Wireless Mouse

Surfaces

Table of Contents

Wireless Mouse

Introduction

Surface modelling is clearly a lot more work than solid modelling. Surface modelling forces you to work face by face and faces must fit together. These action are all performed automatically in solid modelling. Where Surface techniques become beneficial is in a situations where solid modelling become clumsy or inefficient, or when a given modelling task is simply impossible with solids. The focus of this exercise is to add to the skills based on the advanced surface modelling materials and explore further the functionality of surfacing tools.

Learning Intentions

In addition to the normal command the following commands will be used.

- o Splines
- o Projected Curves
- o Lofted Surfaces
- o Filled Surfaces
- o Knit Surfaces
- o 3D Sketch Convert Entity
- o Create Solid
- o Dome

Mouse Body

The part is symmetrical with respect to the **Right** reference plane. Open a new part and name it Mouse. Select the **Top** reference plane and sketch a rectangle as shown. Exit the sketch and rename the sketch **Reference Dim**. This sketch will help you sketch the free-form splines approximately the correct size.
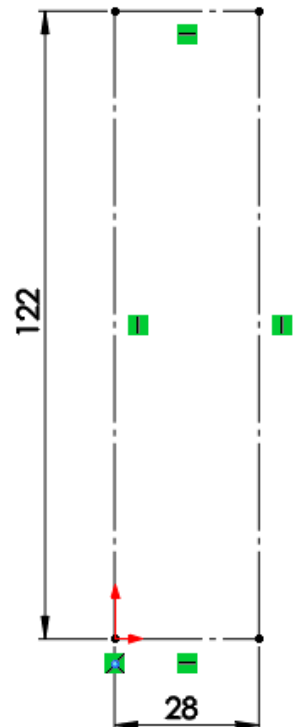
Bottom Edge

Open a new sketch on the **Top** reference plane, and sketch a 5-point spline as shown to represent the bottom edge of the mouse. Draw only half of the outline. Make sure that the endpoints of the spline are **Coincident** with the corners of the rectangle in the **Reference Dim** sketch.
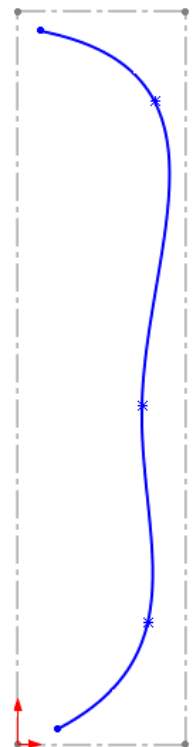
Add Relations

Use relations on the handles at the ends of the spline to create tangency across the line of symmetry. Add **Horizontal** relations to the **Spline** handles
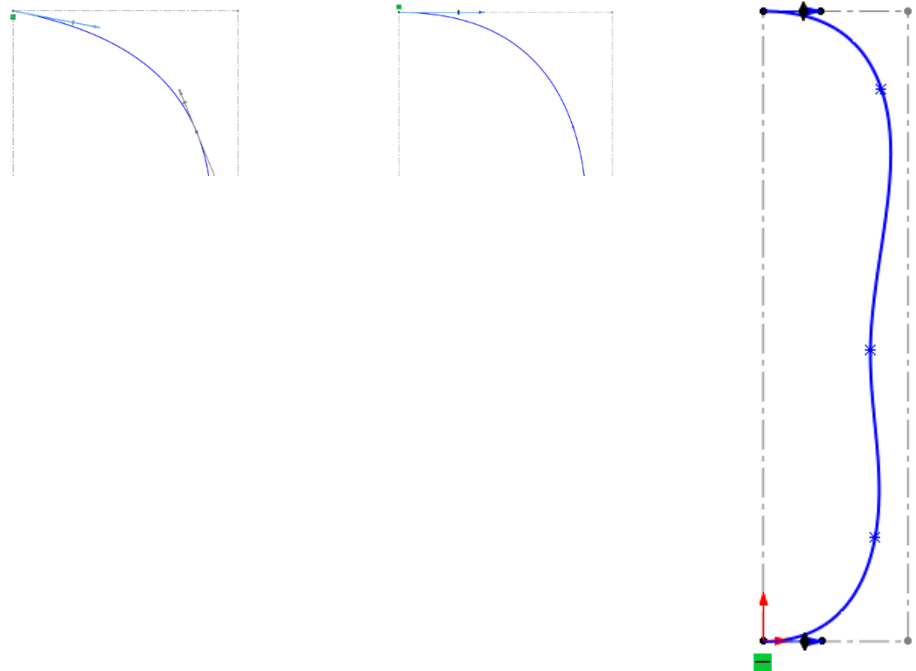
Tangent Relation

To make the spline tangent to the rectangle, sketch a short construction line Tangent to the spline itself, not to a handle or a spline point. Add a **Vertical** relation to the construction line. Make the **Endpoint** of the construction line **Coincident** with the rectangle.
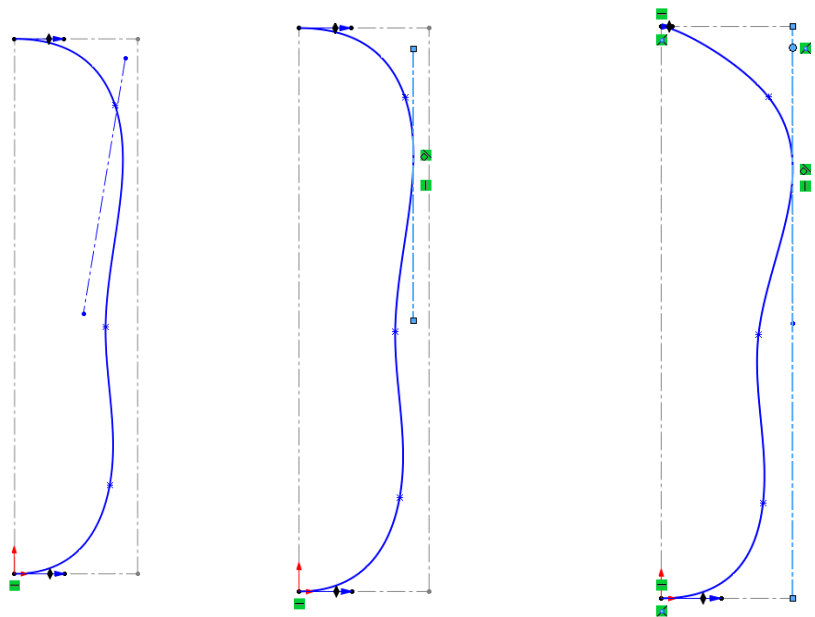
Adjusting the shape of the spline

In the Reference Dim Sketch Adjust the shape of the Spline by dragging the interpolant points. Select fully define the Spline. Exit the sketch and rename the sketch **Base Edge**
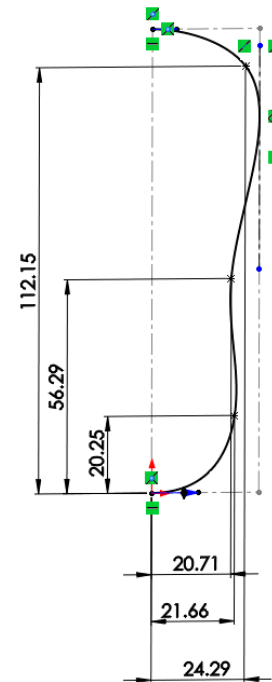

Sketch top profile of the parting line.

