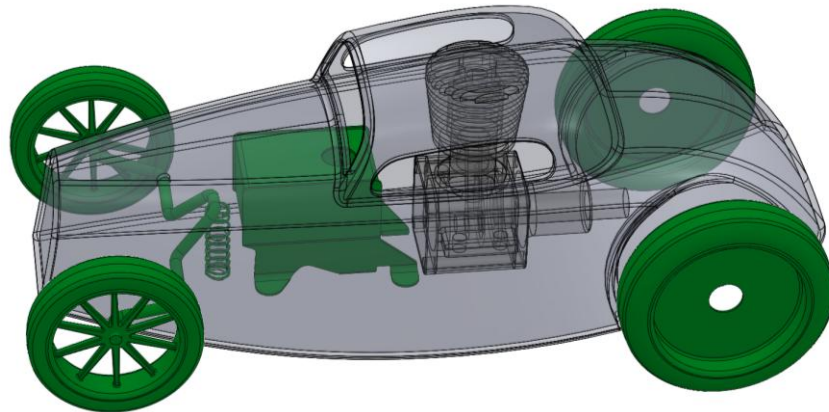


TUTORIAL 2 More Modelling Techniques

In the first tutorial we focused on the engine components, which mainly involved sketching, extruding, revolving, and filleting. We saw that using combinations of simple features created by these tools can create seemingly complex geometry. Now we will continue exploring some additional tools to model the wheels, front axle and suspension, and the fuel tank.

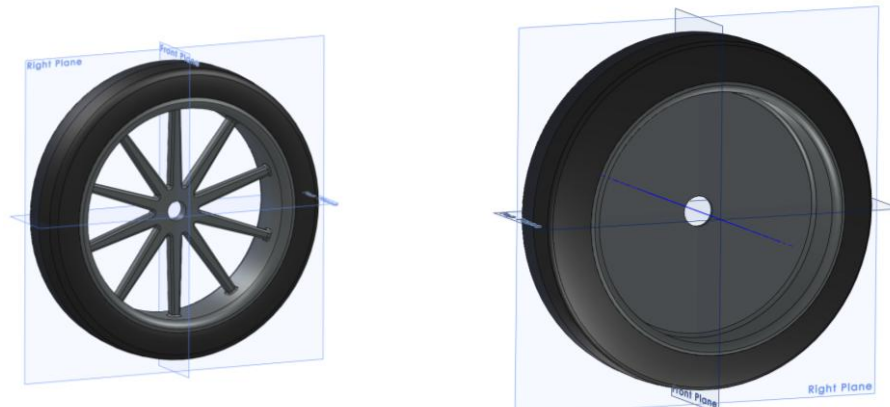
Goals:

- Understand the difference between 2D and 3D sketches, and be able to create a 3D sketch to generate a sweep feature
- Be able to create helical curves and model springs
- Use open profiles to create thin solid parts
- Apply rotary patterns of features
- Understand and use a multi body part
- Create sheet metal parts using a variety of different tools

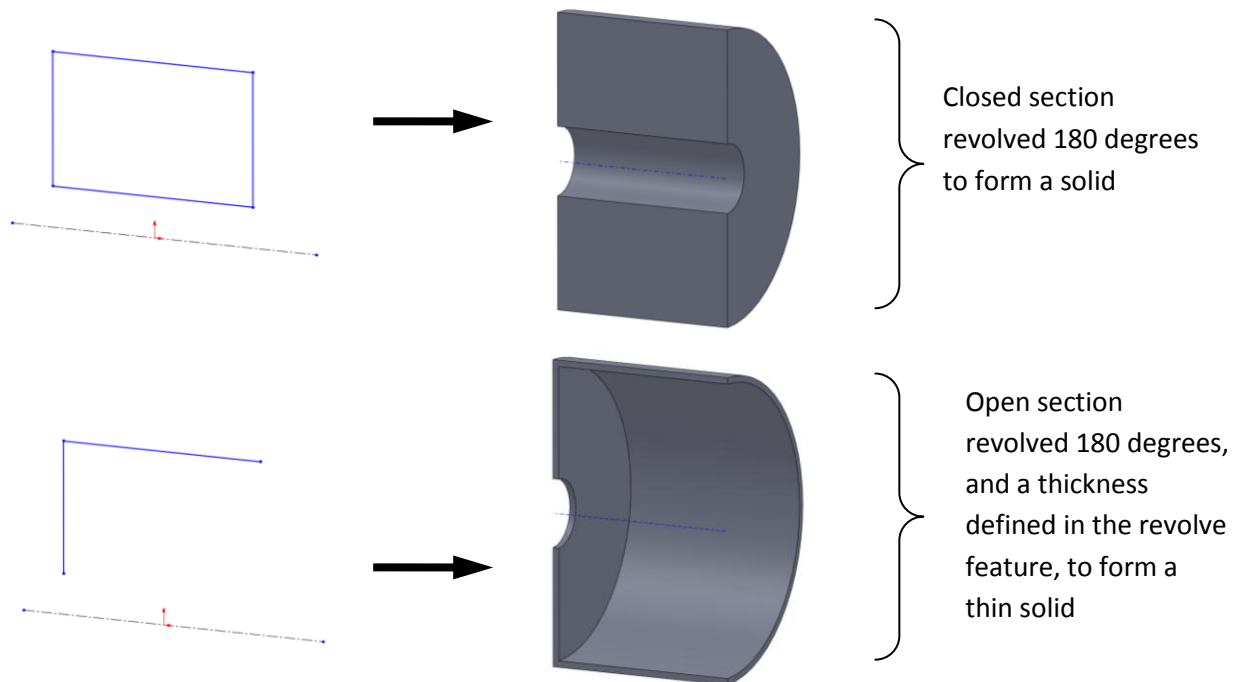


Wheels & Tyres

The front and rear wheels are similar, but are differentiated by their size and the absence of spokes in the rear wheels. This is a typical arrangement for a drag racing car which reflects the requirements for the front and rear wheels. The rear wheels are wide so they can have large tyres to generate lots of traction for acceleration, and are often disc wheels (no spokes) so they are able to withstand a large torque applied from the axle. The front wheels don't need much traction at all, and the only torque applied to them is when the relatively small front brakes are applied. Accordingly they are narrow and have thin spokes which make for a light wheel.

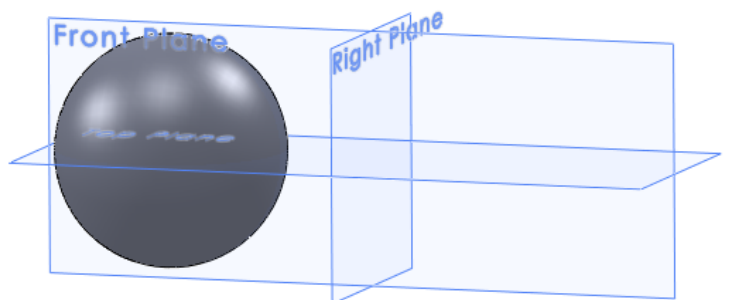
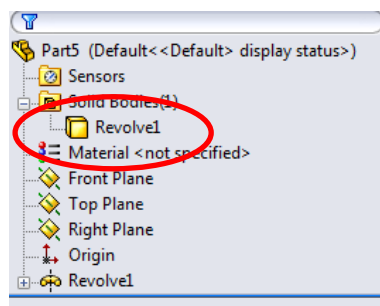


Our plan to model both wheels is similar too, and will introduce some new concepts. The wheel itself (not including the tyre) will be created using a thin Revolve feature. A thin Revolve is similar to the revolves that we have done previously, but instead of using a closed sketch to revolve around an axis, we'll use an open sketch and then define a thickness for it.

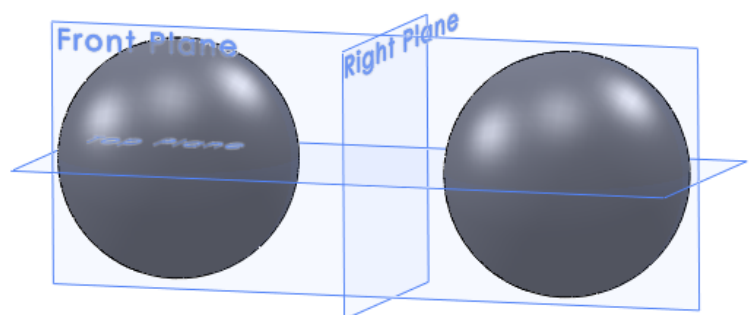
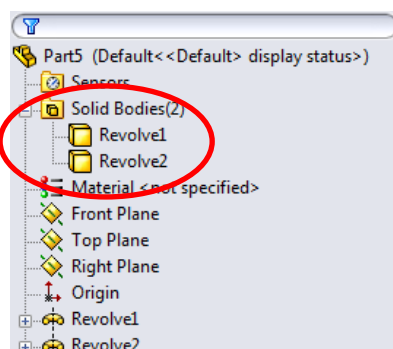


The other new concept is that of multiple bodies in a part. In SolidWorks we are not limited to having only one 'lump' of material in each part. We can have as many as we like, and each 'lump' is called a body. In this case we will model the wheel and tyre as two separate bodies within the same part. This allows the tyre to conveniently reference the size of the wheel; therefore it will change automatically if the wheel size is modified.

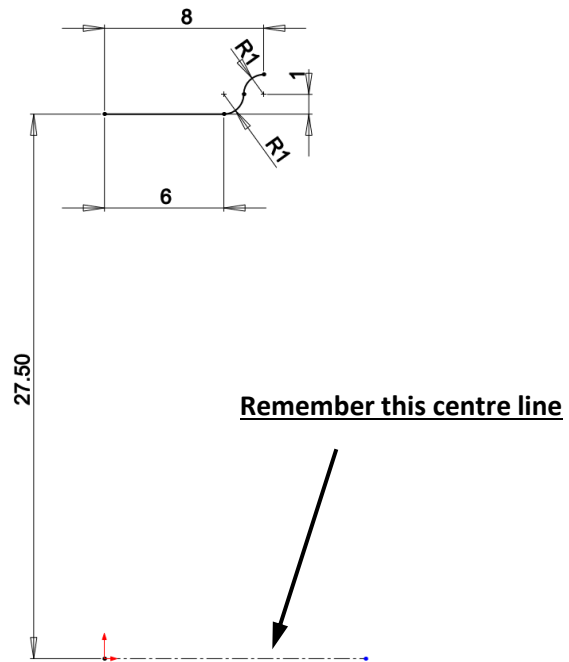
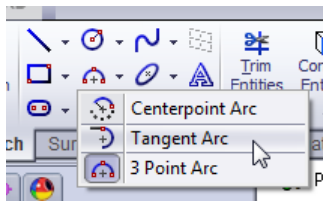
One body in
Solid Bodies
folder



Two bodies in
Solid Bodies
folder

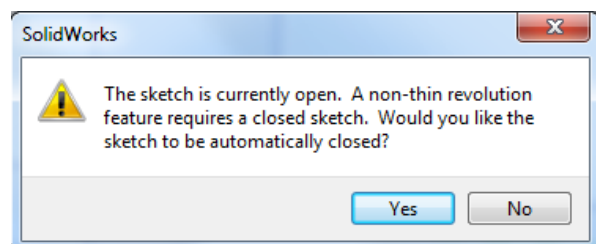


Begin with the front wheel by creating a sketch on the Front plane as shown. The left hand end of the horizontal line should have a sketch relation to make it coincident with the Right plane. The two arcs are sketched using the Tangent Arc tool in the Sketch tab of the Command Manager. Don't forget the Centreline extending from the origin, which will be used as the axis of revolution.



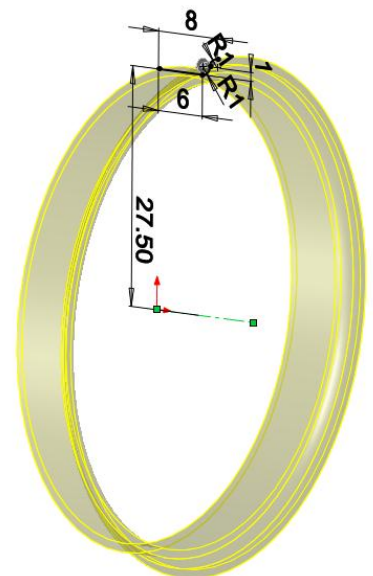
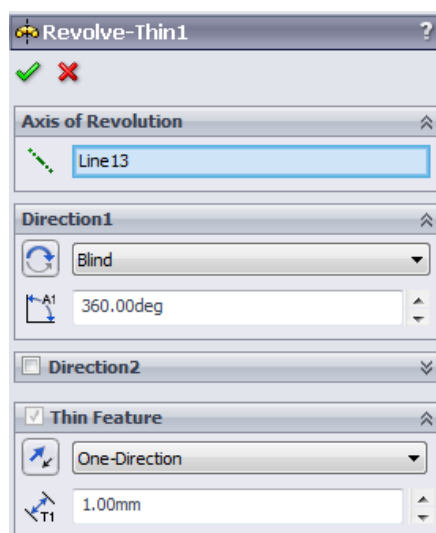
Note that this is an open sketch, or in other words it does not form a closed loop.

