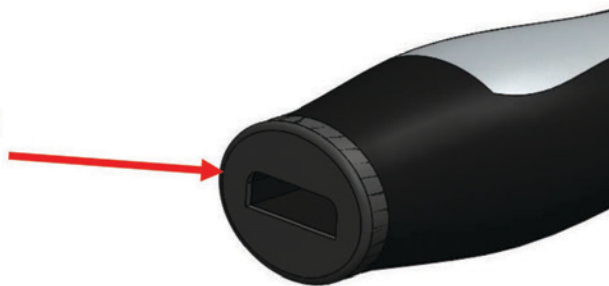
Select the **Add** key to complete the mate.

Select the front face of the handle and the back of the washer, and select **Add** to complete this mate also.

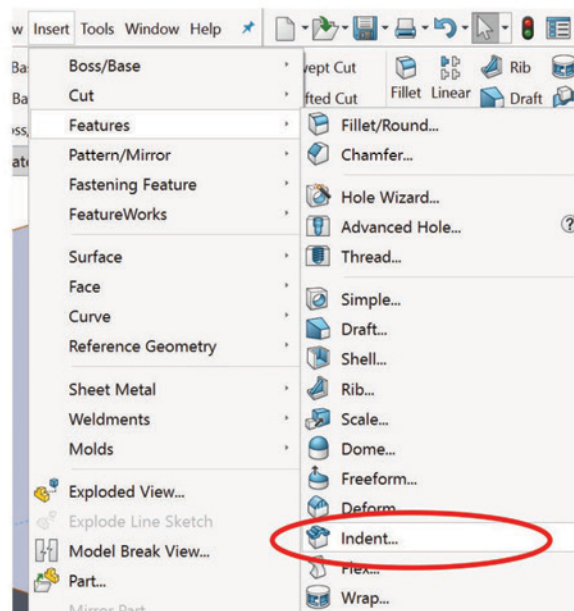


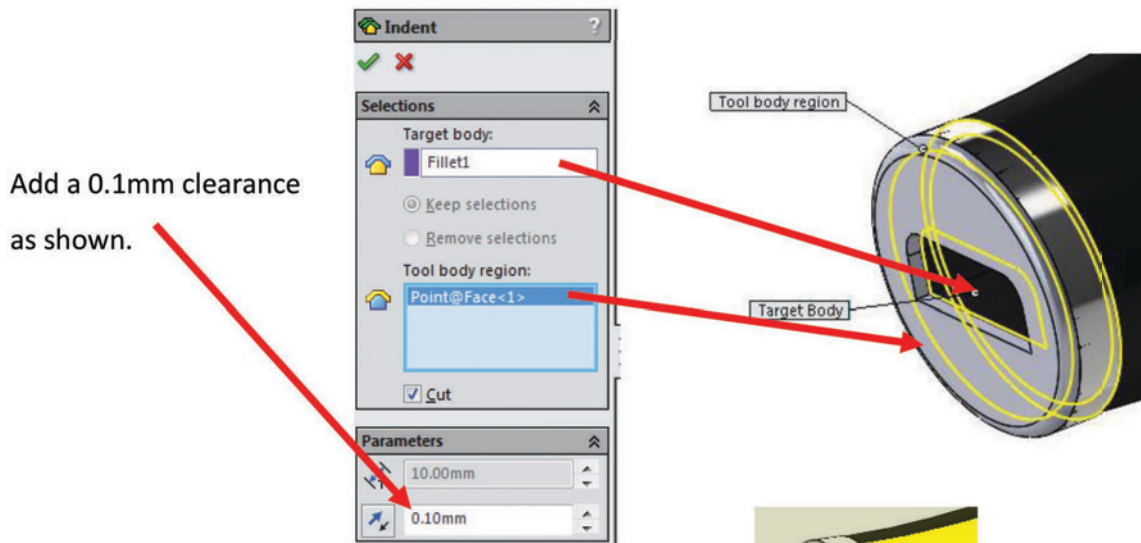
Finally select the side of the hole on the handle and the side of the hole on the washer, and **Add** mate to fix washer in position.

Washer part located onto the end of the handle

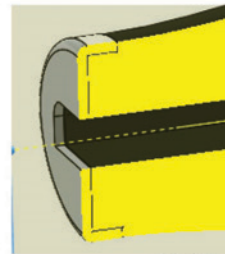


Indent the washer onto the handle, select Inset, Features and **Indent**





The result can be seen in the **section view** shown.



Assembly

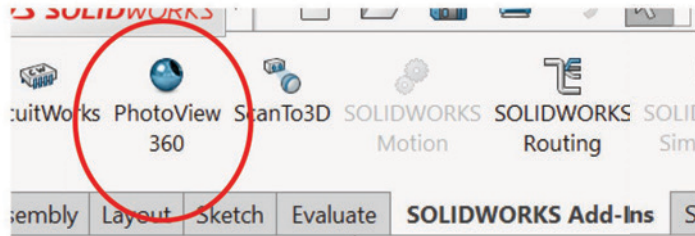
To assemble, bring in the **Blade** first, then the **Handle** and **Washer**.

Use mates to assemble.