Open a new sketch on the Top reference plane. Sketch a spline which lies slightly outside the Base Edge sketch. The endpoints of this spline should have **Vertical** relation with respect Base Edge sketch and should be dimensioned as shown. Add relations to the end handles as you did in the previous sketch. Notice the use of **Show Inflection Points** on the spline to help to define the area of slightly reversed convexity. Exit the sketch and name it Profile Top
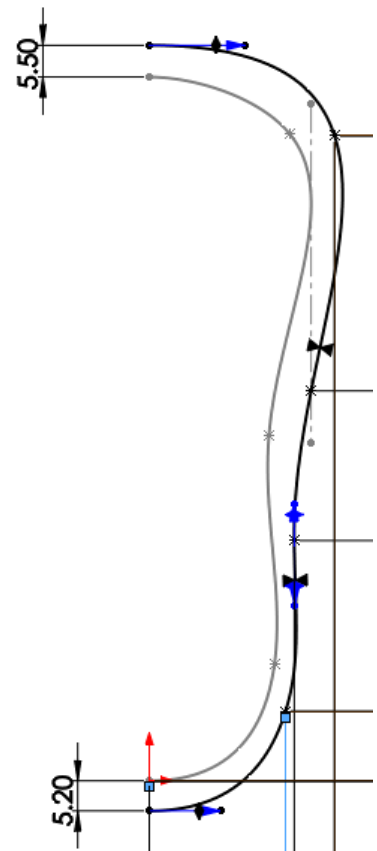
Sketch the side profile of the parting line.

Open a new sketch on the **Right** reference plane, and draw a spline shown below. The ends of the spline should have **Vertical** relations with respect to the Endpoints of the spline in the Profile top. Add a **Horizontal** relation to the handle at the end of the spline furthest from the Origin. Exit the sketch and name it **Profile Side.**
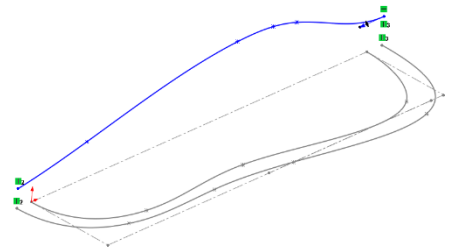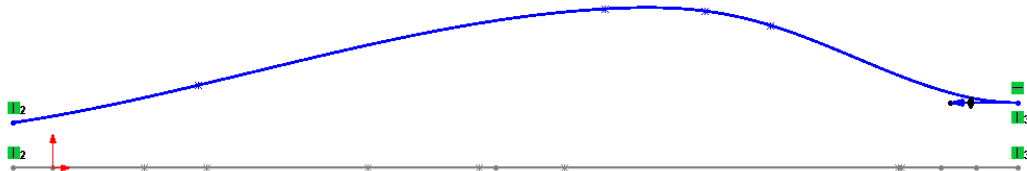

Project Curve

Search for the Project Curve command and drag the command into the sketch toolbar. Use the **Sketch onto Sketch Option**. Select the Profile Side and Profile Top. Rename the projected curve **Projected Curve**
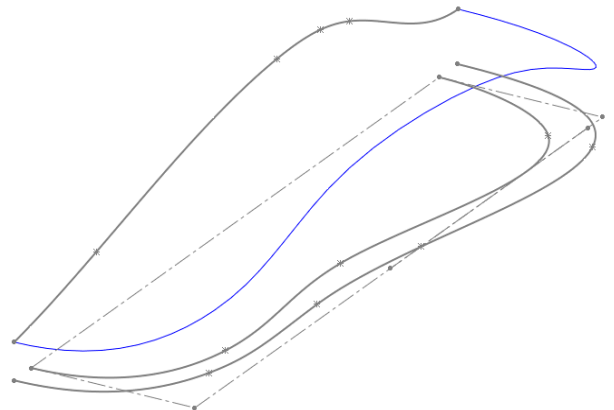
Create loft profile sketch.

Open a new sketch on the **Right** reference plane, and sketch a pair of arcs as shown. The arc near the Origin should be tangent to a line 15 degrees from horizontal and the other arc should simply have a radius of 10 mm.

Both arcs should be **Coincident** to the ends of the **Base Edge** sketch and be pierced by the Projected Curve.
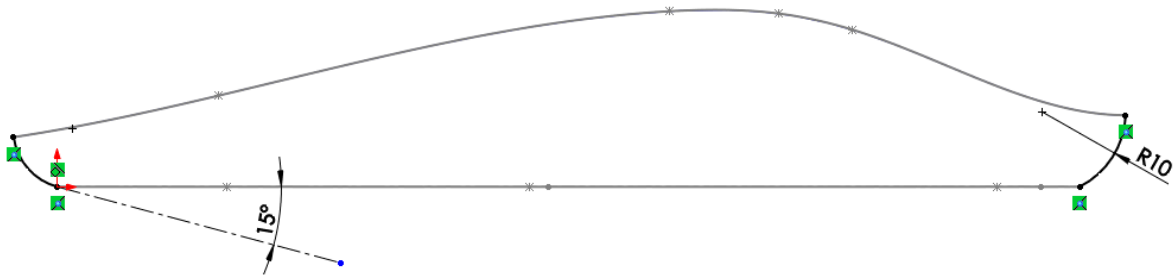
Note

Using the Selection Manager, multiple, disjoint profiles in a single sketch are manageable and valid.

Create third loft profile.

Create a new plane parallel to the **Front** reference plane through a spline point from the Base Edge sketch. Rename this plane Mid Plane.
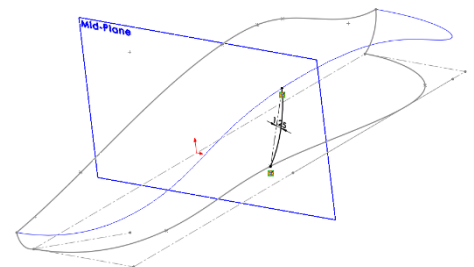
Mid Profile.