Exit the sketch and select the Revolve Boss/Base tool. Select the sketch in the Graphics area and a popup window should appear warning you that this is an open sketch and giving you the option to automatically have it closed so a solid can be generated in the revolve. In this case select No.

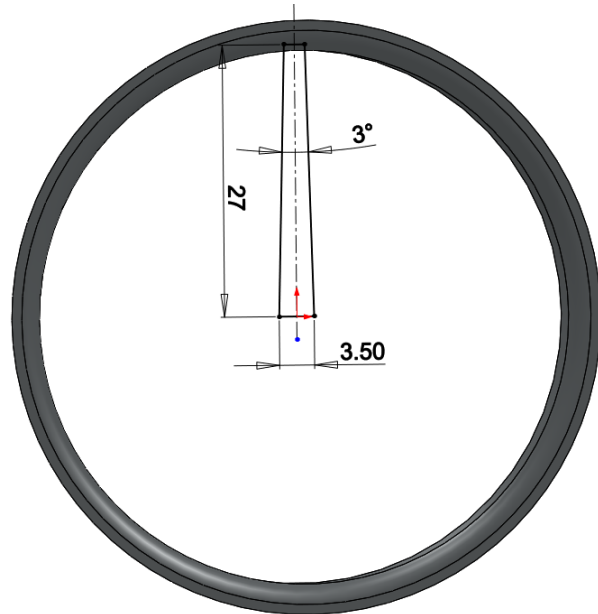


Because it is an open sketch, and we've confirmed this is what we want, the Revolve dialogue changes so that a thin revolve can be created. This is the same as a normal revolve except a thickness is specified, and a direction for which side of the open sketch the thin solid will be formed on.

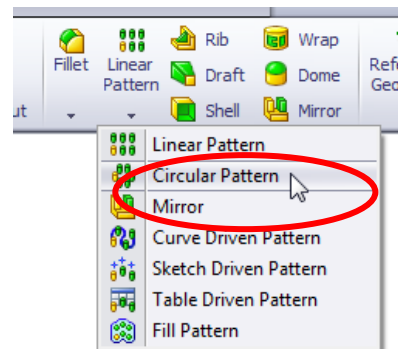
The Axis of Revolution in this example is Line13, but it will likely be called something different in your model. The important thing is that it refers to the centreline in the sketch. Enter the other values as shown so the sketch will revolve around 360 degrees and provide a part thickness of 1mm. Accept this by clicking on the green tick and the thin solid rim will be generated.



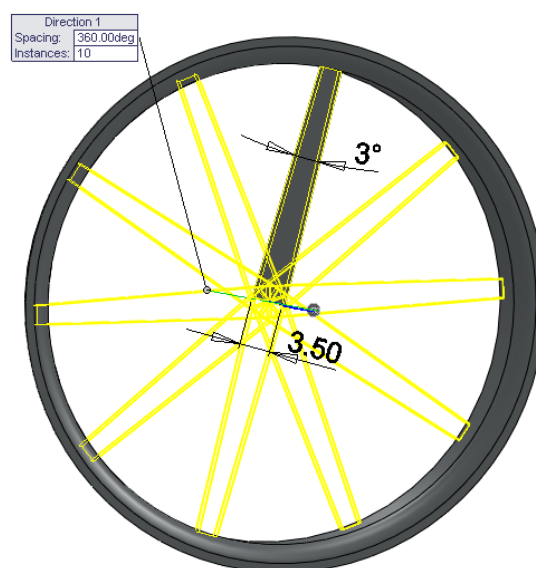
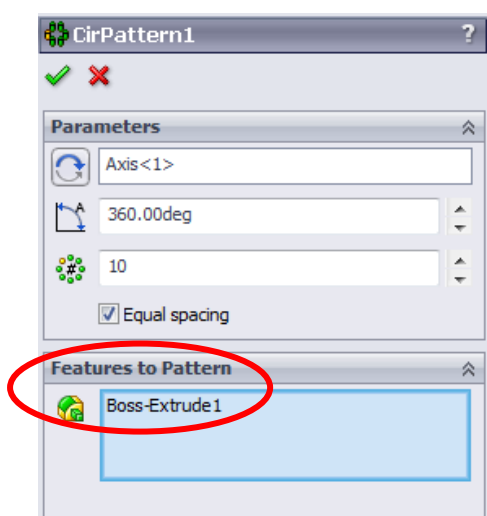
The spokes are modelled next, by first modelling one of them, and then patterning this feature to generate the others. Sketch on the Right plane using the dimensions as shown and extrude this profile 1.5mm.



Select the Circular Pattern tool from the Features tab of the Command manager. Similarly to when we created a Mirror feature, there is the option to pattern Features, Faces, or Bodies.



In this case we want to pattern a feature only, so select the **Features to Pattern** selection box in the Circular Pattern dialogue and enter a value of 10 for the number of instances. You should see the yellow preview as shown in the example.



Click the green tick and the pattern feature will generate all of the wheel spokes as rotated copies of the original spoke. If we were to edit the dimensions of the original spoke the pattern feature will update to include the change in all of the other spokes also.

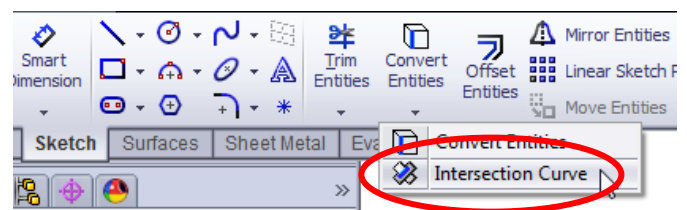
The spokes now need to be blended into the rim and have their edges rounded. Use the Fillet feature to first place a radius at the inner end of the spokes where they contact each other.



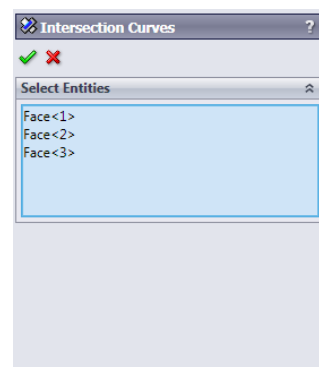
To complete the filleting, choose the Fillet tool again and this time select the front face of spokes and apply a 0.5mm fillet. Selecting a face when using the fillet tool automatically selects all of the edges that surround that face. This is often an efficient way of selecting a lot of edges.

The front rim half (we will mirror it later) is now completed, so next is to model the tyre. Again, we will actually model only half of the tyre because our design intent is that it should be symmetrical about its centre plane. The tyre will be generated using a standard (not thin) revolve as we want our model to have solid tyres rather than the hollow pneumatic tyres of a full size car. Because the wheel and tyre are different components and we might like to actually separate them into different parts later, we will model the tyre in such a way that it will be a separate body in the part, making a total of two unconnected bodies: the tyre, and the wheel.

Create a new sketch on the Front plane and select the Intersection Curve tool from the Sketch tab of the Command Manager. This tool creates sketch lines for selected surfaces where they intersect with the plane you are sketching on.



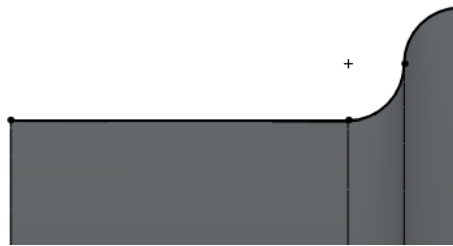
Use this to create sketch lines on the surface of the outer of the wheel rim. As the surfaces are selected they will highlight green in the Graphics area.



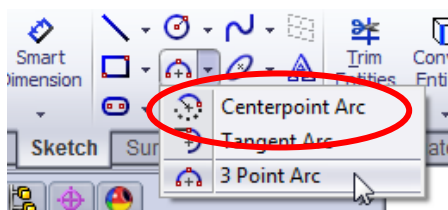
Because we are selecting a cylindrical surface it actually intersects the sketch plane twice, once at the top and once at the bottom. We only want the sketch lines at the top, so after accepting the results of the Intersection Curves tool, select the three curves at the bottom of the wheel and press the delete key to remove them.

Your sketch should now look as shown.

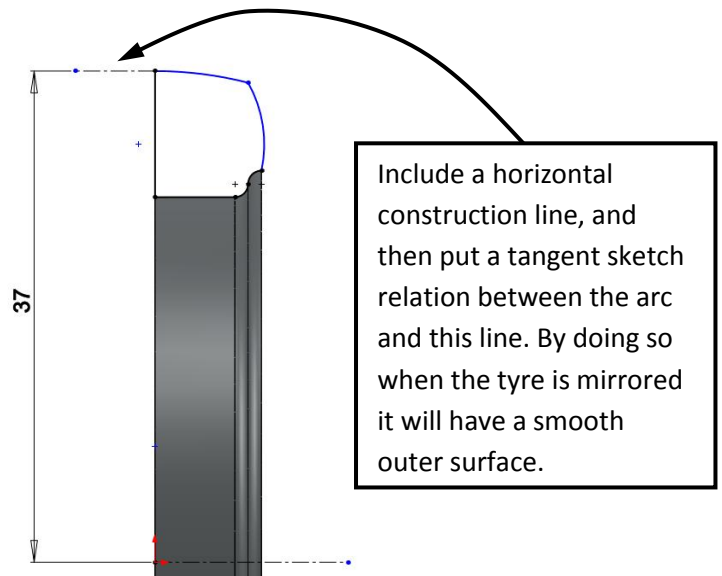
The black lines are those created by the Intersection Curve tool.



Complete rest of the tyre sketch as shown, using the Line and the 3 Point Arc tool in the Sketch tab of the Command Manager.

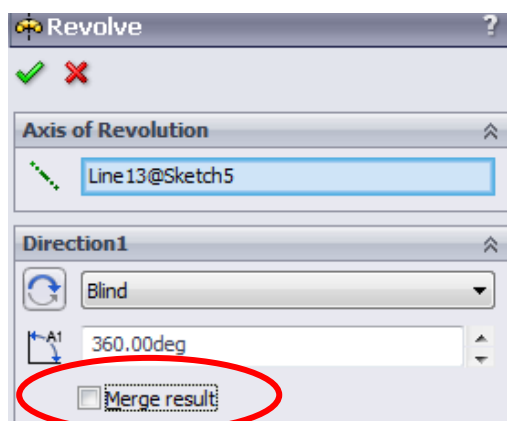


The 3 Point Arc tool works by clicking where you want the start of the arc, then where you want the end, and finally moving the mouse until the radius is what you want and then making the third click. For this exercise the dimensions are not important, just make it look like an inflated tyre profile.



Don't forget to include the centreline in the sketch so we can use this as the axis of revolution for the tyre.

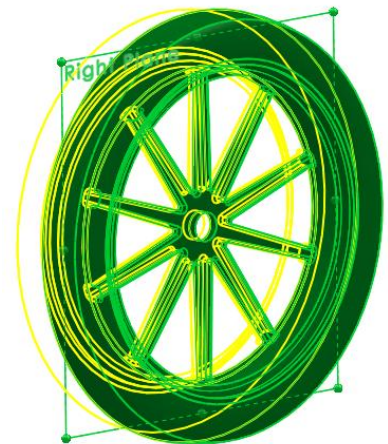
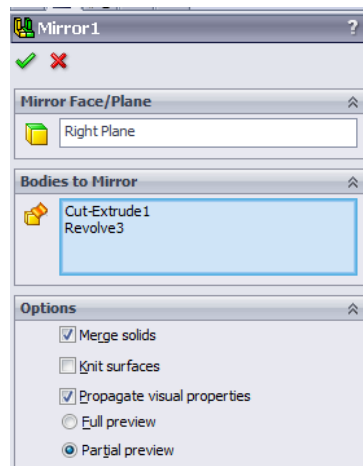
After exiting the profile sketch select the Revolved Boss/Base tool and select the sketch you have just created. By default you should see the Merge Result check box ticked, but in this case we want to have this switched off (click in the tick box to remove it). In the default condition the tyre revolve would be merged with the wheel and form part of the same body. We want it to be a separate body, and turning off the Merge result will provide this. Note that if there is a gap between features (they do not contact or intersect) then the Merge Result is automatically turned off because physically they cannot be merged if there is a gap between them.



Accept the Revolve feature by clicking the green tick, and to make sure it has gone as you expect check in the Feature Manager tree and see that there are now two bodies in the Solid Bodies folder.

