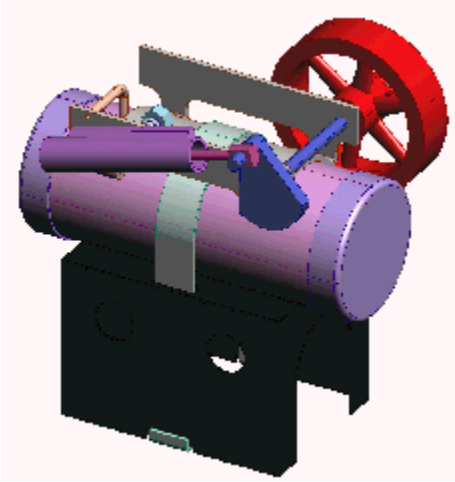
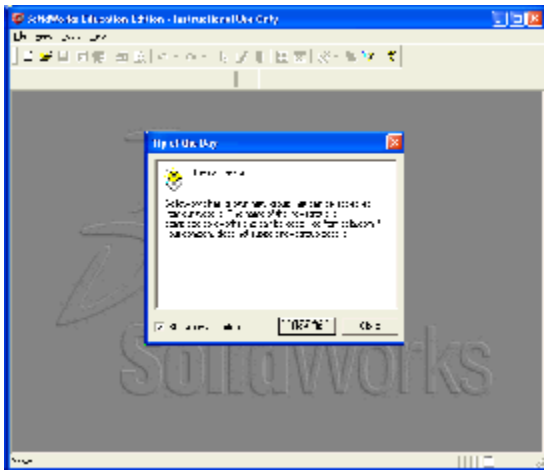


Introduction to SolidWorks

SolidWorks is a powerful 3D modeling program. The models it produces can be used in a number of ways to simulate the behaviour of a real part or assembly as well as checking the basic geometry. This tutorial guides you through construction of the model steam engine shown here. First you'll learn the basics of creating solid features needed to build the major functional parts and assemble them. In later sessions you'll generate engineering drawings and experiment with animation and 'photo-realistic' rendered views too. This should give you the knowledge needed to create more complex designs as you explore the enormous functionality of SolidWorks. Make sure you try the tutorials and other help topics under the Help menu. It's easy to learn how to use SolidWorks fully by following the tutorials supplied with the program.



Start SolidWorks

Start SolidWorks by clicking on the SolidWorks icon in the opening window. Our network doesn't let the program write files to the normal location the program expects, so when warning messages appear, cancel them or browse to your H: drive if asked for a file loation. Eventually, you'll see the screen shown here, or something like it. If you get the 'Novice User' box just cancel it. There may be a 'tips' window too, it's worth reading these as you get to know the software. Shut down the tips window for now.

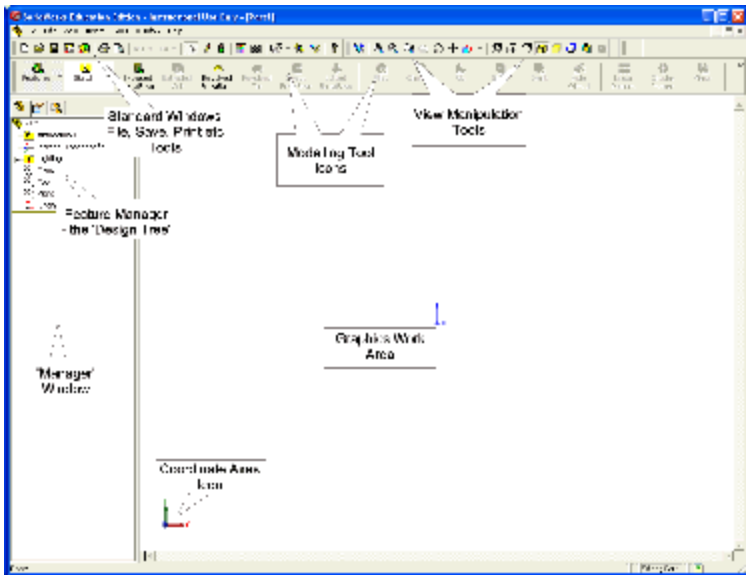
In SolidWorks we can build ***Parts***, ***Assembies*** and ***Drawings.*** We're going to start by creating a new part. Later we'll build that into an assembly and then create an engineering drawing of the whole thing.

1. Click on **File,** move the cursor over **New**.

2. A dialog box appears, listing the types of document you can create.

3. Move the cursor and click on **Part**.

Your screen should look something like that shown here now.

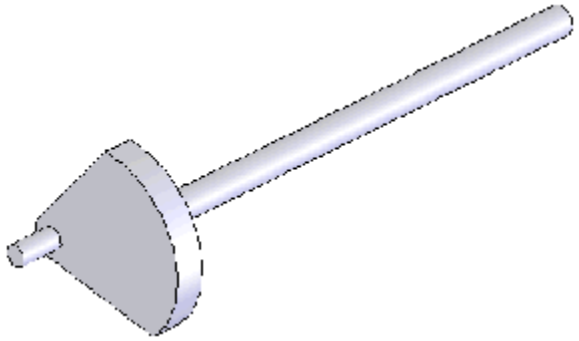


The toolbars at the top of the screen may also be in slightly different positions to those shown but you can move these around if you wish. The screen colour may

also be different but this too can be changed to your preference.

I've labeled some of the important things on the screen but it's important to realize that some of these will change depending on what you're doing with the software.