Open a new sketch on the Mid Plane reference plane. Sketch an arc. Add a Coincident relation between the bottom endpoint and the spline point the plane was created from. Add a Pierce relation between the top endpoint and the Projected Curve. Draw a construction line between the endpoints of the arc. Using the Smart Dimension tool, select the construction line, and then hold down the **Shift Key** and select the **Arc**. This will give a dimension as if the Min arc condition in the dimension properties was used. 1.25 mm. Exit the sketch and name it **Mid Profile**
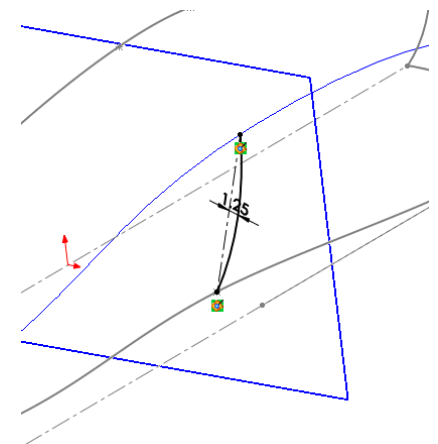
Create the Surface Loft.

Use the Selection Manager to select open profiles at the ends.
The Mid Profile sketch will not require the Selection Manager.
For Guide Curves, select Projected Curve and Bottom Edge
Edge. For Start/End Constraints, use Normal to Profile for both
ends so that it is smooth across the plane of symmetry.
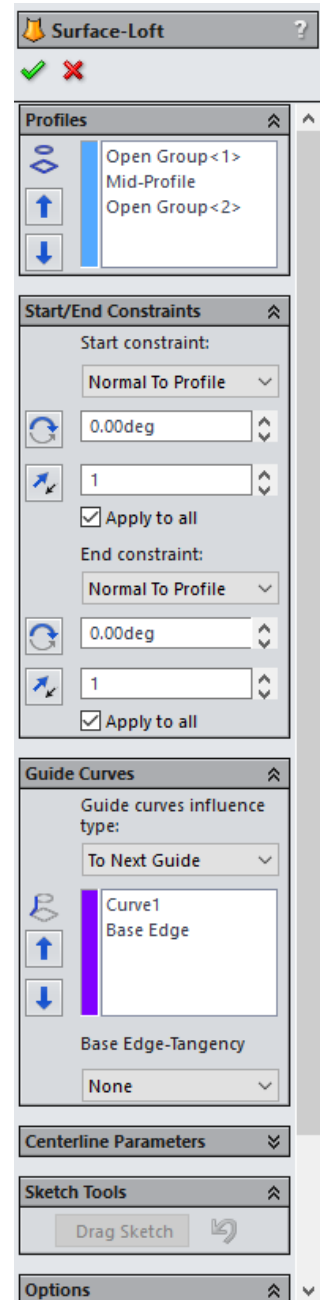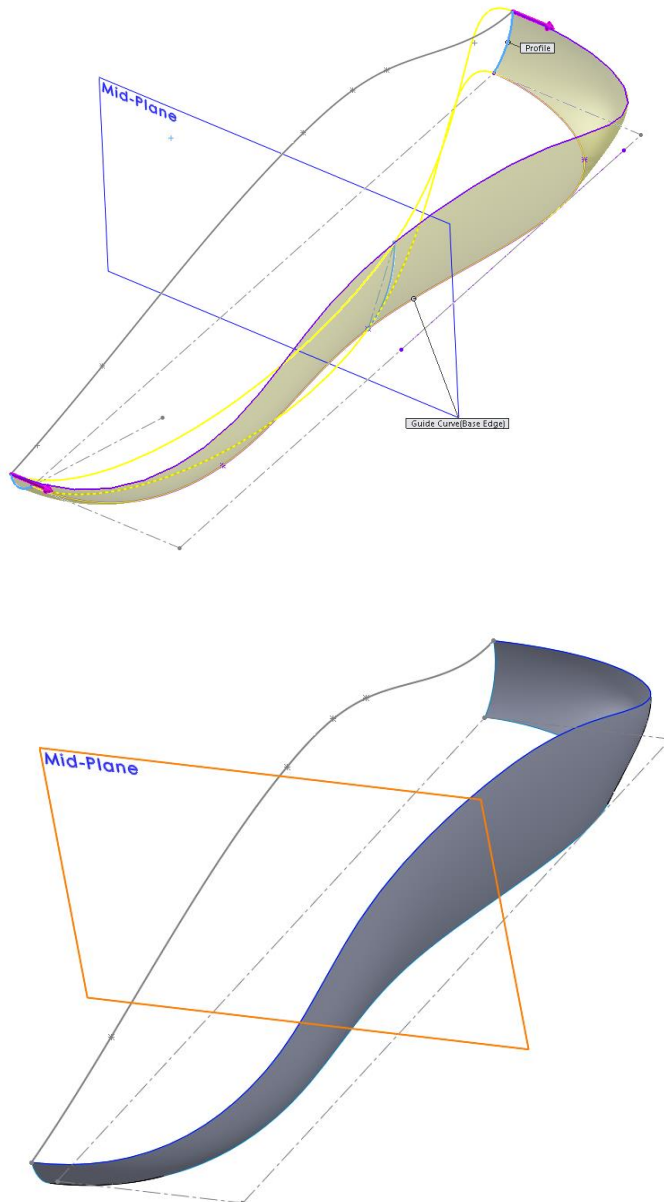
Top curve of the mouse.

Open a sketch on the **Right** reference plane and sketch a partial ellipse at an angle. Sketch a construction line from the centre of the ellipse to the end of the minor axis to control the angle. Add **Pierce** relations between the endpoints of the ellipse and the Projected Curve Add a **Coincident** relation between the end of the major axis and the rightmost endpoint of the ellipse.
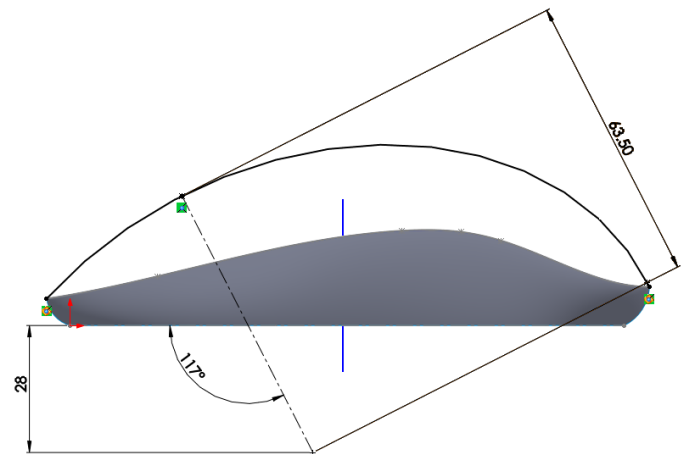
Extrude a reference surface.