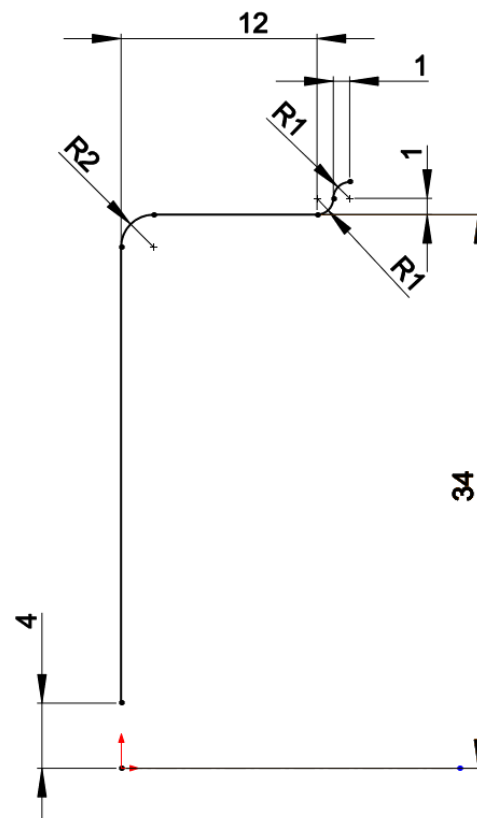
Before applying a mirror to both the tyre and wheel, sketch a circle of diameter 5mm on the Right plane and extrude cut this through the centre of the wheel so make the hole for mounting the wheel on the axle.

To complete the part, apply a Mirror feature as shown. In this case we want to mirror bodies so that all features are included, and there are two bodies to be selected. Make sure the Merge Solids check box is ticked, which will merge each body with its mirrored body. The result of this is as we want it, one complete wheel and one complete tyre. If the Merge Solid was off we would end up with four bodies in total.



The rear wheel is modelled in a similar process as the front, except the sizes are a different and there are no spokes. Because there are no spokes we will create the 'face' of the wheel in the same thin revolve operation used to create the rim.

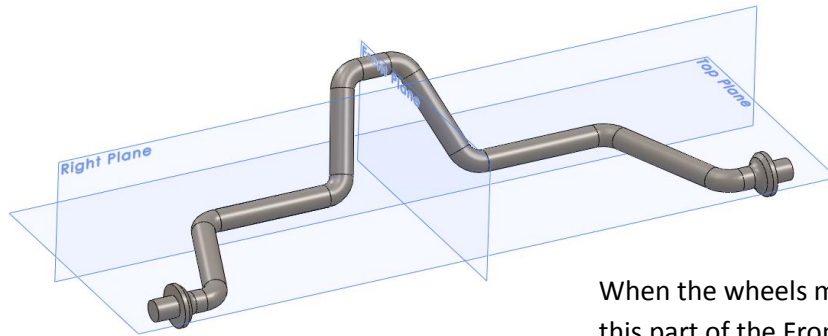
Create the sketch as shown and then revolve as a 1mm thin solid.



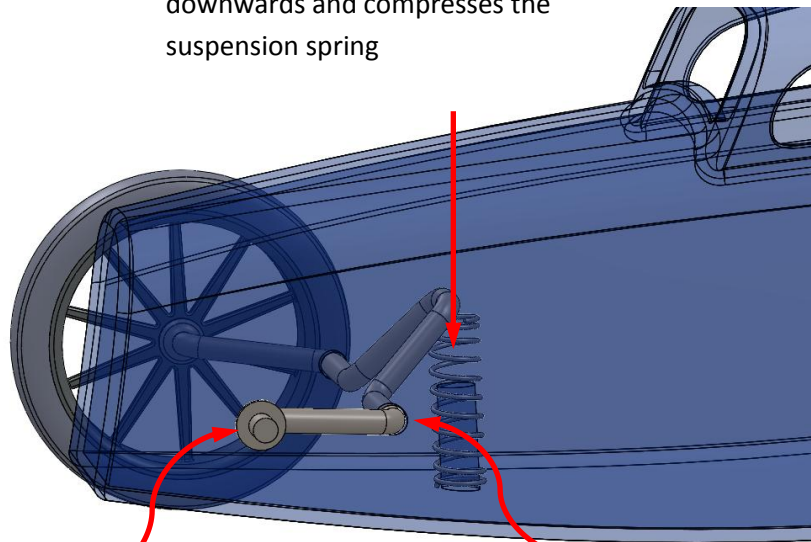
The rear tyre is modelled in exactly the same way as the front tyre.

Front axle modelled by sweeping along a 3D sketch path

The front axle consists of a steel rod that is bent in such a way that it mounts the wheels and provides simple front suspension. The suspension works by allowing the axle to pivot about its mount through the body, with a centre section of the axle pushes against a compression spring. On the ends of the bent rod we'll add some hubs to mount the wheels up against.

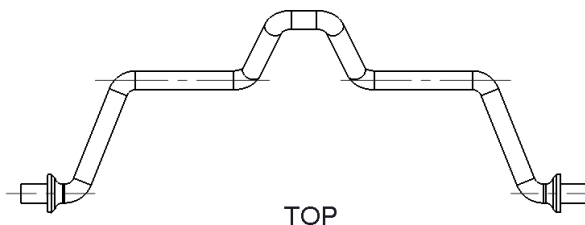


When the wheels moves upwards, this part of the Front Axle moves downwards and compresses the suspension spring

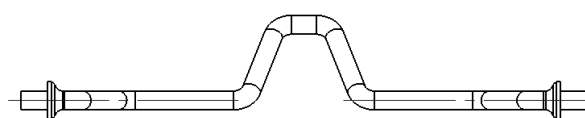


Wheel mounts on here so it can move up and down (as well as rotating!)

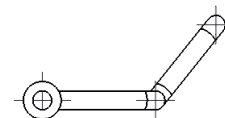
Front Axle pivots around this axis through the car body



TOP



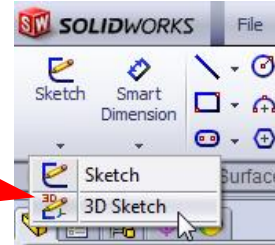
FRONT



SIDE

Geometry such as this, where there is a constant section along some path, occur frequently, such as in the case of pipes, ducts, tubes, cables, bent bars. Because of this there is a specific tool for creating these, called the Sweep. In its most simple form the Sweep tool requires only a profile sketch and a path to sweep this along, but there are also a lot of other controls that can be applied to change the resulting geometry.

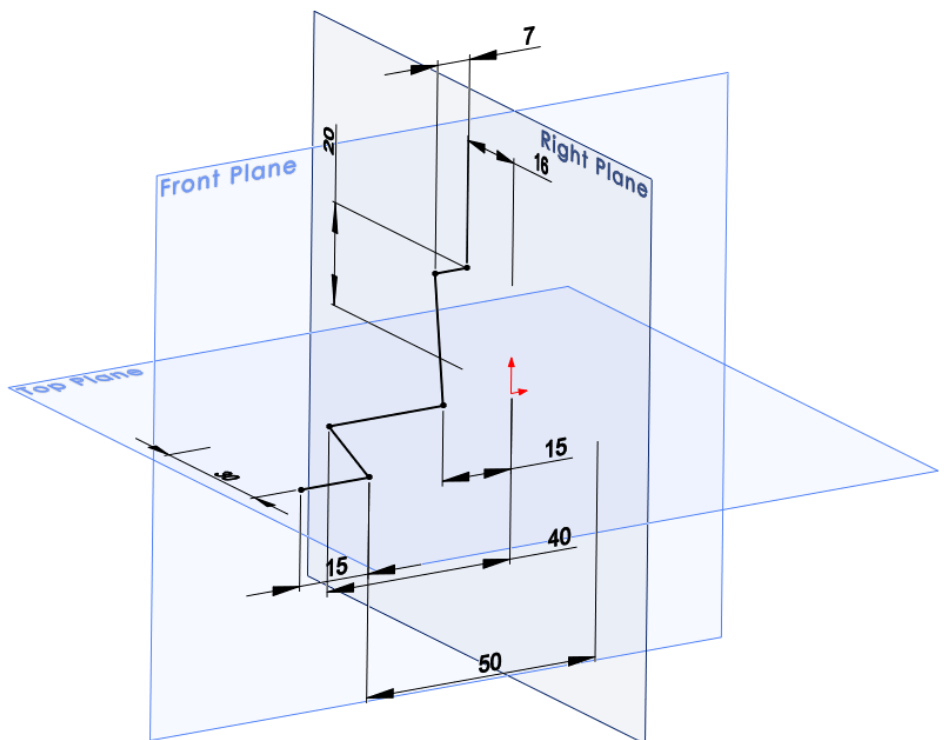
In our case we have a requirement to sweep a circle along a path. However the sketch path we want is 3D, in that unlike the previous sketches we have done it does not all lie on one plane. SolidWorks supports this by allowing us to **create a 3D sketch**, which works similarly to a 2D sketch, but allows lines to go in any of three dimensions.



Start a new part **using the UC Part template** and save it under the name Front Axle.prt

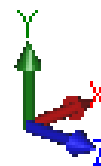
We will create a sketch as shown here. While 3D sketches are very useful, they can also be quite confusing to create, so we'll go through it one step at a time.

Create a new sketch by selecting the 3D Sketch tool in the dropdown from the Sketch tool in the Sketch tab of the Command Manager.



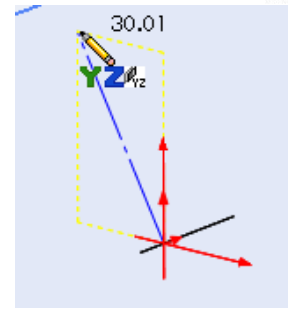
Notice down in the bottom left of the Graphics Area there is a triad showing the three axes of the model – these are important when 3D sketching.

Begin the sketch by selecting the Centreline tool so that we can create a little bit of construction geometry to help us position the start of the sketch. Just to make it easier to follow, orientate your model so it looks the same as in the example. With the Centreline tool selected you will notice that the pointer looks different to when you are doing 2D sketching. The pointer includes two letters that indicate the orientation of the plane that you are currently sketching on. The plane orientation can be changed by pressing the Tab key, to any one of YZ, ZX, or XY.

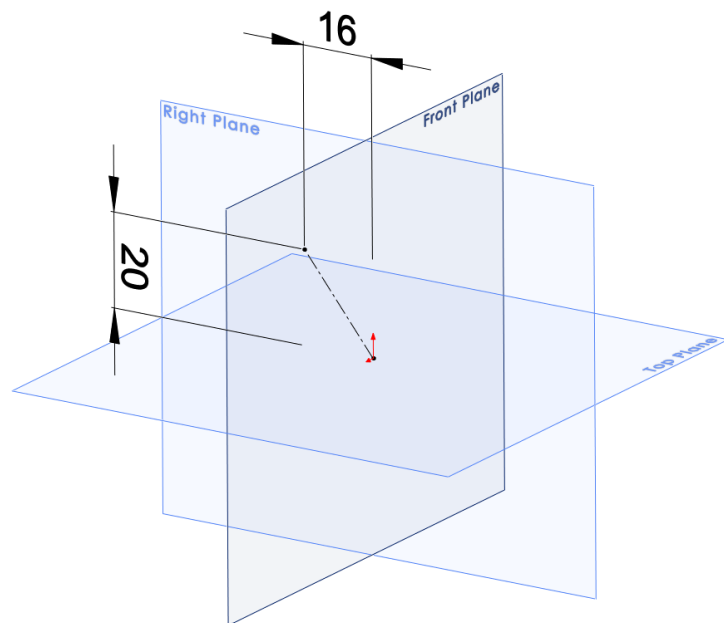


This is only a planning stage: read and follow the thought process but don't start modelling yet!

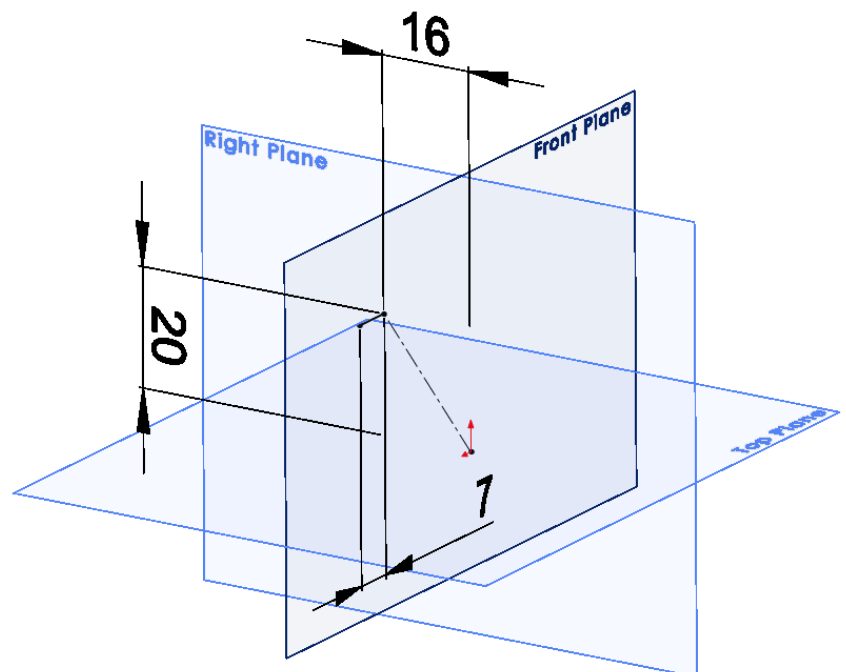
We want to draw the construction line **start at the origin** and moving upwards on a diagonal on the **right plane**, which is the YZ plane orientation. With the Centreline tool initiated, click on the origin and then press the Tab key until the YZ orientation is active. Click the second point on the screen with the line about 30mm long, and then press Esc to exit the Centreline tool.