Have a look at the ***Feature Manager*** that I've labeled. Notice that there are three tilted squares marked Front, Top and Right. You always get these ***Sketch Planes*** free at the start of each part. If you let your cursor move over them you'll see red lines or a rectangle appear in the ***Graphics Area.***



Making the Crankshaft

We're going to start by creating the crankshaft shown here. This will be done by using the ***Extrude*** method to create cylinders for the shafts and the wedge shape for the *web*. Creating solids always starts with one or more ***Sketches*** which are a major part of SolidWorks. They're usually 2 dimensional and are used in various ways to generate the 3 dimensional forms that make up a solid object. We'll start with a sketch to create the wedge shaped 'web' that connects the shafts.

1. Click on ***Front*** in the Feature Manager window to select the Front Sketch Plane. A red rectangle in the graphics area should turn green with square dots. The Front sketch plane is now active – notice it's highlighted in blue in the feature manager too.

2. Click on the ***Sketch*** icon, just above the feature manager. Notice that the modeling tool icons are now drawing tools like lines, arcs and rectangles etc. Press the **Create Circle** button. Move the cursor (now a pen shape) to the intersection of the two red arrows in the middle of the screen. You've got there when a red dot appears and the cursor gets an orange square beneath it.

3. Click the left mouse button and move the cursor upwards. A circle should now be 'rubber-banding ', centered where you first pressed with radius to the cursor position. Notice the radius reading changing as you move. Make the circle about

3mm radius for now, you'll adjust this to the precise size later.

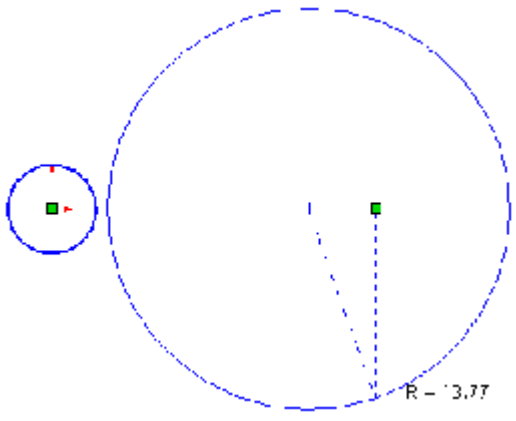
4. Click the left mouse button to finish creating the circle which should go green. Double left click and it will go blue. Roll back the scroll wheel on the mouse to enlarge the view of the circle until the size of a 10 pence piece. Try moving the view around on screen by holding down the ctrl key while dragging with the scroll wheel pressed. If the circle goes elliptical you've probably let go of the ctrl key too early – recover the view using the **Standard Views** button then clicking **Normal To** .

5.

Press the Centerline button . Hover the cursor over the centre of your circle and click when you get the red blob. Notice the symbol under the cursor change to an orange square; orange symbols mean that you're at some kind of snap point. Move the cursor rightwards and a dashed line will stretch from the centre. The line should snap to horizontal (or vertical), as it gets close. Click when the line looks like the picture.

6. Now for another circle, this time about 14mm radius and with its centre on the centerline just drawn. You should be able to do this without much help, remember to watch for the spikey orange symbol when placing the centre on the centerline. Click when your circle looks something like the next picture. Remember that we can easily re-size things later on when we add the precise dimensions.

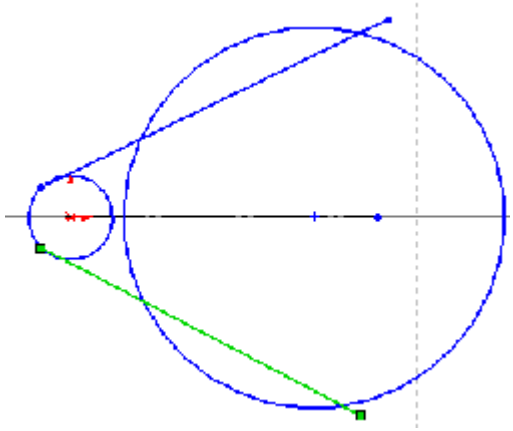
7.



Next we're going to create the straight edges to form the wedge shaped profile. Press the **Create Line** button . Move the cursor to somewhere near the upper left

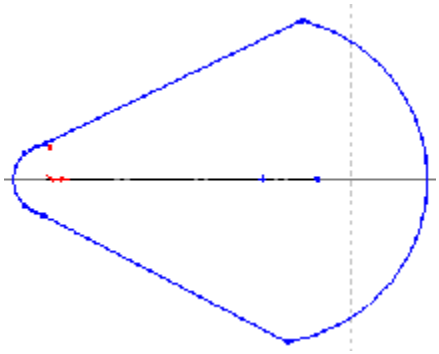
of the small circle, watch for the spikey orange symbol and left click. Move the cursor beyond the top right of the large circle and click. Repeat this procedure to get an angled line at the bottom too, giving you something like the next picture.

8. Now we need to tidy up the outline ready for the ***extrusion*** process. 'Hygienic' geometry, joining precisely at the corners, without extra fragments of lines is the key to easy solid modeling. A little practice and you'll soon get it right first time but be prepared for a bit of frustration at first.



9.

The ***Trim*** tool  is very useful for clipping off extra bits of lines or arcs. You'll probably need to hunt for it as it's off the default screen. Look for the 'More Icons' button,  near the top right of the graphics area. This will show you what you've been missing. Click the trim tool and snip away the extra bits and pieces until you get something like the next drawing.



10.

11.