Filled Surface requires a reference surface to define the tangency condition along the top. Extrude the partial ellipse sketch away from the rest of the model. The distance doesn't matter.
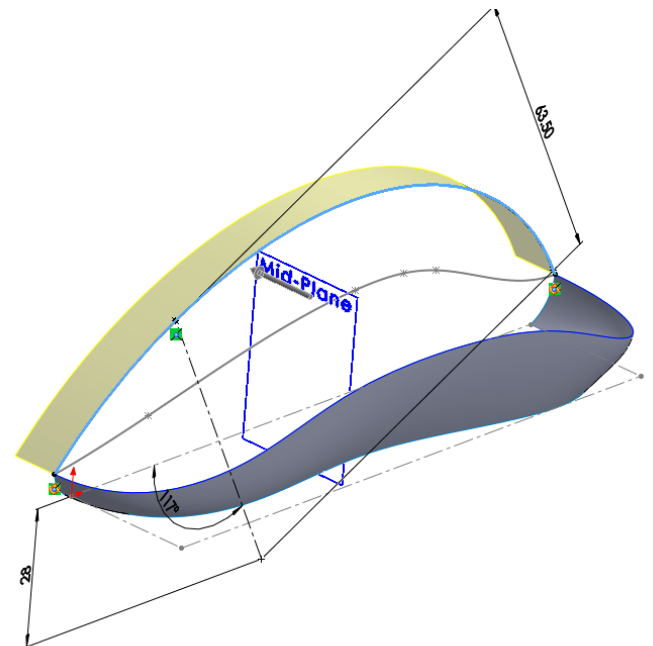
Filled surface

Click the **Filled Surface**. Select the **Edge** of the extruded surface and the edge of the lofted surface. Use the end condition **Tangent** for the extruded surface and **Contact** for the lofted surface. Notice that with Optimize surface selected, the surface again becomes degenerate. This is because the Optimize surface option applies a simplified surface patch that is similar to a lofted surface.

Clear **Optimize surface** and you will get a better, four-sided patch. Click OK.

Hide the Extruded Surface
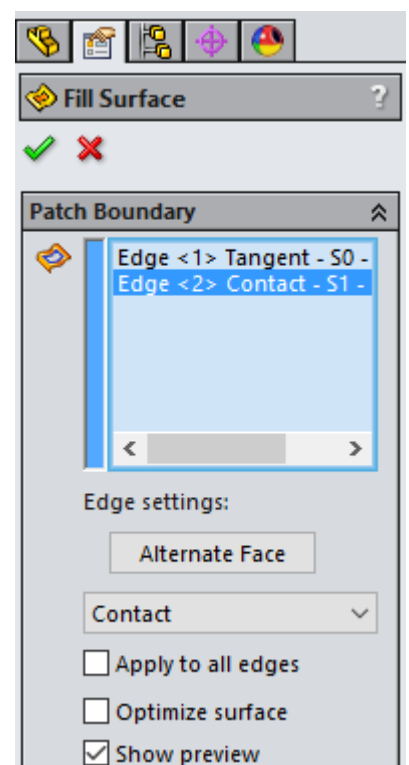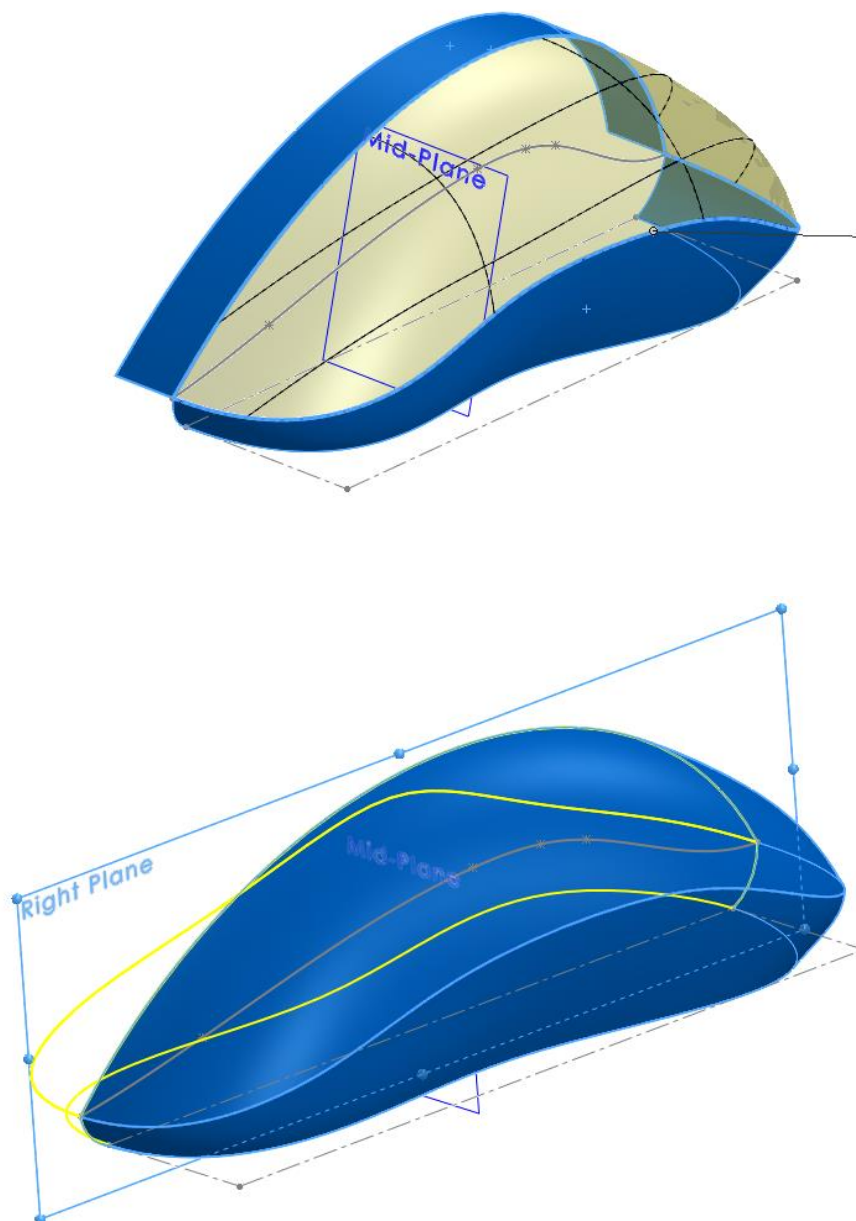
Mirror the surface bodies.