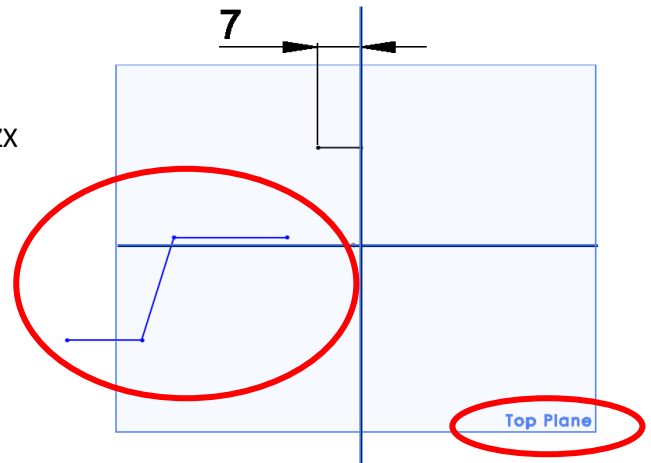
Use the Smart Dimension tool to apply the dimensions as shown. These dimensions are between the end point of the line and the Front and Top planes. To select the planes, make sure you select the edge (border) of the plane otherwise it will not be recognised. The construction line is on the Right Plane, but actually there is no relation in place to make it stay there, so select the line and the Right Plane and choose the On Plane relation. Notice when you do this the line will change from blue to black (fully defined). The end of this line now defines the start of the sketch that we want for the path of the Sweep.



Select the Line tool and draw a line starting at the end of the construction line and going along the XY plane orientation. Apply a dimension of 7mm to this.



The next lines we will create are all on the top plane (ZX orientation), so orientate the model so that the Top plane is almost parallel to the screen. This will make it easier to get the lines approximately the shape we want. Make sure the orientation is set to ZX (by using the Tab key) and draw the lines approximately as shown.

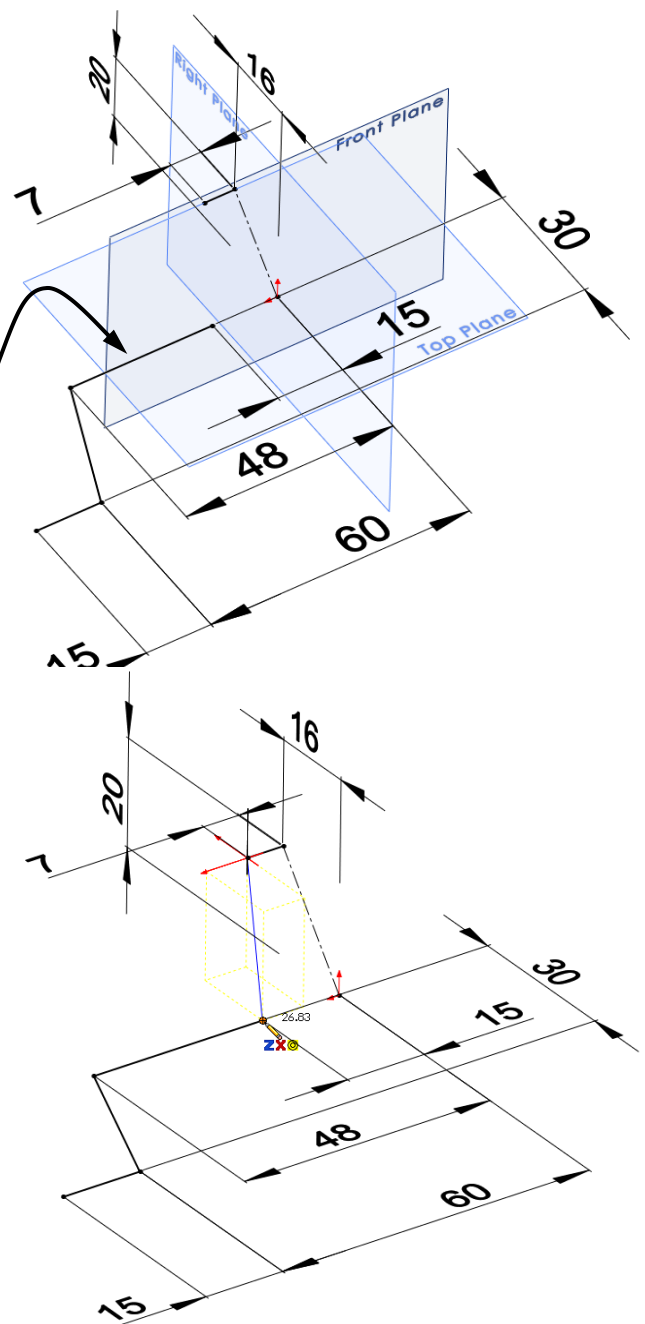


When first drawn all three of these lines are 'sitting' on the top plane, but are actually completely unconstrained in 3D space, so we need to apply some sketch relations and dimensions:

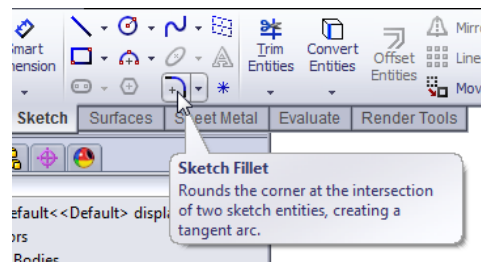
- **On Plane** between each line segment and the Top Plane
- **On Plane** between this line and the Front Plane
- Dimensions as shown

When creating the dimensions, if they are not generating on the plane you want them to, press the Tab key before clicking to set the position of the dimension line. Just like selecting the plane orientation when creating the lines, pressing Tab during the dimension creation scrolls through the options for orientation of the dimension line display.

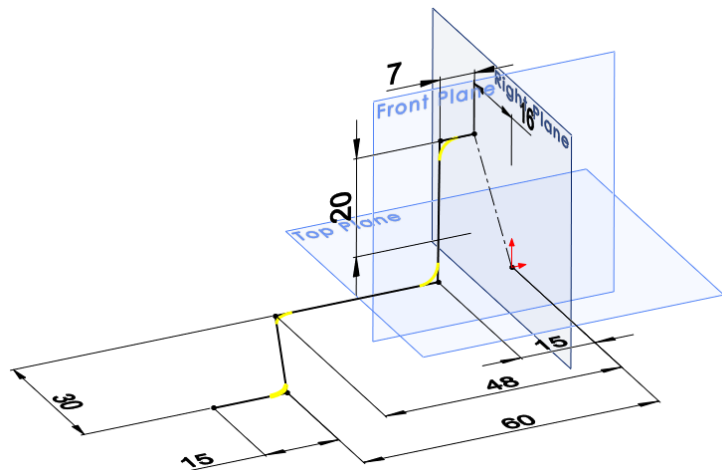
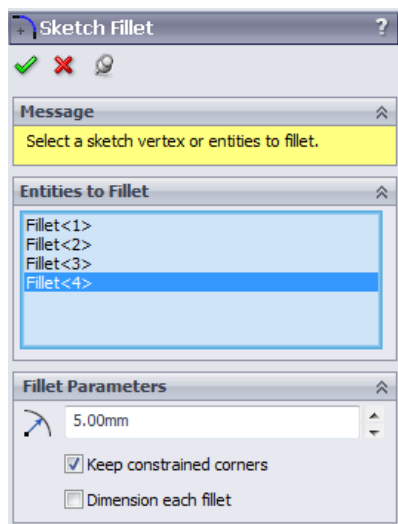
Finally put the last line in, which joins the end of the 7mm line with the lines we sketched on the Top plane. Make sure the line snaps to the end points (look for the orange highlighted dot).



Because we want to model a metal rod that has been bent, we need to have a bend radius at the joins between each of the lines. This can be done easily using the Sketch Fillet tool found in the sketch tab of the Command Manager.

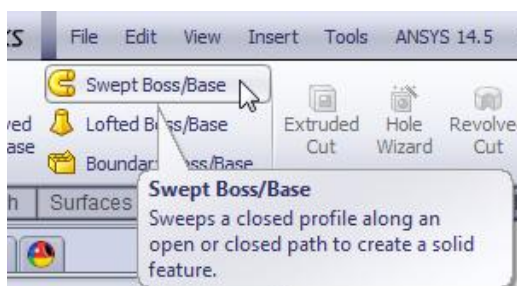
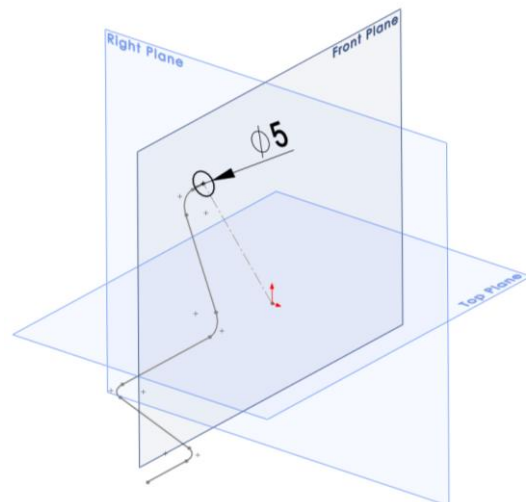


Set the fillet radius to 5mm (but no need to type the mm) and then click on each of the corners where two lines connect in the Graphics Area.



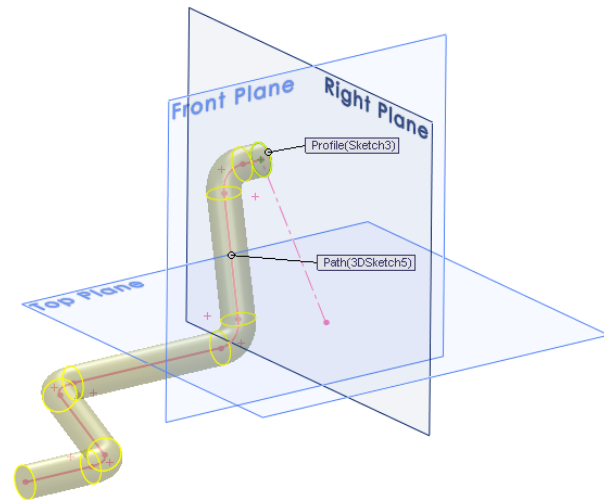
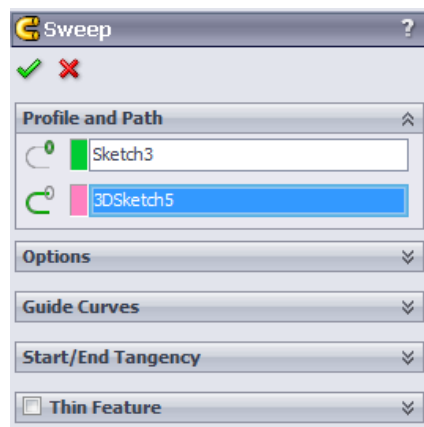
Make sure the **Keep constrained corners** check box is ticked as shown, otherwise the dimensions that you have created to the corner points will be deleted and the sketch will be under constrained.

The path for the sweep is now completed, so exit from that sketch and create a new sketch (**standard 2D sketch this time**) on the Right plane. Draw a circle with the centre on the end of the 7mm line, making sure that you click only after seeing the orange dot appear which indicates that a sketch relation will be applied. Use the Smart Dimension tool to apply a dimension of diameter 5mm. Exit the sketch.



Now we have a path sketch (the 3D one) and a profile sketch (the circle), we can use the Sweep tool to create a solid with them. Select the Sweep tool from the Features tab of the Command Manager.