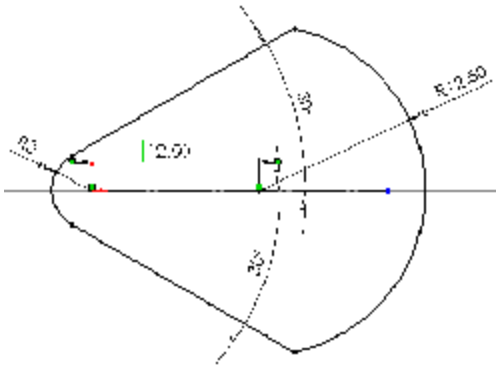
You might notice that my sketch is a bit lumpy around the left hand circle. This is because I wasn't very thorough with the trimming job. Zooming in (click  and

stretch a box around the bit you want to enlarge), gave me the next view, showing the extra geometry. Trim one of these bits away so there's no overlap. As you'll see in a moment, it doesn't matter which bit we trim. Your geometry may be neater than mine and you may not need to do this. Zoom all, will show everything if you tried a close-up view.

12. 

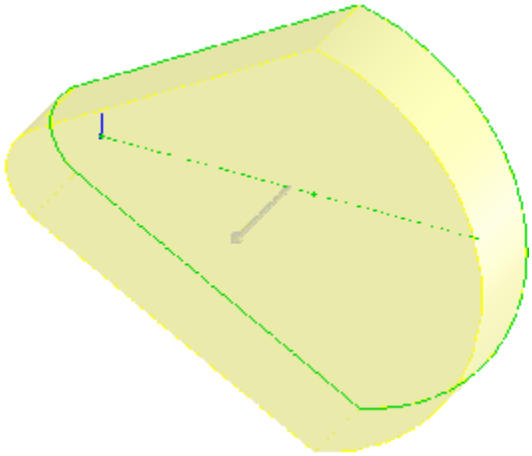
Next we'll add some geometric ***Relations*** to the sketch. We want the straight lines to be tangential to the small, left hand circle. Click on the Pointer Icon and click on the small left circle (what's left of it), then hold the ***ctrl*** key while selecting one of the straight edges. These two clicks should turn the geometry green. At the left of the screen you should see the Property Manager window look something like the next picture. Notice the two ***Selected Entities,*** and look for the Tangent button near the bottom. Click this and the selected line and arc will jump to make them tangential. Repeat this with the other pair.

13. The final step is to set a few dimensions so the sketch precisely defines the shape wanted. Find and click the ***Smart Dimension*** button ; It's toward the left of the horizontal toolbar above the graphics area. Let's set the radius of the two arcs first. Hover the cursor over the left hand arc. When it goes red click the left button. Move the curser to the left and you should see an arrow with a text-box follow you. When this is in a sensible place, left click again and the dimension is added. Now double click on the number itself and you'll get an edit box to change it to the right value - 3mm radius for the left one. Repeat this with the right hand arc but make this 12.5mm radius.

14. 

Now for the distance between the two arc centres. With the dimension tool still selected, hover over the centre of the left hand arc and left click when a red blob appears. It should go green. Hover over the right hand arc's centre and when the red blob appears here, press the left mouse button and drag upwards to see a linear dimension stretch from the centres. Release and click again when it's in a sensible spot. Double click on the number to change it to the 12.5mm we want.

15. Angles next! Still using the dimension tool, click on the centre-line we drew near the beginning. It should go green. You'll also have a blue dimension line following you that looks like the wrong thing. Simply click on one of the straight sloping lines and it'll change to an angular measure. Drag the text box to a sensible spot and click again to fix it. Double click and edit both angles to 30 degrees. Hopefully you now have something like me.

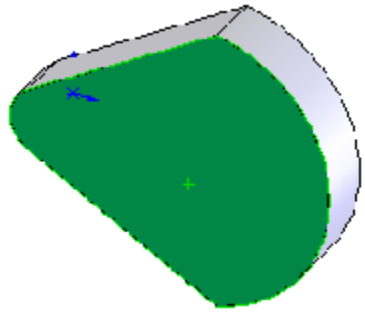
16. 

At long last we're ready to extrude this shape into a solid object. It doesn't always take this long but I've introduced dimensioning and relations as well as a selection of sketching tools too. Find and click the ***Exit Sketch*** icon at the top right of the graphics area. Looking in the ***Feature Manager*** window you should now see ***Sketch1*** at the bottom of the tree. Click on this to highlight it. Now click on the

Features button on the left of the lower horizontal toolbar. That toolbar will now get the solid modeling tools instead of sketch tools. Click on the ***Extruded Base/Boss*** button and your sketch should swing round to an *Isometric* orientation and acquire some thickness and a pale yellow tint.

17. The Feature Manager window shows the various parameters controlling the way the extrusion happened. In particular, look for the distance control , and the Blind extrusion method. A ***Blind*** extrude means that it has a set thickness, starting from the sketch plane. Experiment with the Mid-Plane option if you like, but the others need other geometry defined before they'll work. Click on the tick when you've set the distance to 3mm. and the yellow preview should turn grey – a solid part at last!

18. Try holding down the middle mouse button while moving the mouse and you'll see the part rotate about three axes. Rolling the mouse wheel will zoom in and out. Holding the middle mouse button down with the ctrl key pressed will pan the view. A bit of practice and you'll be able to get any view you wish.