Click Mirror on the Features toolbar. Select the Right reference plane as the mirror plane. In the **Bodies to Mirror** selection list, select the lofted and filled surfaces to be mirrored Uncheck Knit surfaces. Leaving the Knit surfaces option cleared this way there is no confusion about what will or will not knit by the Mirror feature
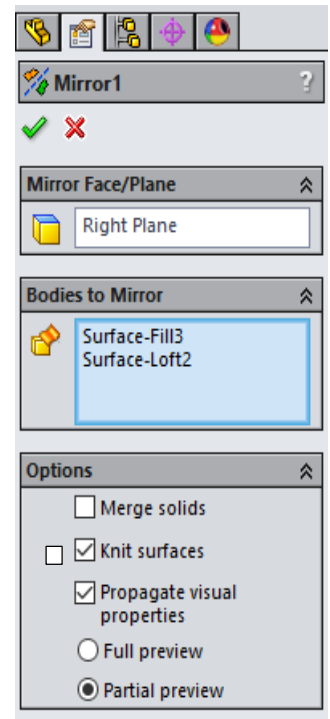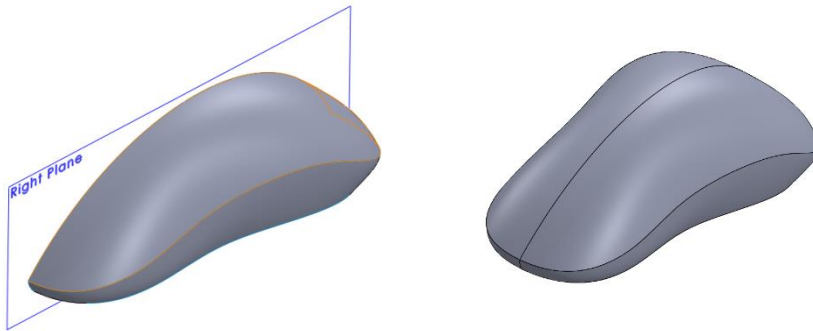
Planar surface.

Select the edges of the lofted surface and mirrored loft on the bottom and create a planar surface.
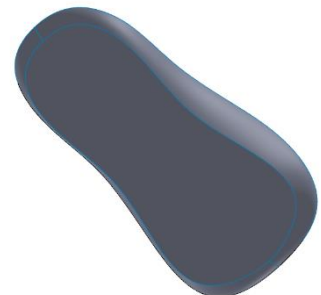
Knit

Knit the five surface bodies into a solid Save the Part Select

Try to form Solid
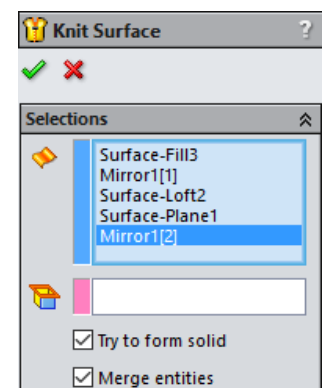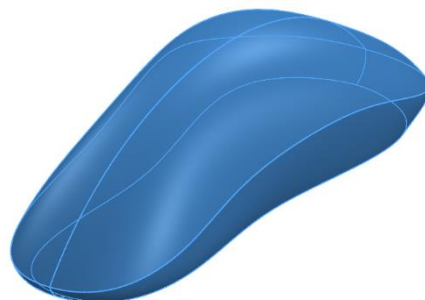
Edge Cut (Extrude Cut)

Select the **Right** plane and draw a rectangle on the Left hand side to the dimensions shown exit the sketch.
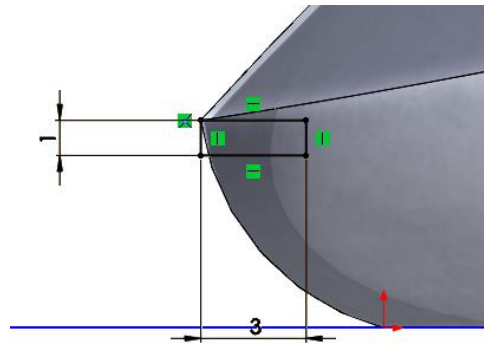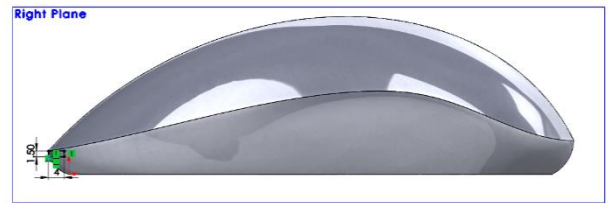
Convert Sketch Top Plane

Select the top plane and use the convert entity command to the top plane
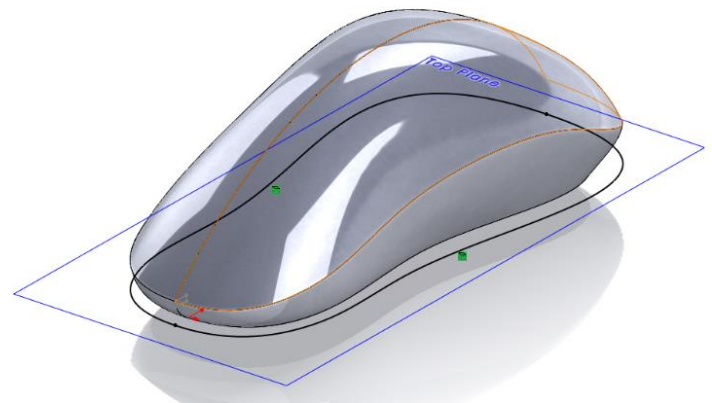
3D Sketch Convert Entity

Select **3D Sketch** and use the **Convert Entity** command for the curve.
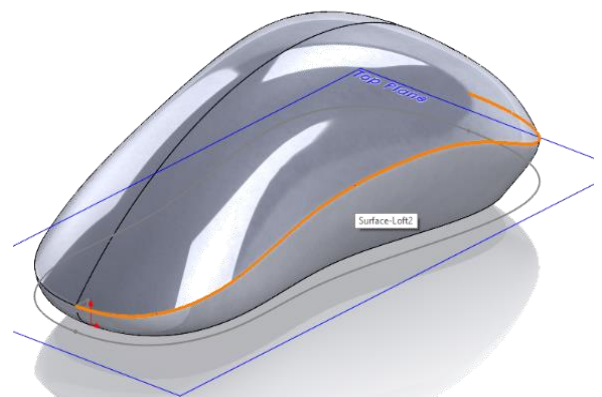
Edge Cut

Select the Swept Cut command Select the

Profile Path and Guide Curve

Centre Cut

Select the **Right** plane and use the Convert Entity sketch command and trim the line to the dimension shown.
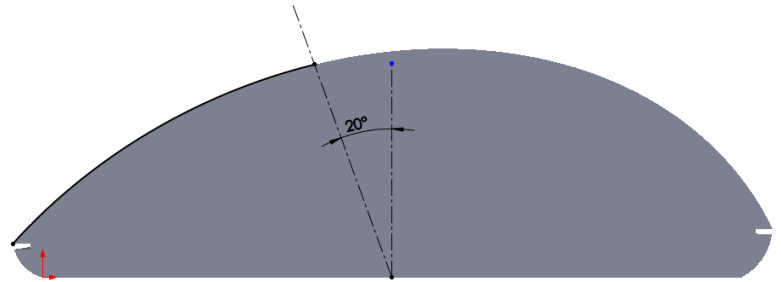
Note: A vertical construction line at the centre.