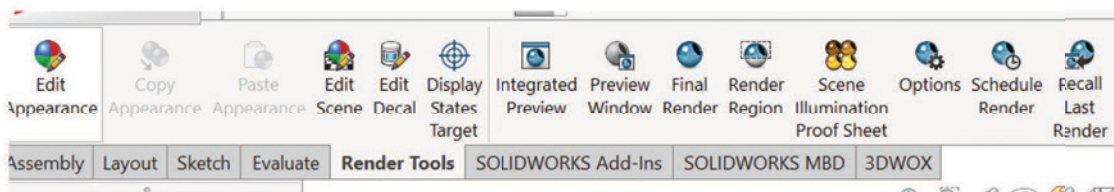
Photorealistic Images

Turn on **PhotoView 360** by selecting the SolidWorks Add-Ins toolbar and select PhotoView 360



PhotoView 360 Toolbar

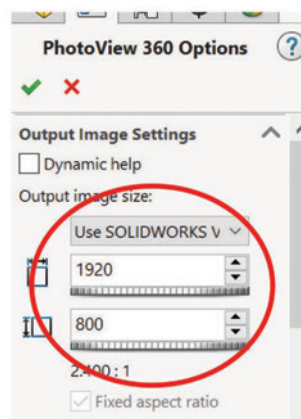
The PhotoView 360 toolbar is now added to the tabs list.



Select **Options** from the toolbar to adjust the Photorealistic image settings.

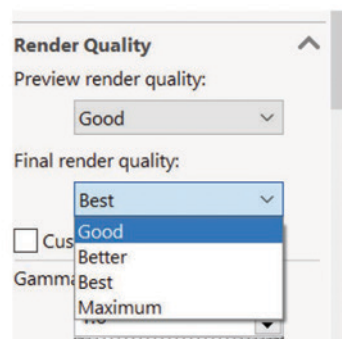
Image Size

Increase the Output image size of the image to achieve a better quality photorealistic image



Render Quality

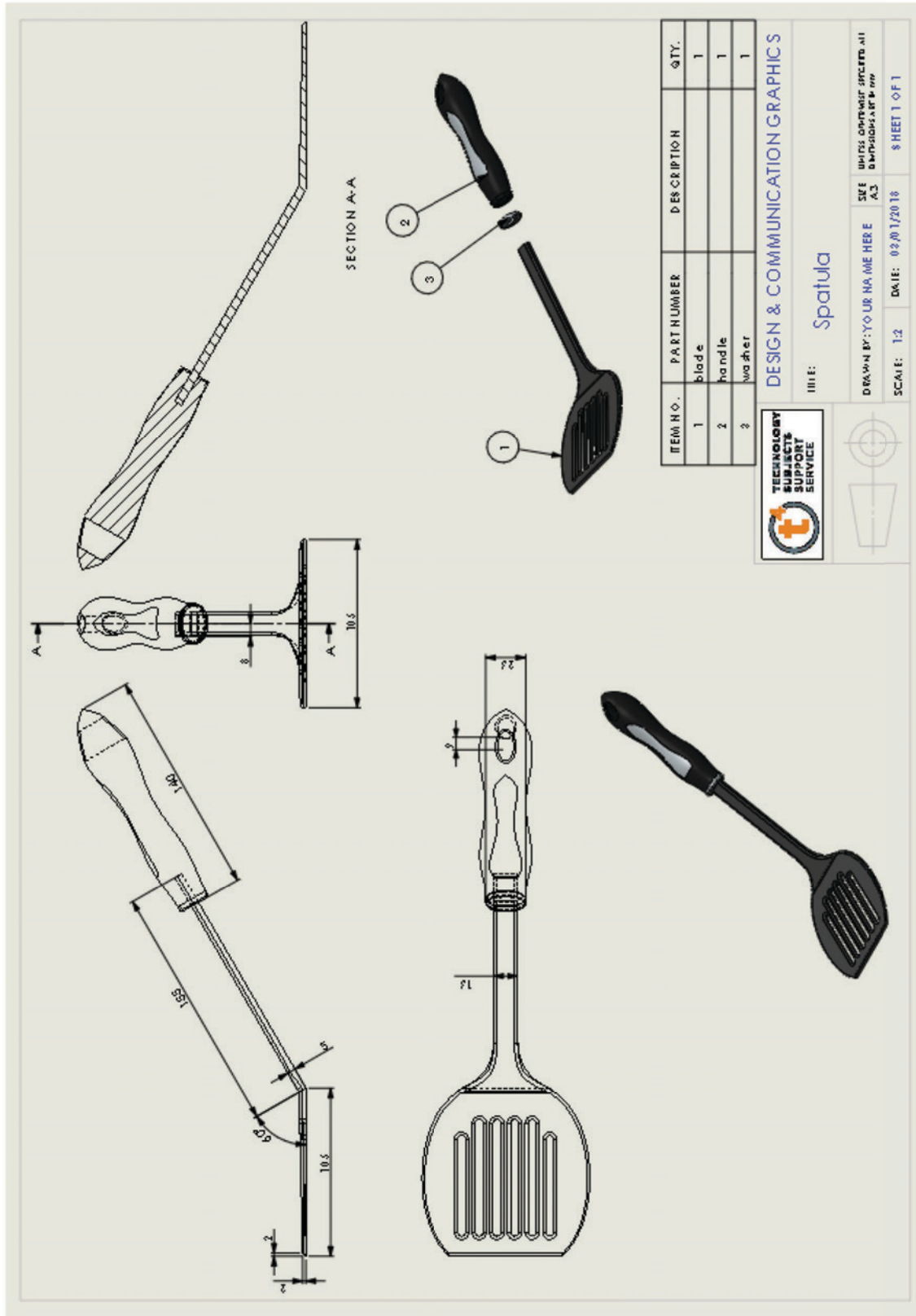
Adjust the **Final render quality** up to **Best** to achieve a very good quality final image.



Accept these changes



Drawing





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This resource is available to download from www.pdst.ie