Adding the Shafts

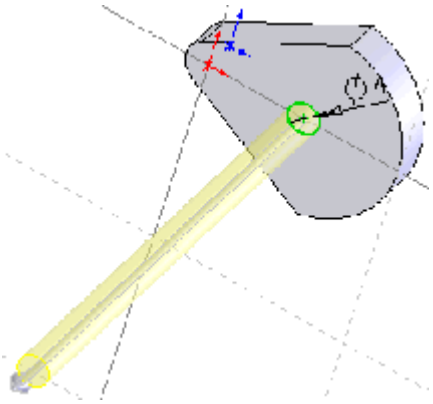
1. This is going to be easy. All we need to do is draw a circle on the face of the bit we've just made and extrude it for the right distance. First make sure your cursor is the basic pointer – click on if not. Now click on one of the large flat faces of the crank web. It should go green like mine…

2. We could sketch a circle and then extrude it but I'll describe a short- cut. Click on the ***Features*** icon and then on the ***Extruded Base/Boss*** button . You should now be in sketcher mode and the program knows you want to extrude with the sketch when finished.

3. We need to draw a circle at the centre of the large arc which is a bit tricky with the isometric view we have here. Luckily we can get a view normal to the face selected. Find and press the ***Standard Views*** icon and then press the ***Normal To***

button from the drop down list.

4. Click on the Circle button and draw a circle about 4mm diameter, centered on the large arc. Click the tick to finish it. To make sure the circle really is concentric with the large arc we're going to add another relation. Click on the pointer icon if the cursor is something different and then click on the large arc (which should go green), hold the ctrl key and click on the new circle which should also go green. The ***Property Manager*** should appear in the left hand pane with an entry for the green arc and circle. Click on the ***Concentric*** button in the property manager, click the tick and the circle becomes tied to the centre of the arc.

5. 

Use the dimension button to set the diameter of the circle to 4mm if you've not done this already.

6. Close the sketch with the ***Exit Sketch*** button , and you should see a pale yellow preview of the extrusion of the circle shown here. Remember we asked for an extrusion before starting sketching. Edit the distance value to be 63mm. and click on the green tick to finish off the main shaft.

7. Now for the crank pin. Create this by following steps 1 to 6 above but this time start the process on the opposite side of the web so the crank pin 'grows' in the opposite direction to the main shaft you just finished. Make the crank pin circle concentric with the small arc on the web and set its diameter to 3mm. Extrude it 10mm. from the face of the web. Look back at the finished part I showed at the start to see what you're aiming at. Don't forget to save your work.

8. Hopefully you've finished the crankshaft now. Congratulations! You've met quite a lot of SolidWorks techniques in just that one component so progress should get faster from now on.

Creating the Piston and Con-Rod

The engine you're building is called an Oscillating engine because the cylinder and piston oscillate as the crank goes round. As the cylinder swings back and forth it opens and closes ports letting steam into and out of the chamber. This makes the engine very simple which the next part shows well – the piston and connecting rod are just one piece unlike in most car engines where the connecting rod pivots at the piston connection. We're going to make this part by revolving a profile around a centreline.

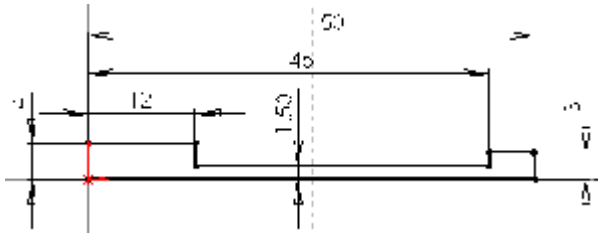
1. Start a new part using **File>New>Part** or clicking on the ***New Document*** button and clicking on ***Part.***

2. Click on the ***Front*** sketch plane in the ***Feature Manager*** pane.

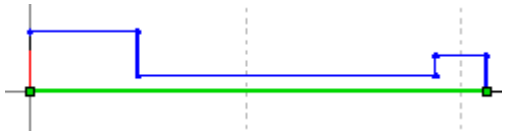
3. Click on the ***Revolved Base/Boss*** button and you should be taken into sketcher mode looking flat onto the front sketch plane.

4. Click the ***Centreline*** button and draw a horizontal centreline from the origin to about 50mm. rightwards.

5.

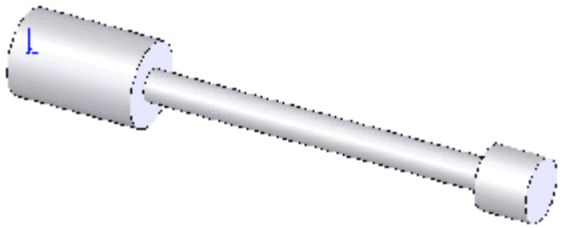


6.



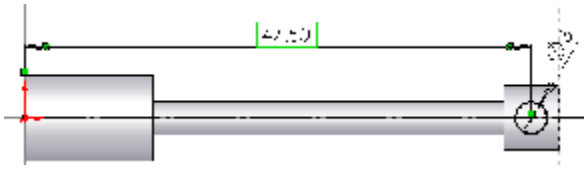
Click the ***Create Line*** button and draw the approximate shape shown here. We'll set precise dimensions later but try to make the lines either vertical or horizontal or you'll need to correct them later. (which is not a difficult task – simply drag the offending corner point.) Make sure that you draw a closed shape i.e. keep drawing back to where you started without any gaps in the chain.

7. Use the dimension tool to define the geometry as shown in the next diagram.

8. 