Use the Section View command to assist in the create the centre line

Create a Plane on the end of the trimmed sketch.
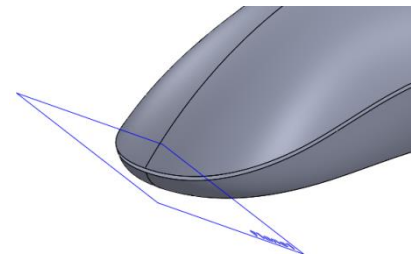
Sketch a square to the dimensions shown
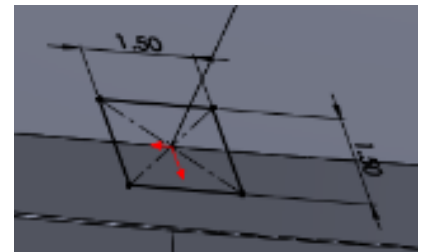
Select the Swept Cut to create the mid Centre Cut
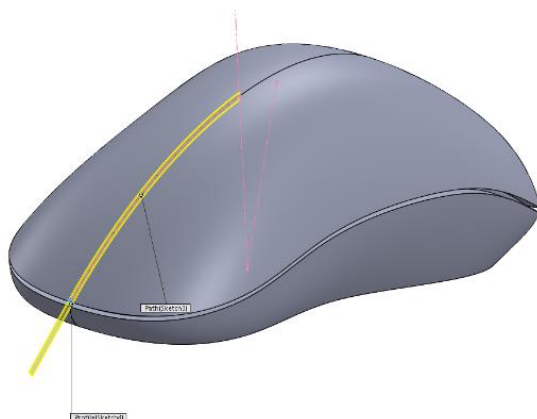
Wheel Opening

Select the **Right** plane and create sketch

to the dimensions shown
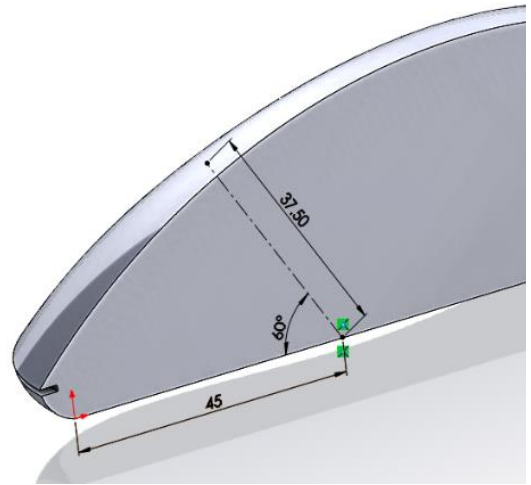
Plane Creation

Create a plane at the end of the

construction line
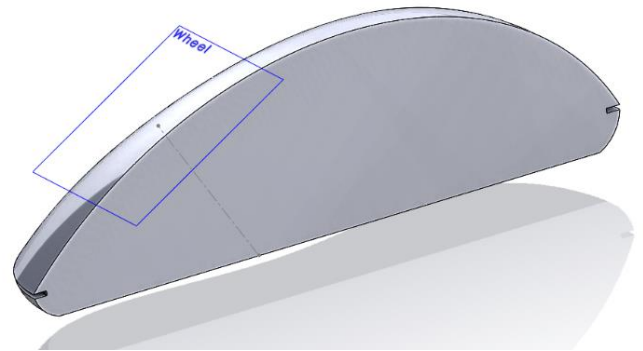
Straight Slot

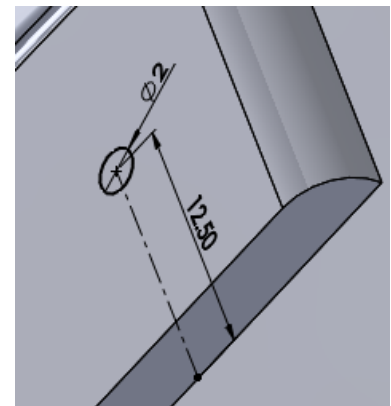Use the **Straight Slot** command to create a sketch to the dimension shown and

Extrude 21mm
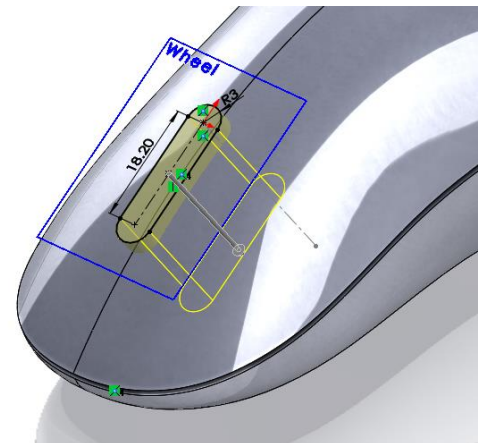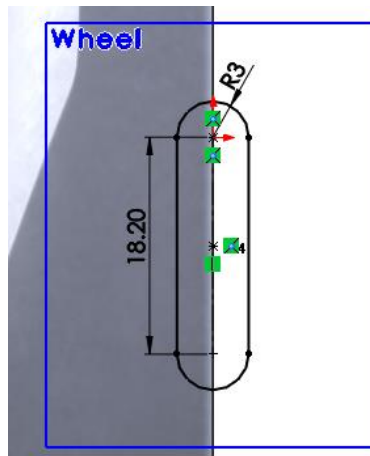
Fillet 5mm