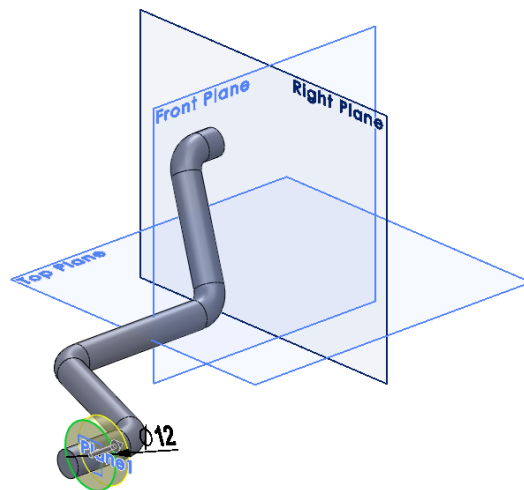
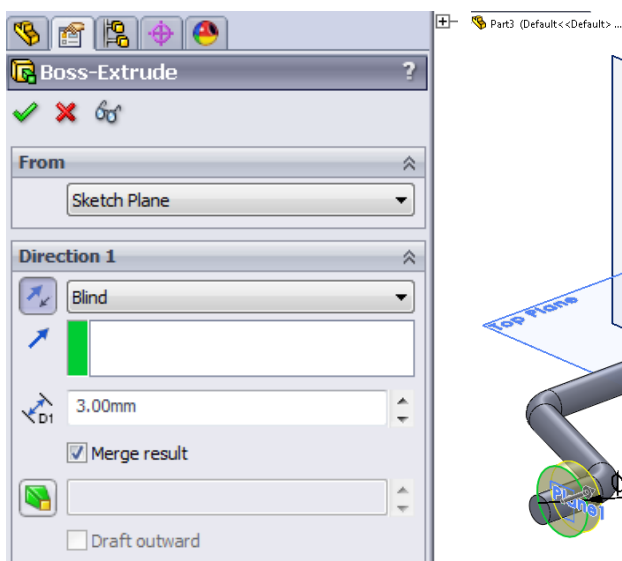
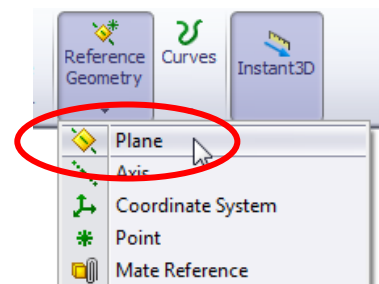
Click in the top selection box in the Sweep Feature dialogue and then click on the circle sketch in the Graphics area. After this is selected SolidWorks automatically changes so that it is looking for the path sketch, so select one of the lines in the path sketch. A yellow preview of the geometry should appear as shown. Click the green tick and the solid will be generated.



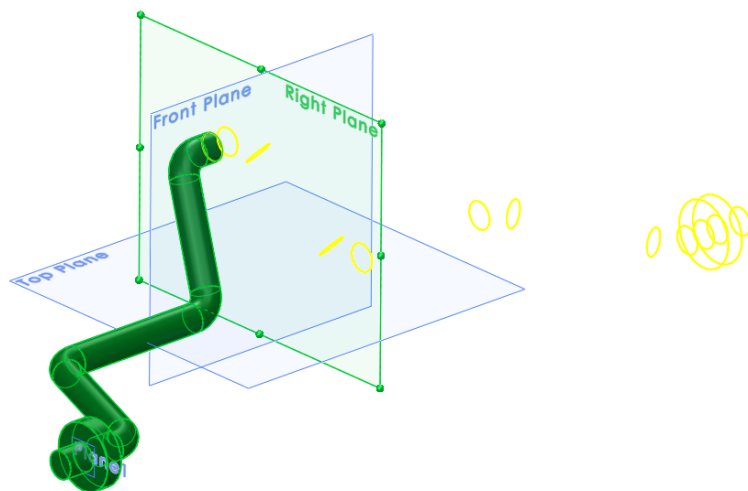
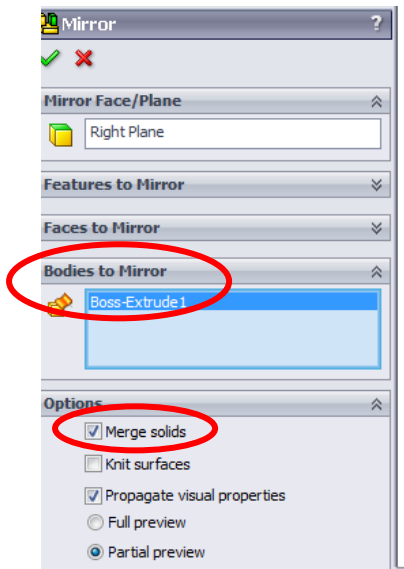
Notice this time that both the path and the profile sketches have been 'borrowed' by the sweep feature, so have apparently disappeared from the feature tree. If you click on the + symbol next to the Sweep feature you can see that the sketches are still there and can be accessed if you want to go back into them to modify them.



To create the hub on the end of axle, offset a plane 8mm from the end face using the Plane tool in the Reference Geometry dropdown in the Feature tab. Sketch a diameter 12 circle on this plane, concentric with the axle, and then extrude a sketch 3mm.



To complete the part, apply a Mirror about the Right plane. The mirror feature can apply to faces, features, and bodies. In this case we want to make sure that it is the *body* we are selecting, as this supports our design intent that everything before the mirror feature will be copied onto the other side. For any additions we want to make to the part we would just have to roll back before the mirror to add it in and then when rebuilt the mirror would automatically include it. If we only mirrored the sweep and extrude features then any subsequent additions would need to be manually added into the mirror.

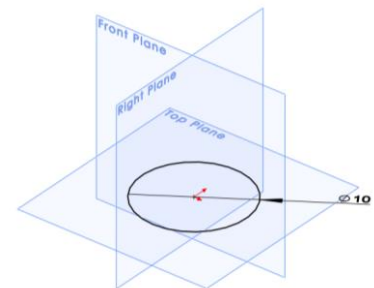


Suspension Spring

Coil springs are a common component in products and mechanical systems, and fortunately SolidWorks provides a way to model these.

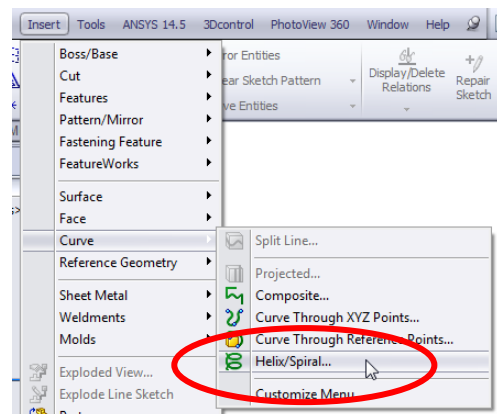
The spring is created using a Sweep feature as we have done previously, but the difference here is that the path is a helical curve.

To generate this curve start by creating a sketch on the Top plane with a diameter 10mm circle centred on the origin. This circle will define the diameter, to the centre of the spring wire, of the coil spring.



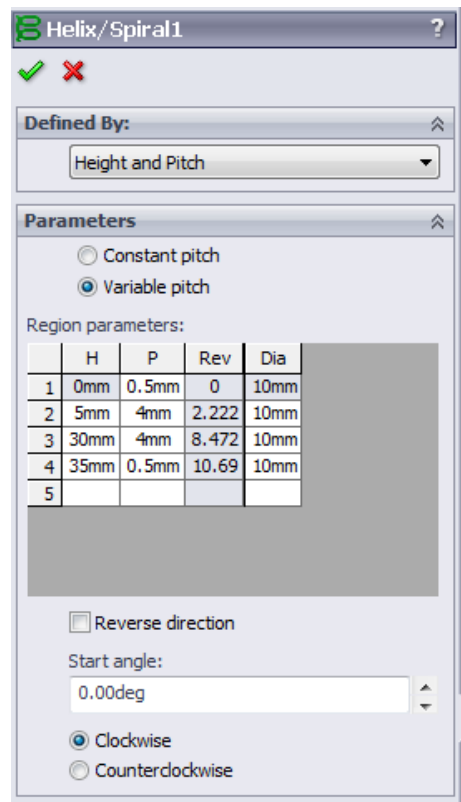
Exit the circle sketch and select the Helix/Spiral tool from the Curve submenu in the Insert menu on the Main Menu Bar.

As requested, select the circle sketch on from the Graphics Area to define the start location and diameter of the helix.

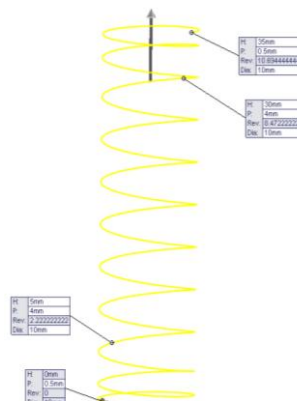


There are a number of different ways to define the helix to suit different situations, but in our case we set the definition type to Height and Pitch. Using this approach we define what the pitch (mm of spring length per revolution) is at various point along the length of the spring. The pitch between these point will transition smoothly between each of these values.

As is often the case with coil springs, we want our spring to have a smaller pitch (tighter coil) at the top and bottom. This makes it easier for the spring to 'seat' against whatever it is mounted with. The centre section is a larger pitch to allow the spring to have a greater range of movement. To achieve this enter in the values as shown and you should see the yellow preview helix appear with a tighter helix at each end than in the middle.



Click on the green tick to finish the curve is blue in color, sketches appear (when not and sketches are different something that you create and curves are created tool such as the Helix/Spiral used in a similar way, as we will place of a sketch for the sweep

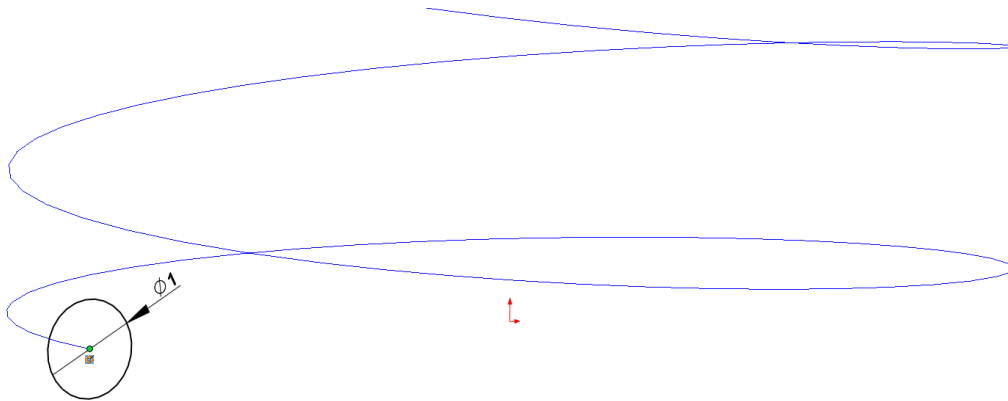


In addition to the path for the the Right plane and draw a circle close to the end of the helix curve and then apply a Pierce relation between the centre of the circle and the helix curve. There are a couple of tricks to doing this. Firstly sometimes it can be difficult to select the helix in the Graphics Area (the pointer just doesn't recognise the curve), so instead select the Helix in the Feature Tree. Selecting an entity in the Graphics Area has exactly the same effect as selecting it in the Graphics Area, and sometimes it is easier and more reliable.

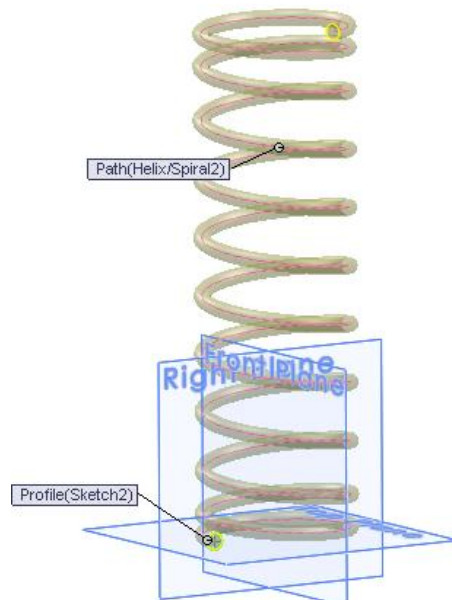
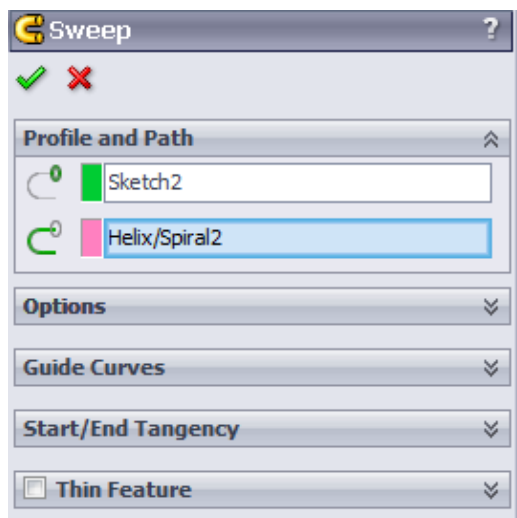
accept the helix curve. Notice how the rather than the default grey that editing them). This is because curves things in SolidWorks. Sketches are directly in the sketching environment, indirectly using some other creation too. Regardless of this the two can be do here with the helix being used in path.

sweep, we need the profile. Sketch on circle of diameter 1mm. Locate the

The second trick is that applying the Pierce relation could attach the circle to the helix on any one of the coils that intersects the circle sketch plane – it may not necessarily go onto the end of it. To ensure that it does go on the end you need to zoom in really close on the end of the helix before you select it. When there are multiple possibilities for how a relation could be formed, SolidWorks will go for the one that is currently dominant in the Graphics Area.