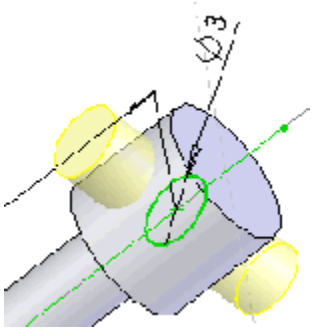
That's finished all the sketching for the revolve so click on ***Exit Sketch*** and you should see the pale yellow preview which by default has a full 360 degree revolution. Click the green tick to get something like the next picture.

9. Next we're going to cut a hole through the little end of the piston / con-rod for attachment to the crankshaft. Select the Front sketch plane which should go red and slice the piston through the middle. Click the ***Extruded Cut*** button and press the ***Standard Views*** icon and then press the ***Normal To*** button in the drop list.

10. 

Draw a centerline from the sketch origin, horizontally through the whole part. Now draw a circle, centered on this centerline, roughly in the middle of the smaller 'end' of the part. Dimension the sketch to look like the next picture.

11. 

12. 

Exit the sketcher in the usual way and you should see the usual pale yellow preview of the hole. Unfortunately the default setting only takes the hole one way

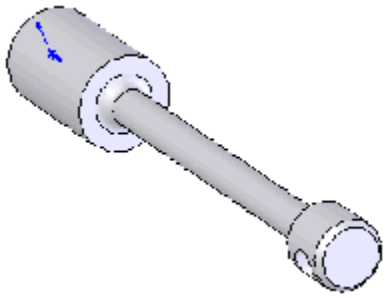
from the sketch plane. To fix this click the drop down arrow next to ***Blind*** in the property manager and then select Mid-Plane. The total distance of 10mm for this two sided cut will be enough to go right through the part. Your preview should look like this one. If so, click the tick and the hole will be drilled.

13. Before you finish the piston, now's a good time to play with ***filleting*** and ***chamfering***. These are very simple operations that do exactly what they suggest to an edge. First manipulate your view of the piston / rod so you can see the bottom edge near the hole just drilled. Then hover the cursor over the bottom edge until it goes red. Notice that other things go red as the pointer hovers over them and try to tell the difference between the bottom *face* and the bottom *edge*. The pointer has to be exactly on the edge for it to be *pre-highlighted* in red and when it is, the red line appears stronger. Click when you've found the edge and it should go green. If the whole face turns dark green you've picked the face and need to try again.

14. 

Now press the ***Chamfer*** button  on the features toolbar. Edit the values in the ***Property Manager*** as shown here and click on the green tick. You should now have a nice chamfer to blunt the bottom corner.

15. Use the same basic approach to create a ***fillet*** where the piston joins the slender connecting rod. Select the edge where they join, click the fillet button  , edit the fillet radius value to 1mm in the property manager and click the green tick. This is how mine turned out…don't forget to save yours.



Create the Flywheel

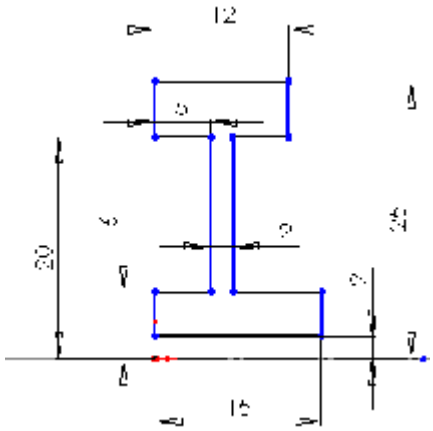
The flywheel will be created as another solid of revolution with holes punched through it to form gaps between the 'spokes'. First let's work on the basic ***Revolve***.

1. Begin a new Part by clicking **File>New>Part.** Or using the ***New Document***

button and choosing ***part***.

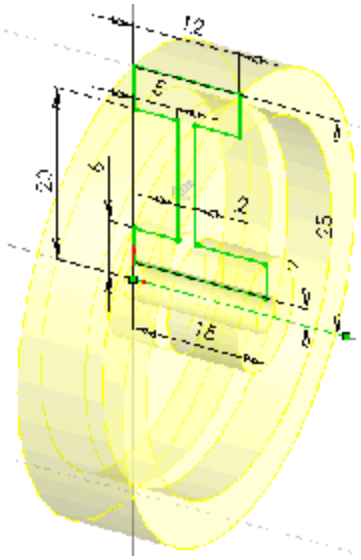
2. Click on the ***Front*** sketch plane in the ***Feature Manager*** pane.

3. Click on the ***Revolved Base/Boss*** button and you should be taken into sketcher mode looking flat onto the front sketch plane.

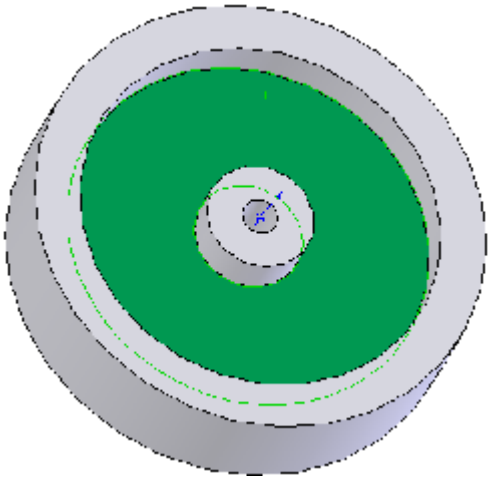
4. 

Now draw the shape shown here. As before, you can sketch roughly to start with and add precise dimensions later. Notice that the bottom of the shape is a little above the x – axis. We'll also need a horizontal centreline through the origin but it doesn't show up very clearly here.