Create a fillet 5mm
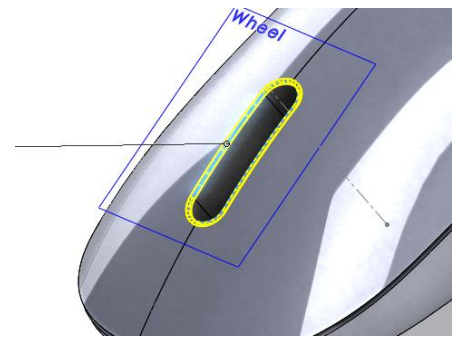
Wheel Location

Select the **Right** plane and reference the **Midpoint** of the wheel slot create a line 12.5mm and a circle radius 2mm. Use the Extrude Cut command and select midpoint extrude distance of 14 mm
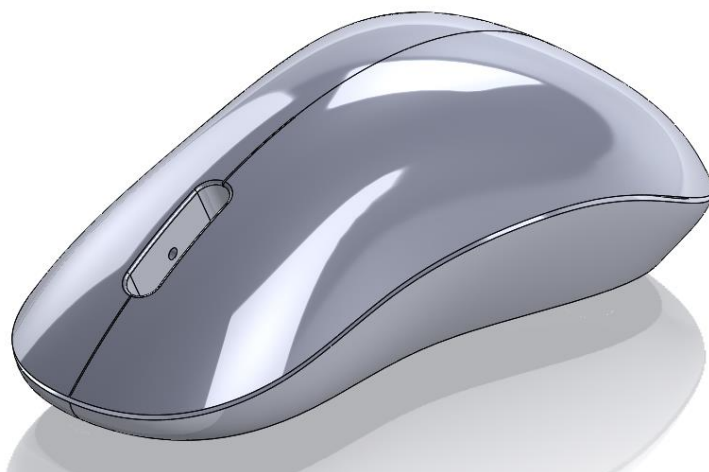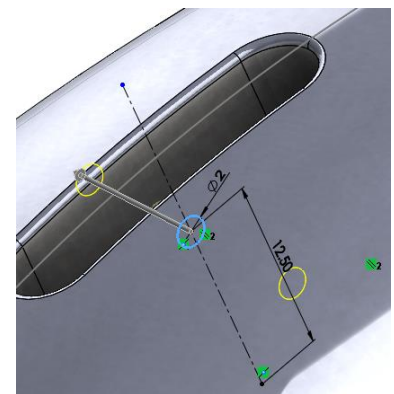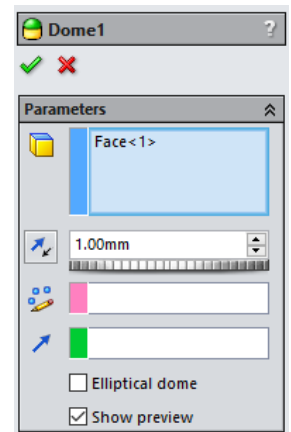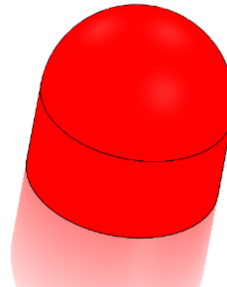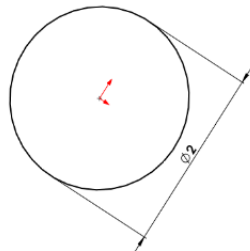
Base Pads

Select the base of the mouse body and create the location of the mouse pads to the dimension given. Extrude the four pads to a dimension of .25mm. Select the Extrude Cut command to create the laser opening dimension 4mm
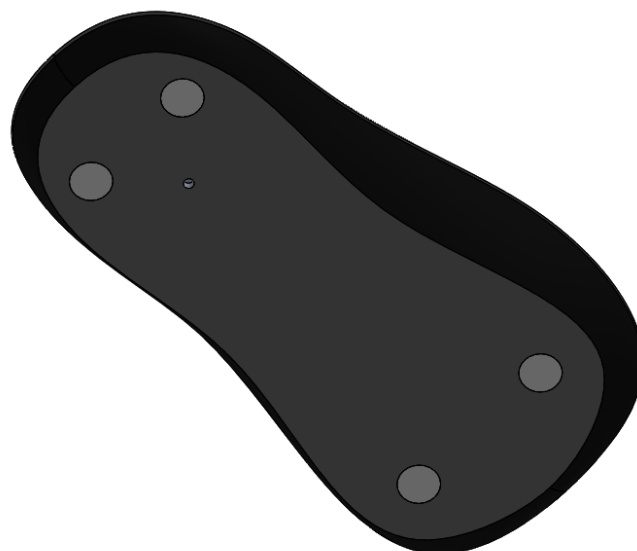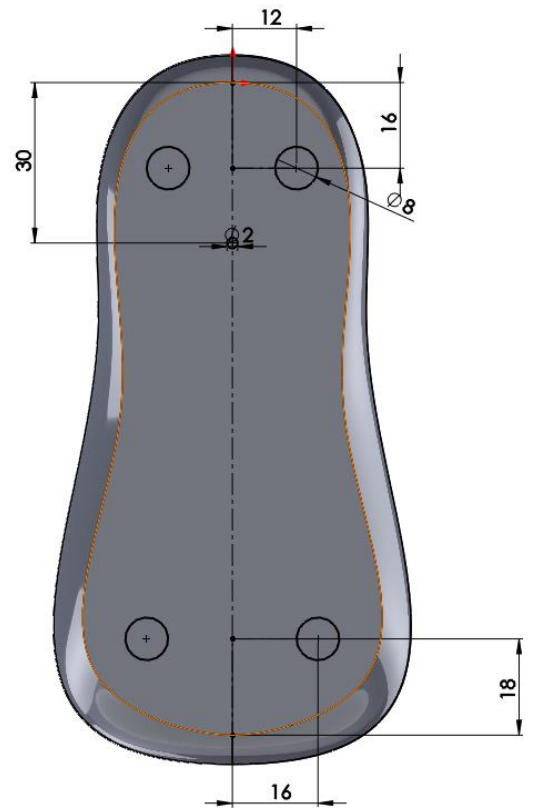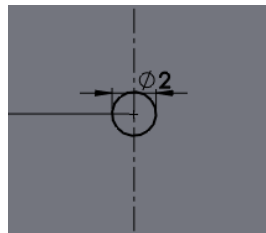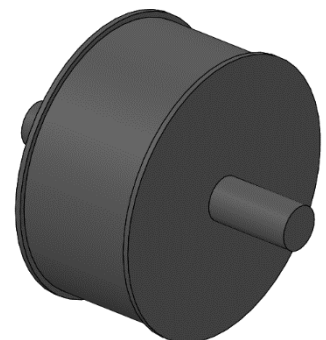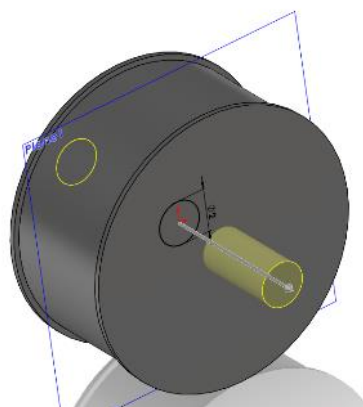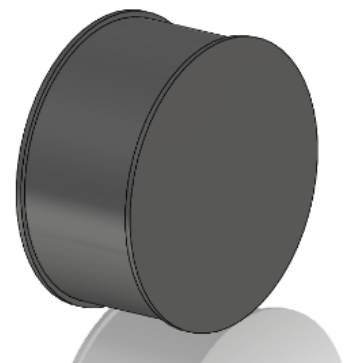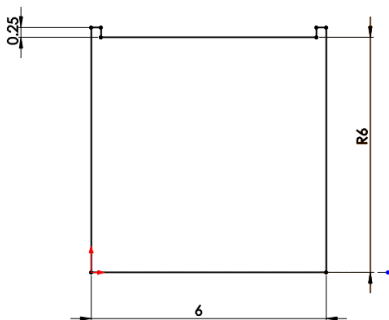
Laser (New Part)