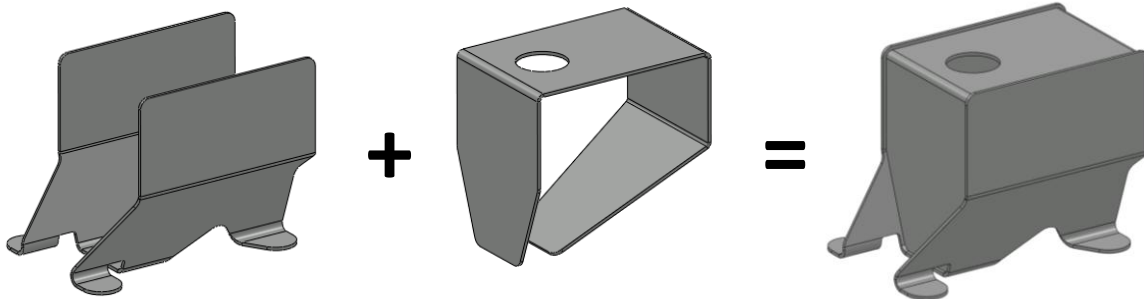


With the path and profile now completed, use the Sweep Boss/Base tool from the Features tab of the Command Manager to generate the coil spring.



Fuel Tank

The fuel tank is going to be manufactured from two pieces of folded sheet metal (brass) that will be soldered together. It is possible that it could be designed as a single piece, but in cases such as this it is often easier overall to have two pieces that are easy to bend than one piece that is very difficult. Also for our tutorial having two pieces gives us the opportunity to introduce two different workflows for creating a sheet metal component in SolidWorks.



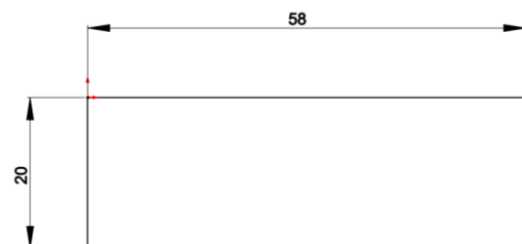
Sheet metal components, in the real (non CAD) world, start out as a flat sheet and then have various forming operations applied, such as bending, stamping, and pressing. In CAD software we can replicate this process as in the real world; start from flat and then modify, or sometimes it's easier to model the end result and then allow the software to flatten this out. In either case it's the flat pattern for the finished part that is a vital input for manufacturing.

In the past, because sheet metal CAD modelling uses a range of specialised tools, these parts were completely different and separate from 'standard' solid models. More recently sheet metal modelling capability has been incorporated into 'standard' models so a combination of solid modelling and sheet metal modelling techniques can be applied on the same part. In SolidWorks sheet metal capability is fully integrated so a sheet metal part starts out exactly the same as any other part.

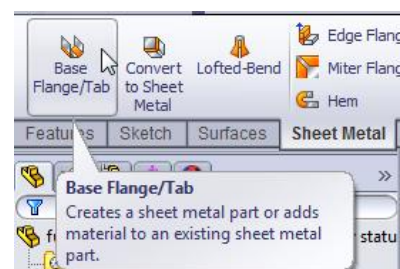
Open a new part and save it as *fuel tank top.prt*

Make sure that the Command Manager has the Sheet Metal tab present, and if it doesn't then right click on one of the tabs that are there and select Sheet Metal in the dropdown menu.

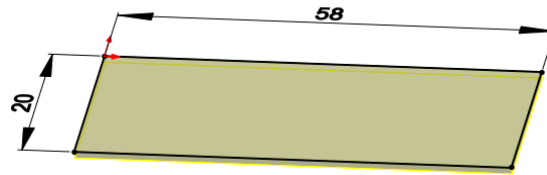
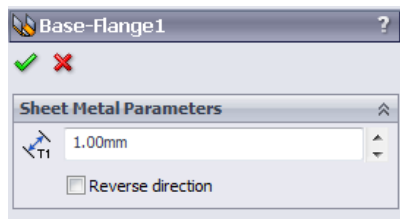
For this part we will use create a part of the sheet metal first, and then add extra pieces one feature at a time. Note the origin in the top left of the sketch rectangle. Create a sketch on the top plane and dimension as shown.



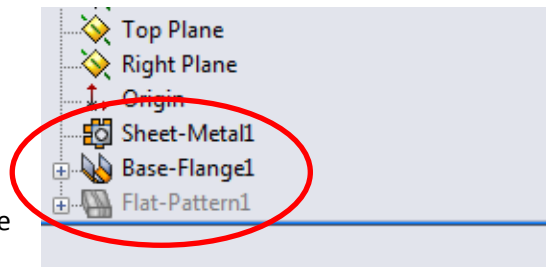
Exit this sketch and then select the Base Flange/Tab tool.



Select the rectangle sketch, and enter a value of 1mm for the thickness. Click the green tick to generate the first piece of sheet metal.

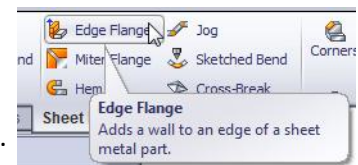


Usually when we create a feature that feature will appear in the Feature Tree, and this is the case here too, but there are two additional features that are also added. Base-Flange1 is the sheet metal part we just created, containing the rectangle sketch within it. The Sheet-Metal1 feature is automatically inserted to establish that this is a sheet metal part, and contains various parameters that determine the forming behaviour of the material. The third feature is the Flat-Pattern1, which is greyed out due to being suppressed by default. If this is unsuppressed it will unfold the sheet metal part into a flat blank. All subsequent features we create will automatically sit above this feature so that it is always the last feature in a sheet metal part.

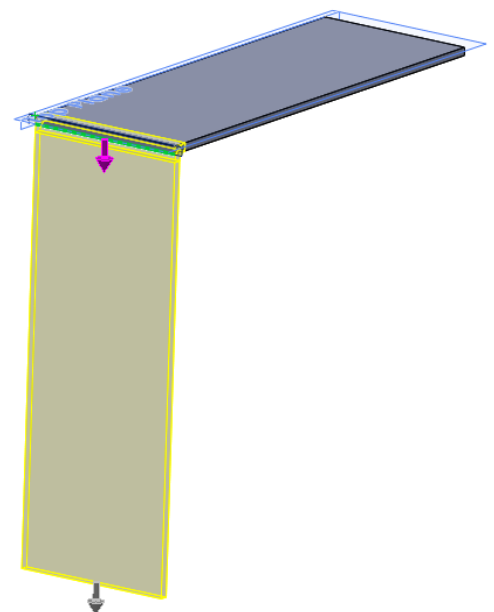
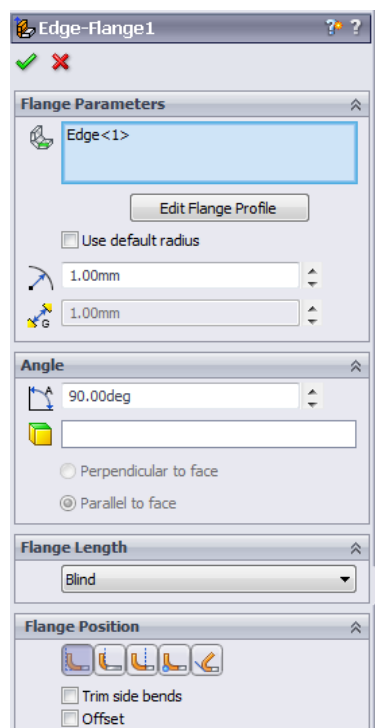


Based on our first feature, SolidWorks now knows this is a sheet metal part so we can use other sheet metal features to build upon it.

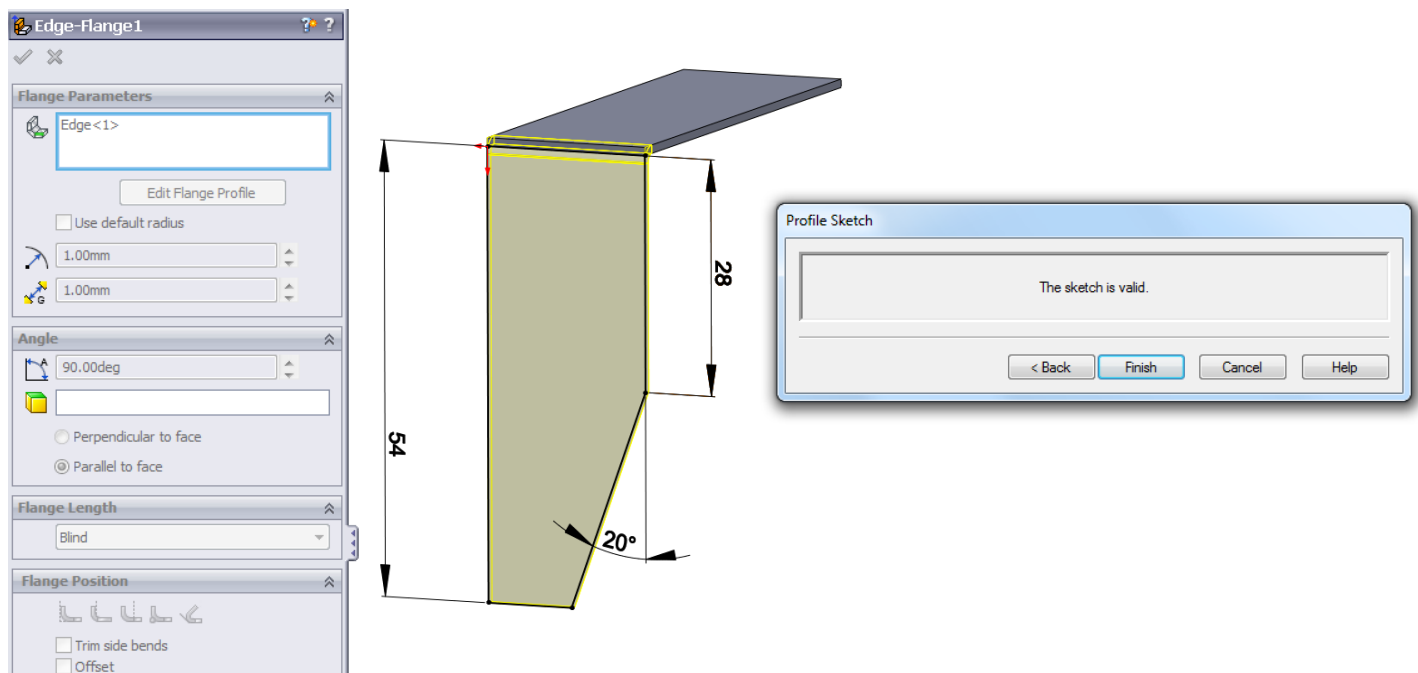
Select the Edge Flange tool from the Sheet Metal tab in the Command Manager. This tool is used to create additional pieces of material that are joined, via some bend angle, onto another piece of sheet metal at its edge.



Enter the values as shown in the example and then select the edge that is highlighted in green. You may notice that that by default the flange that is created is a simple rectangle, and the example has a corner cut off it. The flange can be any shape we like by clicking on the Edit Flange Profile button and modifying the default sketch.

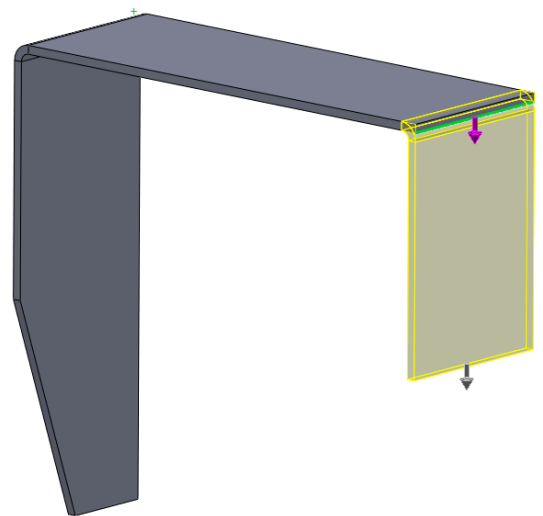
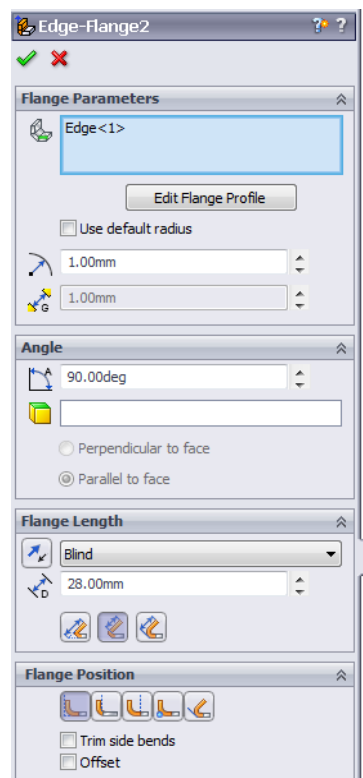


For this flange the sketch is as shown. Even though this is a sketch created within a sheet metal feature, the sketch behaves no differently to any other so you can use the standard sketch tools which you are familiar with.