5. It takes a while to get efficient with the sketch tools, with a little experimentation you should improve. They don't show in the picture, but I've added ***Colinear*** relations between the pairs of broken horizontal lines in the sketch and between the two vertical segments on the left hand side. This saved me dimensioning each line separately but more importantly, the sketch now captures my ***design intent*** more clearly.



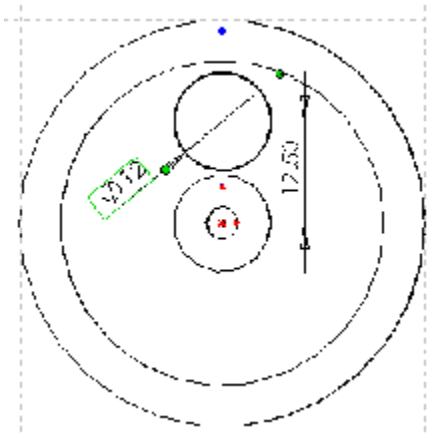
6. When you're happy with the sketch, press the exit sketch button as before and you should see the pale yellow preview of the wheel. If yours looks like this, click the green tick.

7. 

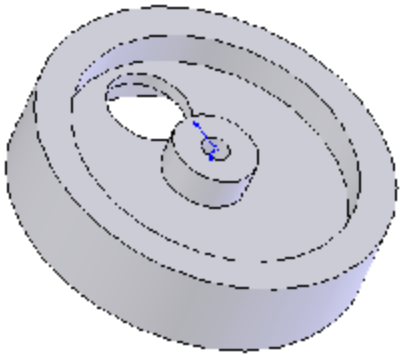
Now for the holes between the 'spokes'. First we're going to drill one hole, then we'll copy it in a circular ***pattern*** around the wheel. So far, the sketches we've used have been drawn on one of the basic sketch planes set up as the program starts. This time we'll sketch a circle on one of the flat faces of the part itself. This is really easy, simply hover the pointer over a face and click. The face you need is shown in the next picture.

8. Now click the ***Extruded Cut*** button  on the feature toolbar and you should be put into sketcher mode. You can probably arrange to view the sketch normal to the plane and sketch a circle without any more help now. I've drawn a vertical centreline through the origin to help place the hole precisely but it doesn't show up

very clearly here. Feel free to experiment with different size and position of hole if you wish.



9. Exit sketcher mode and change the distance setting on the property manager to ***Up To Next*** as shown here. Click the green tick and a neat hole should appear in the wheel's disc.

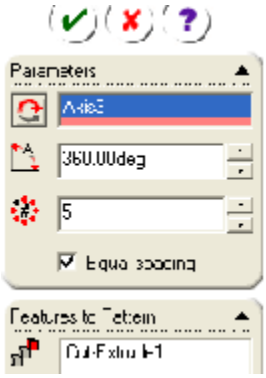


10. All that's left to do is copy this hole in a circular ***pattern*** around the rest of the wheel. To do this we'll need an ***axis*** to tell the program how to position copies. For this you'll need to explore the ***Insert*** menu. Notice the wide range of things we can create in addition to those on the toolbar. Select **Insert>Reference Geometry>Axis** and click. Now hover the cursor over one of the circular *faces* of the original wheel and click. The Property Manager pane will show that surface by name and you should see a yellow chain dotted line through the central axis of the wheel. Click the tick when you see this and you'll see ***Axis1*** appear at the bottom of the feature manager tree. All's set for a circular pattern.

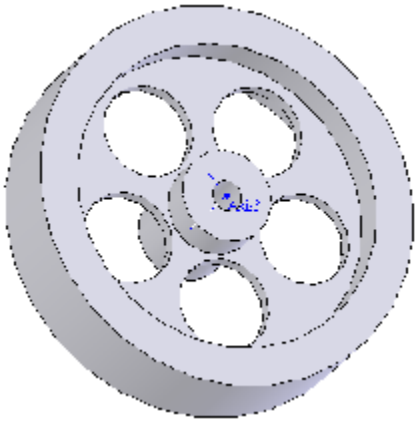
11. Hover the cursor over the inside, curved face of the hole you cut and click

when you get the face itself rather than just the edge. This should have selected the hole *feature* – have a look in the feature manager and you should see *Cut-Extrude1* highlighted. You could have selected it by simply clicking in the feature manager tree but I wanted to emphasise these two ways of selecting features.

12. Now find and click the ***Circular Pattern*** button  on the features toolbar. The Property Manager pane should appear on the left with *Cut-Extrude1* in the *features to pattern* box. The poorly named *Parameters* box should be highlighted in pink showing it's ready to receive some info. This is where we need to enter the name of the rotation axis. Simply click on the axis and its name should appear.

13. 

You should now see a pale yellow copy of the hole outline displaced by 15 degrees from the original. Edit the numbers in the property manager boxes so that it looks like this…(my axis is *axis3* not *axis1*)



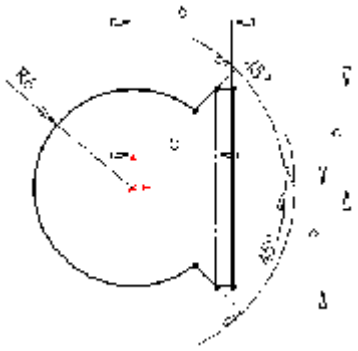
14. You should now have a yellow preview showing the four extra holes equally spaced around the full 360 degrees. Experiment with other arrangements if you wish. Click the green tick when you're finished. Mine looked like this…Make sure you save it carefully!