Sketch a circle of 2mm radius extrude to a distance of 1mm. Use the Dome command to create the top of the laser


Wheel Hub (New Part)

Create the sketch to the dimensions below and revolve to create the wheel hub. Create a circle of 2mm on the mid plane and use the Mid Plane End Condition to a distance of 14mm
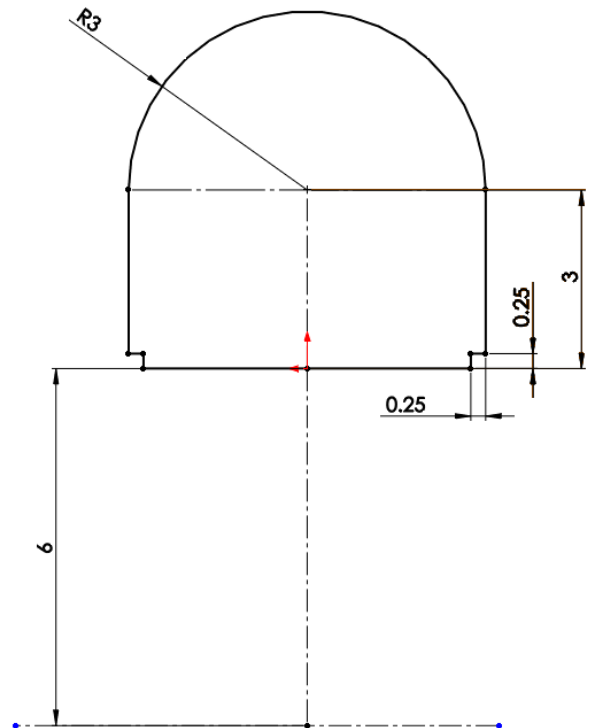
Wheel Hub Design

Create a sketch to the dimensions shown and Extrude Cut to depth of 5mm. Create an Axis and using the **Circle Pattern** command create the design with the number of instances equal to 4.
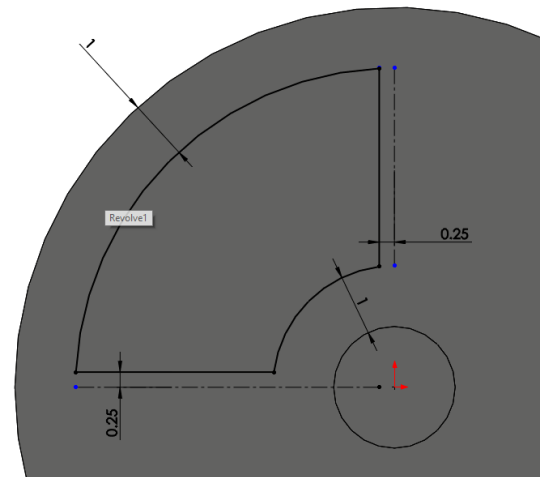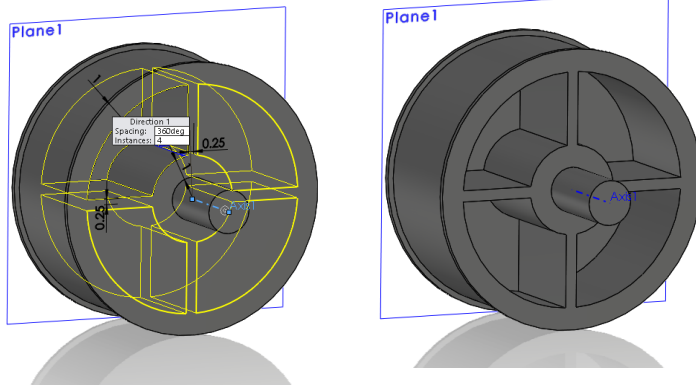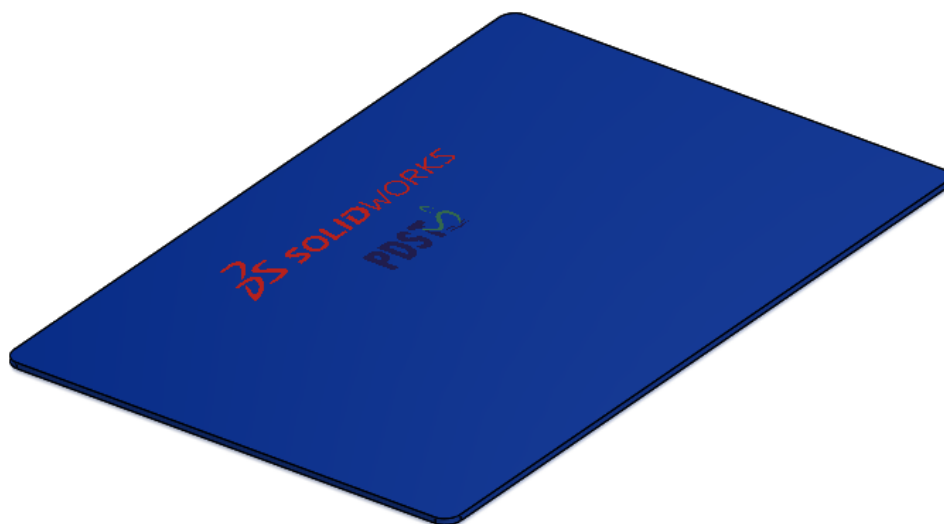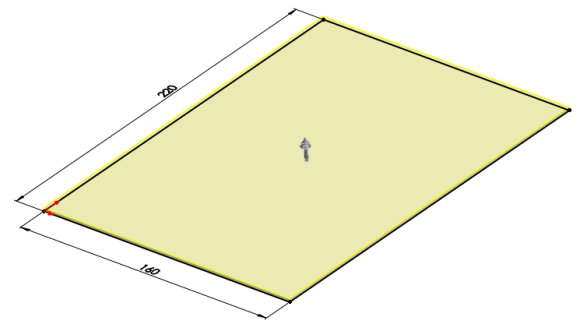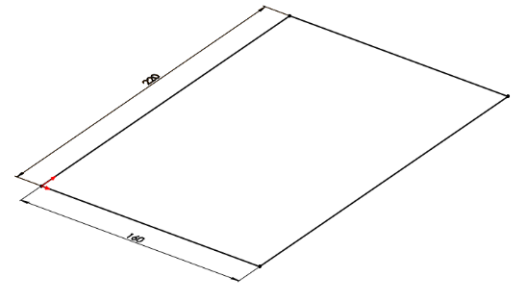

Wheel (New Part)

Create the sketch and using the Revolve Command to create the wheel

---

Mouse Mat

Create a rectangle 200mm by 160 mm and **Extrude** to a distance of 5mm and fillet the edges. Apply Decals to the mouse mat.

Assembly

Apply **Appropriate Appearances** to all parts. Create an assembly.