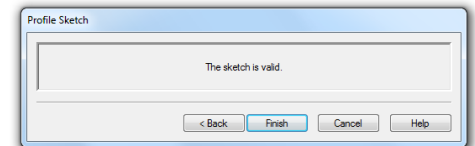
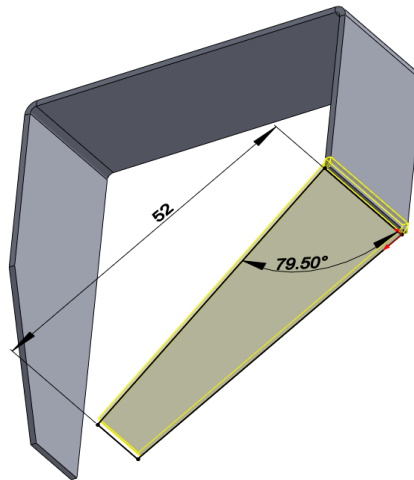
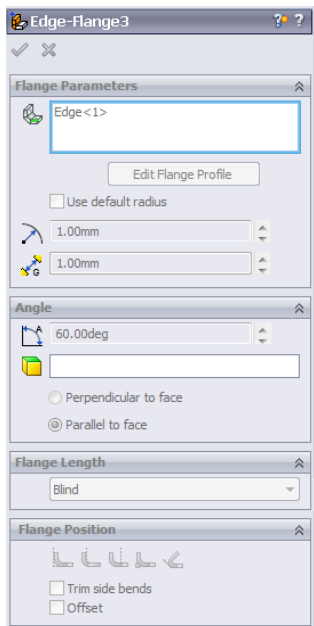


While in this sheet metal sketching mode there is a popup window that monitors the validity of the sketch for creating the sheet metal flange. If you perform a modification that is not possible for the software to build then it will advise you of this. When completed click on the Finish button on the popup, and then click the green tick on the Edge Flange dialogue.

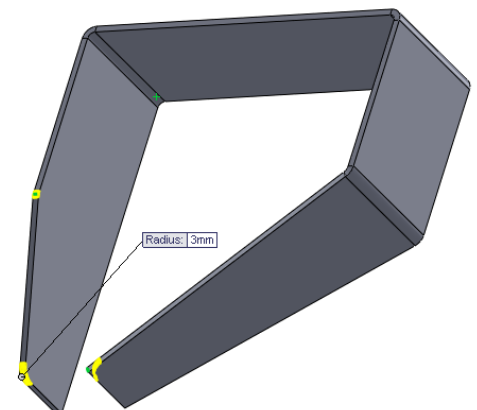
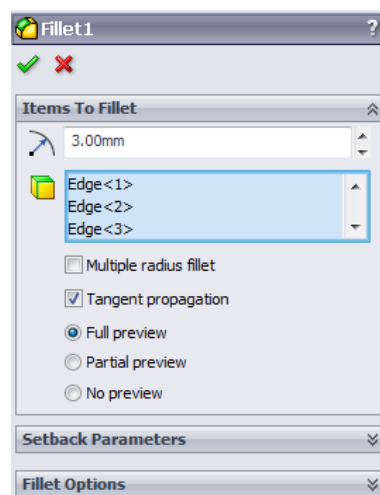
Create another flange, 28mm long, on the other end of the base body. This time, because it is just a rectangle that we want, there is no need to edit the flange profile.



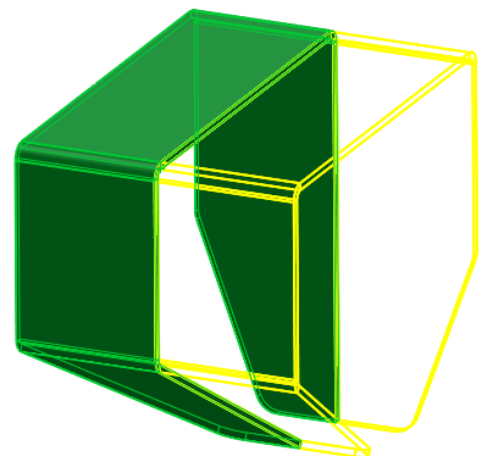
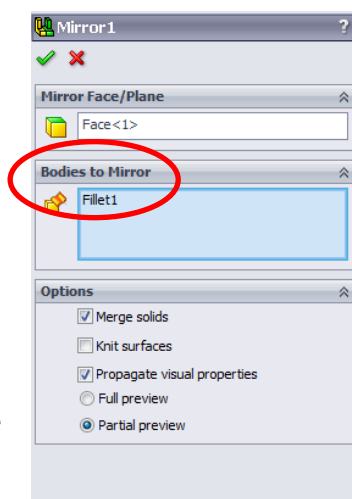
The final Edge Flange is created on the bottom edge of the previous Edge Flange. Create this as shown in the example, with a bend angle of 60 degrees, and edit the flange profile also.



Apply 3mm fillets to the part on the three edges shown. The fillet tool works exactly the same on this sheet metal part as for any other solid model.

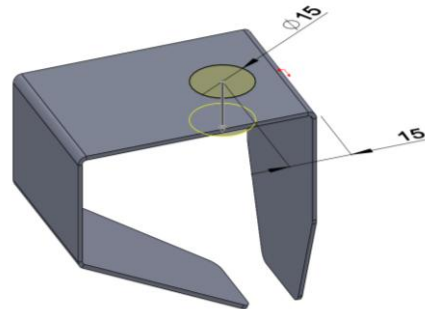


As we should always do when dealing with symmetric parts, so far we have only modelled half of the component. Apply a mirror feature to this go make the piece whole. Similarly to other mirror features we can choose to mirror either Faces, Features, or Bodies. Here we want to do the complete body, so select the body

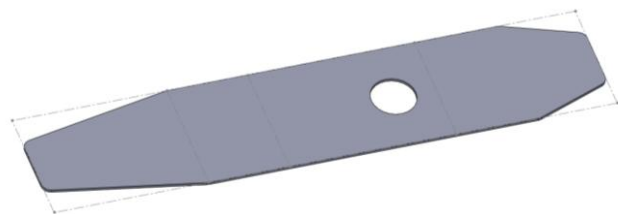
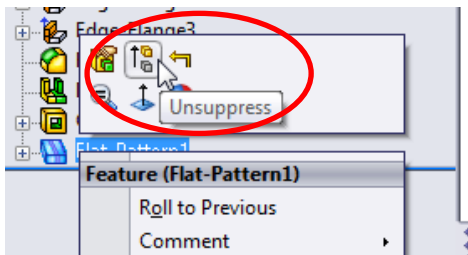


selection box and then on the part in the Graphics Area. For the Mirror Face/Plane you must select the face of the part that we want to mirror about. For other parts we usually select a plane to mirror about, but SolidWorks will not allow this for sheet metal components.

To complete the top part of the fuel tank, extrude cut a circular sketch from the top surface of the part, so this can later be used to include a fuel filler neck.



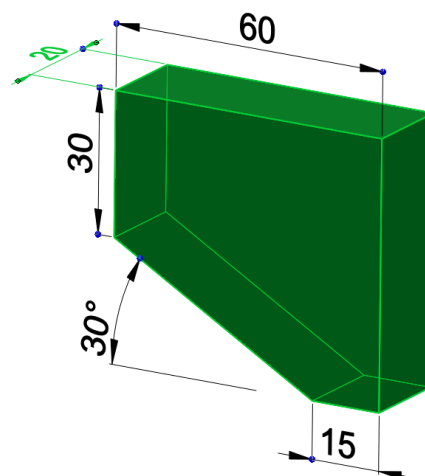
To understand the most useful capability of sheet metal models, click the left mouse button on the Flat-Pattern1 feature and select **Unsuppress**. This allows this feature to generate a version of the part with all of the bends flattened out. This is the shape is called the blank, and is the shape that is cut out of sheet metal as the first step for manufacturing the part.



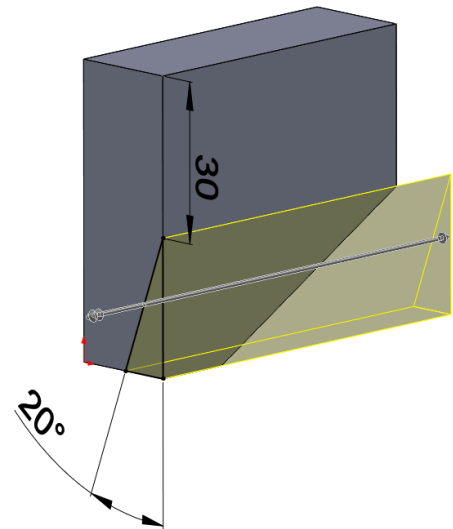
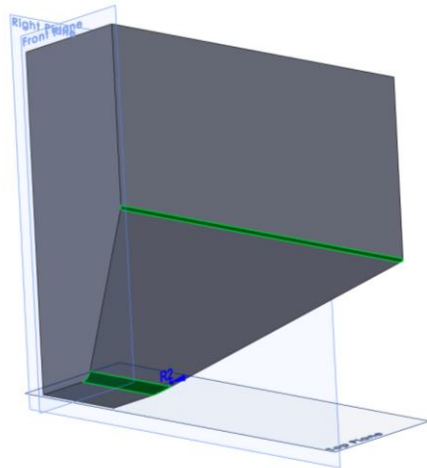
The part can be changed back to the original shape by right clicking on the Flat-Pattern1 feature again and selecting the Suppress icon.

For the bottom part of the tank, which is a slightly more complex shape, we'll use a different technique to make it as easy as possible and demonstrate a useful workflow. The approach is to create a solid model block which has the general shape we want in the final sheet metal part, and then use the *Convert to Sheet Metal* function to change it into sheet metal

On the Right plane create a sketch as shown and extrude it 20mm in one direction.



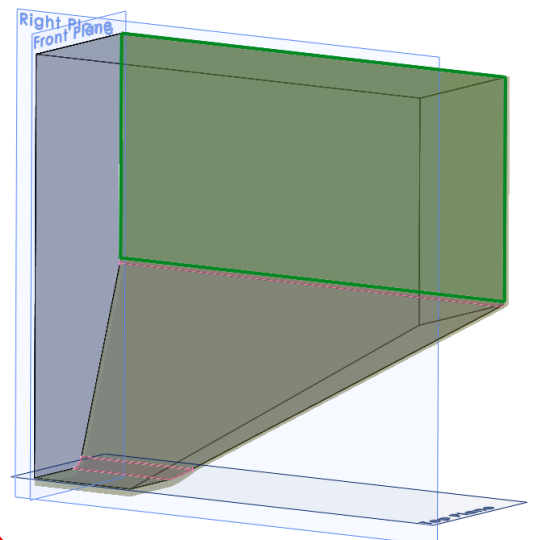
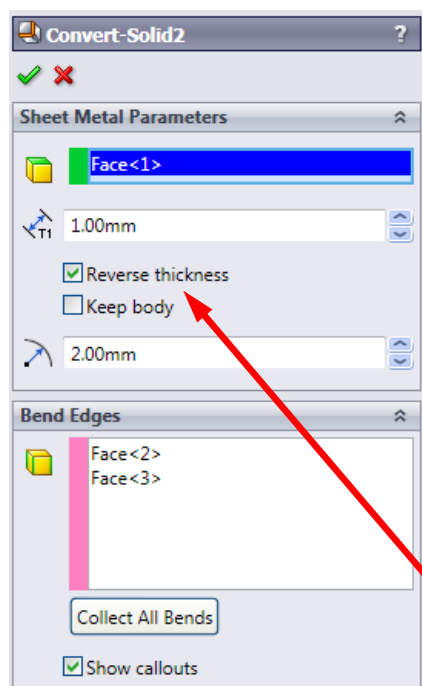
On the Front plane sketch a triangle and extrude cut it through the part as shown.



Apply 2mm fillets on the corners shown.

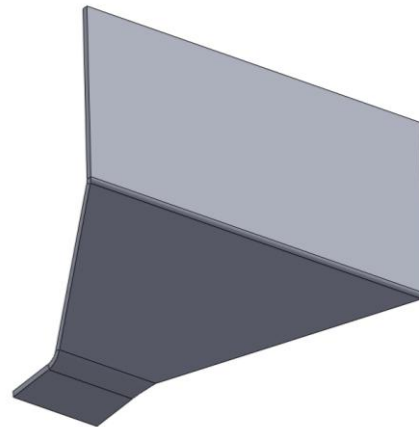
The body we have just created is now ready for creating our sheet metal part from, so select the Convert to Sheet Metal tool from the Sheet Metal tab of the Command Manager.