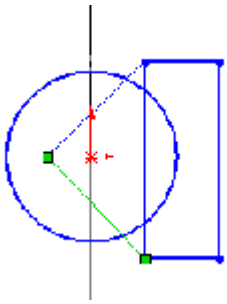
Create the Cylinder

The cylinder should be no trouble now. We're going to build it entirely out of extrusions.

1. Start as usual with **File>New>Part** or use the new document button and select **Part**.

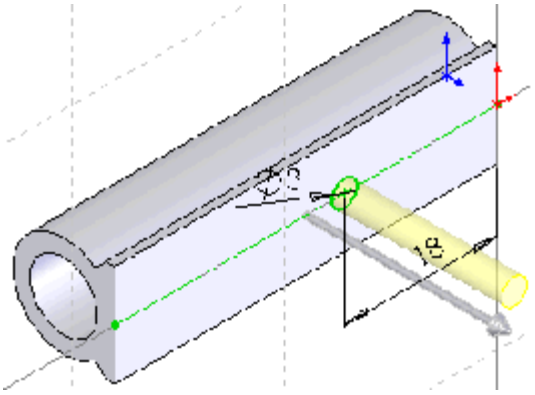
2. Click on the ***Front*** sketch plane in the feature manager and then click the ***Extrude Base/Boss*** button. This should take you into sketcher mode as usual.

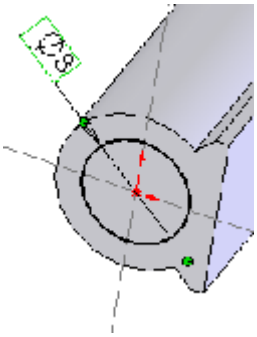
3. 

4. 

You may be getting confident enough with the sketch tools now to create the shape without more help. If so, carry on with your own approach. I created this sketch with a circle, centred at the origin with an overlapping rectangle plus two straight lines. I'm going to trim the spare bits later. Although I'm going to add precise dimensions later, it makes life easier if you start with the size about right – draw the circle with a radius of about 6mm. Oh, I also added horizontal and vertical centrelines from the origin to make construction and dimensioning easier.

5. The next picture shows the precise details after I've trimmed and dimensioned the rough starting sketch. Notice that I changed the redundant vertical line of the rectangle into construction geometry to help keep thing aligned. Click the green tick and set the ***properties*** up for a blind extrude of 45mm.

6. 

7. 

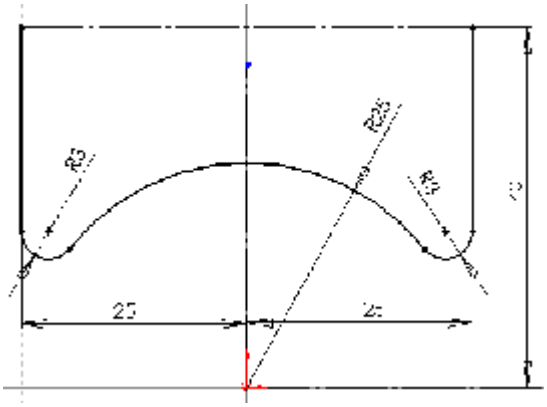
Hopefully you have a solid now, ready to bore the internal hole. Select one of the end faces of the part, which should go green. Click the ***Extruded Cut*** button and sketch a circle with a diameter of 8mm. If you drew the previous sketch centred at the origin, you should find it easy to centre this circle there too. Either way, it's a wise idea to set up a concentric relation between this new circle and the circular edge. When you've got this far exit the sketcher and set up a blind extrude of 40mm. When you click on the tick you should see that the hole doesn't go all the way through!

8. We're nearly finished, just need to add the pivot pin with another simple extrude. Select the large flat face and when it goes green click the ***Extruded Boss/Base*** button. All you need is a 3mm circle sketched as shown next. I first drew a centreline along the length of the face by snapping to the yellow mid-point markers which appear when you hover a drawing tool over a straight edge. This saved me adding an extra dimension and nicely captures my intention that the pin will always be on the centreline. Click the tick when you've got it looking right and save the finished part.

Making the Frame

Making the frame will demonstrate the use of the ***thin extrude*** feature where the program adds thickness to a line to create parts made from thin sheet material.

1. Start as usual with **File>New>Part** or use the new document button and select **Part**.

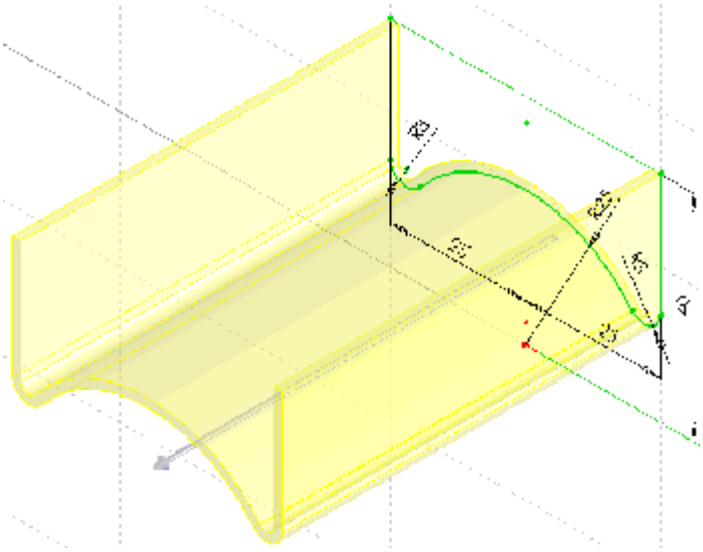
2. 