In the top selection box, select the face in the Graphics Area that is shown shaded green, make sure the thickness is set to 1mm and the bend radius to 2mm and then click the *Collect All Bends* button. This searches all around the first face to find any faces that join onto it with a fillet, and then adds these to the list, searching in turn around those new faces also. In our case, because of the fillets we put on the solid, it will find two additional faces to join our original.



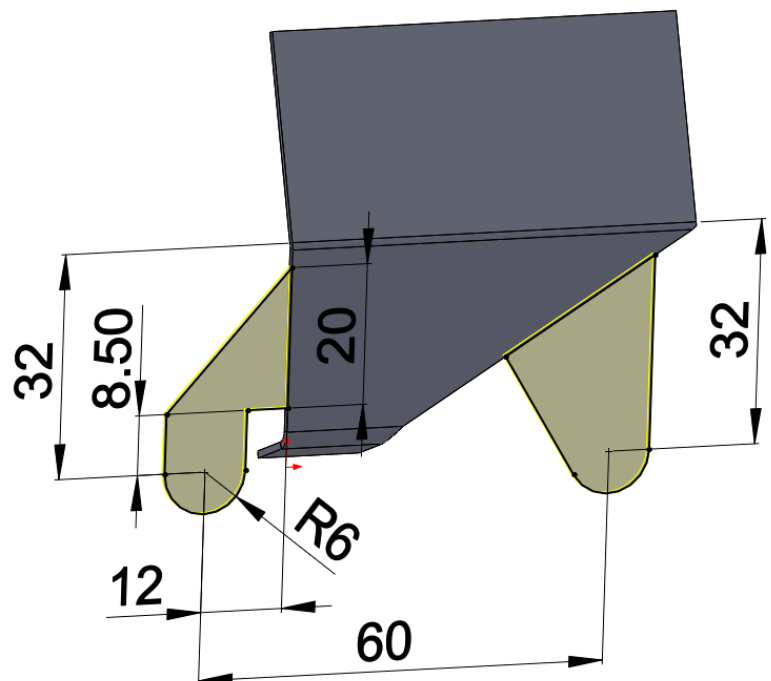
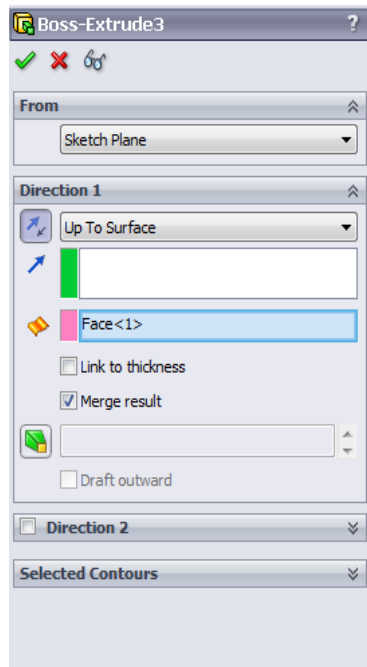
Make sure that the yellow preview of the sheet metal part that is about to be created is coming outwards from the original block. If this is not the case, click on the Reverse Thickness check box just below where the 1mm thickness was specified.

Click on the green tick and the sheet metal part will be generated. Similar to before, because this feature defines the part as being a sheet metal component, SolidWorks includes the Sheet-Metal feature and the Flat-Pattern feature automatically. The resulting sheet metal part should look as shown.

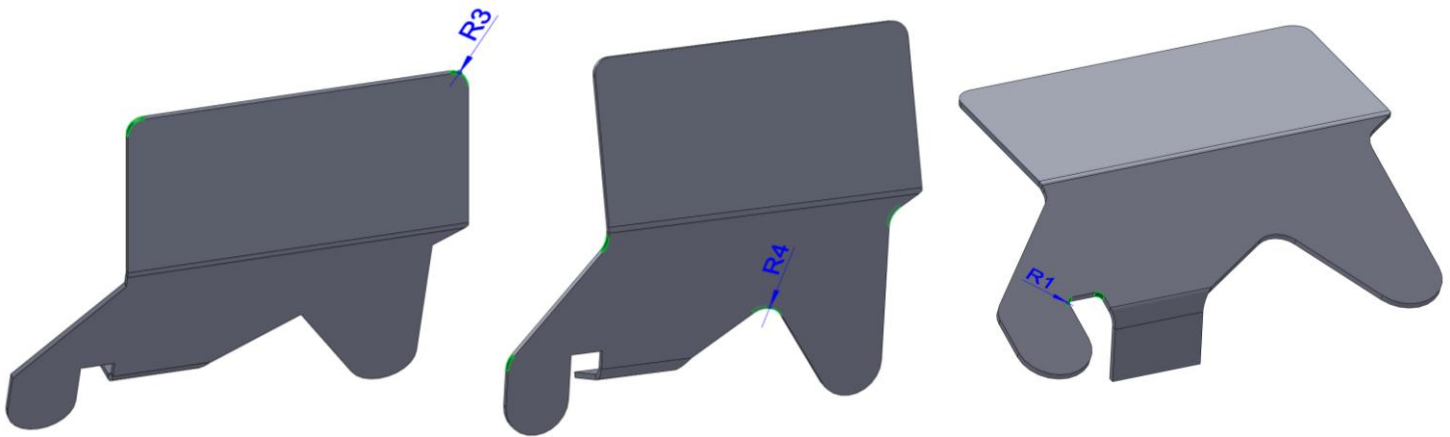


The fuel tank needs some mounting lugs on it, and it is going to be easier to integrate these into the tank itself rather than join them on afterwards. In doing so we'll see a great example of seamlessly integrating 'standard' modelling tools into a sheet metal part.

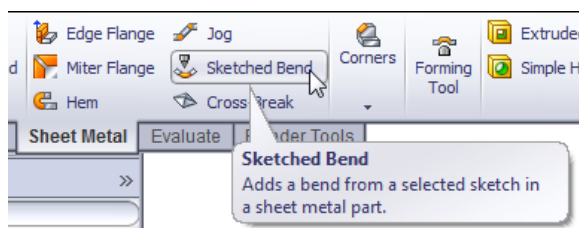
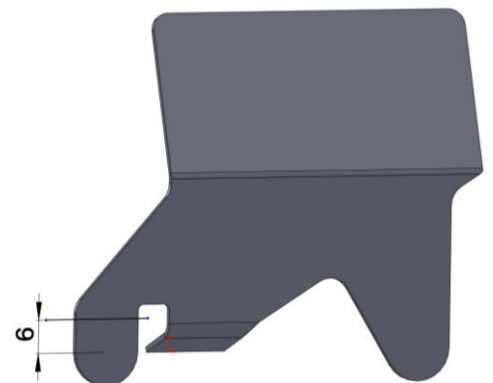
On the angled face of the sheet metal part, create the sketch as shown and extrude the profile. The intent is that the centre of the two arcs have a horizontal sketch relation between them. In the extrude feature, select the Up to Surface option from the dropdown list, and choose the inner face of the angled sheet metal as the target face. This will ensure the extrude matches the thickness of the sheet metal, even if we were later to go back and alter the thickness.



Apply three Fillet features as shown to soften the top edge and blend in the lug features.

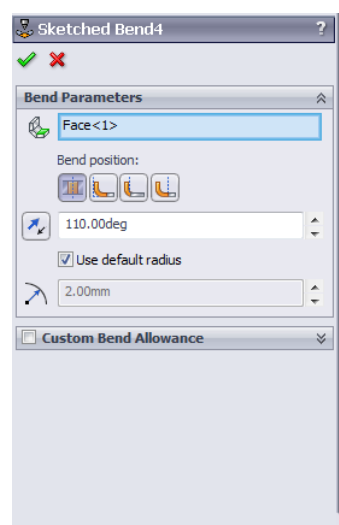


To apply a bend to an existing piece of sheet metal, rather than creating the bend when adding more material, we use the Sketch Bend tool. Before selecting this, create a sketch (ie just a line) as shown on the part face with the new lugs on it. The ends of the line are not important, as long as the line extends over the edges of the part (which we are about to bend).

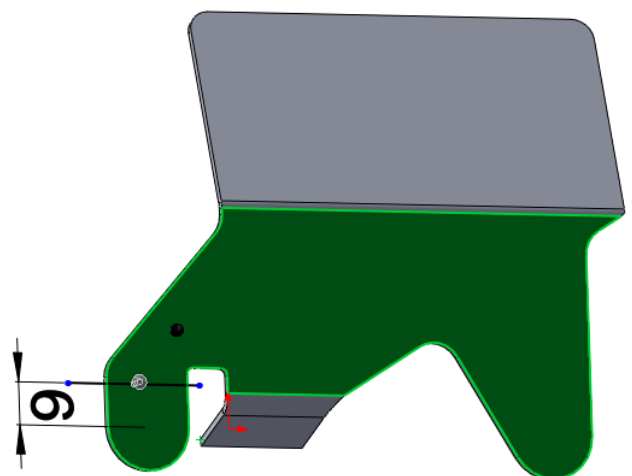


Exit the sketch and select the Sketch Bend tool from the Sheet Metal tab of the Command Manager.

Select the line sketch in the Graphics Area and the Sketch Bend feature dialogue will appear. The tool is going to apply a bend along the line of the sketch, but it needs to know which side of the line is going to move and which side will stay where it is. This is determined by where you select the face



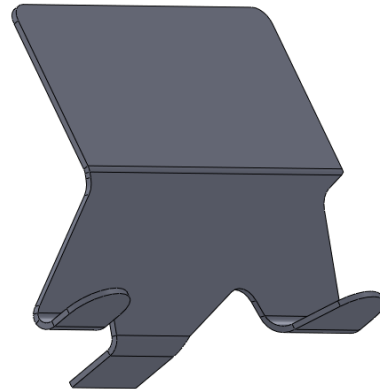
highlighted in green. The side of the line that you click on to select it is the side that will be stationary. SolidWorks records the selection location with a black dot as you can see in the example.



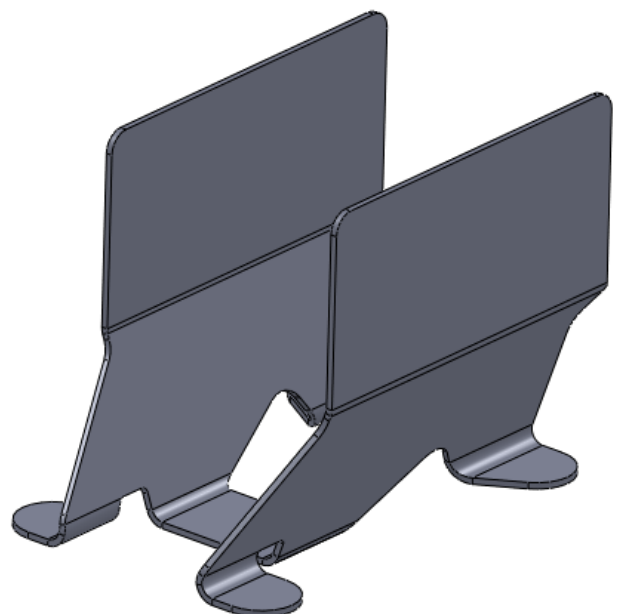
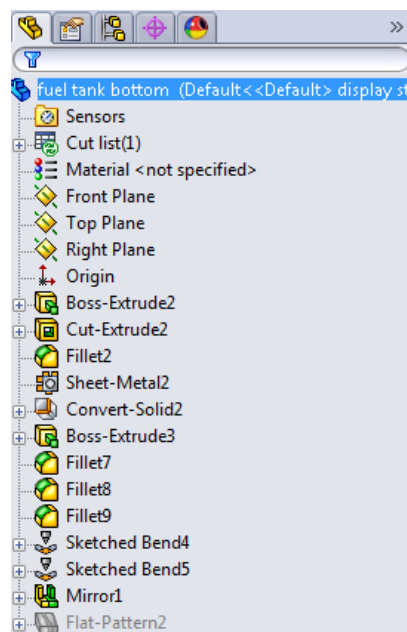
If you want to change which side moves you need to delete the face from the selection box and select it again by clicking on the other side of the line. Enter in a bend angle of 110 degrees and ensure the Bend position option is as shown.

Repeat with another Sketch and Sketched Bend on the other tab, with this bend in line with the first.

The result of the two tab bends is as shown.



Finally, apply a mirror on this body. Remember that with sheet metal parts the mirroring plane needs to be defined by a face on the part. The finished part and Feature tree looks as shown.



Unsuppress the Flat-Pattern feature for this part and check the shape of it. It is quite a complex shape that would be difficult to develop using manual techniques.