Click on the ***Front*** sketch plane in the feature manager and then click the ***Extrude Base/Boss*** button. This should take you into sketcher mode as usual.

3. Although this isn't a very complex shape I'll describe one way of forming it as we havn't made much use of arc segments yet. Click the ***Line*** tool and draw one of the vertical legs starting at the top. Its size and position only need to be roughly right. Now click the ***Tangent Arc*** button and drag an arc from the bottom end of the vertical line until it looks about right. With the same ***Tangent Arc*** tool still active, click on the arc just drawn and drag the bigger, central arc to approximate size. Repeat with the second small arc and finish off with another straight vertical line.

4. You might need to add a tangency relation between the last vertical line and the arc it springs from if you don't manage to finish that arc vertically.

5. I drew horizontal and vertical centrelines through the origin and dragged the large arc's centre to the origin too. Then I added the various dimensions. Something else new here is the use of ***construction geometry***, notice the horizontal line between the tops of the vertical lines. I drew this as a normal straight line to get the two verticals the same height but don't want the line to form part of the solid. I used the ***Construction Geometry*** button which you'll probably have to look for with the 'more buttons' icon at the top right of the toolbar.

6. 

Another way to create the shape would be to use the rectangle and circle tools and then trim the redundant bits of geometry. The small arcs could be formed with the fillet tool working on the sharp points where rectangle cuts circle (or what's left of them). The top of the rectangle can be changed to construction geometry as described above.

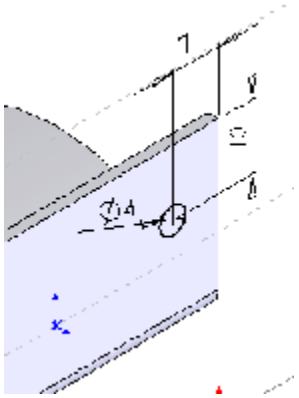
7. 

When you have the finished geometry exit the sketch as usual and you should automatically get a thin extrusion in the preview. You'll need to change the values in the property manager to those shown here and you'll need to click to put the material's thickness on the '*inside*' of the shape.

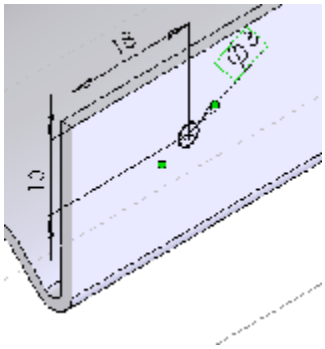
8. Click the green tick when yours looks about the same. Now to finish off with some simple holes for the crankshaft and cylinder pivot to pass through.

9. Click on one of the flat outside faces of the extrusion so that it goes green. Click

the ***Extruded Cut*** button and sketch a single circle dimensioned as in the next picture.

10. 

Exit the sketch and change the extrusion control in the property manager to ***Through All***. The preview should show the cut forming a hole through both of the vertical walls of the frame. Click the tick to accept this.

11. 

For the next hole, the cylinder pivot hole, repeat step 9 above but this time sketch and dimension the hole as in the next picture. When the preview shows, adjust the extrusion control to give ***Up To Next*** which will drill through just the first wall.

12. Exit the sketch and click on the green tick and the frame's done. Don't forget to save it carefully.

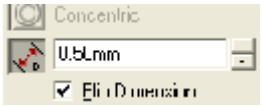
Assemble the bits

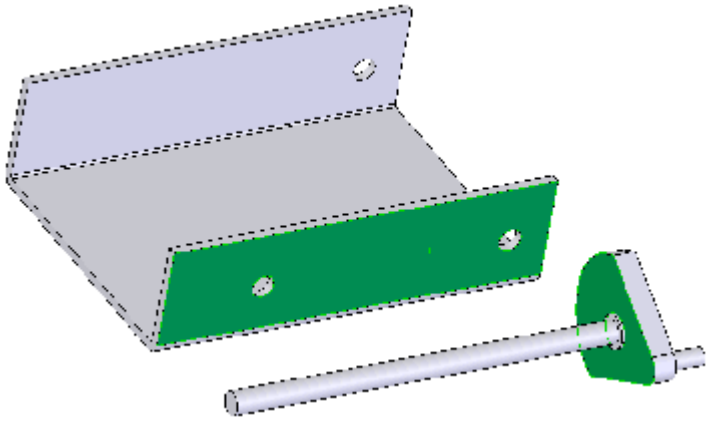
Now for the fun of sticking our virtual components together to form a virtual assembly.

1. Start by selecting **File>New>Assembly** (note the difference) or use the new document button and select **Assembly**. A blank work area should appear and the ***Property Manager*** should be set up to insert a component.

2. Notice the Browse button in the Property Manager pane. Click this and browse to the file where you saved the *Frame* you've just finished. I hope you can remember what you called all the parts and where you saved them too! As you select the file, you should see that part appear on screen. You can move it around if you wish but for this first part just click on the tick to confirm. We started with the frame as SolidWorks makes the first part in an assembly fixed in space. Although we can change this it makes sense for the frame to be fixed.

3. Next add a *Crankshaft* using the ***Insert Component*** button  . When it appears on screen move it around a bit before confirming – I've noticed that unless you move them, added components can get fixed by mistake when using some advanced SolidWorks features. Hopefully you've now got a crankshaft on screen too, although it's unlikely to be in the right spot. We'll set up the ***Mate*** conditions to put that right next.

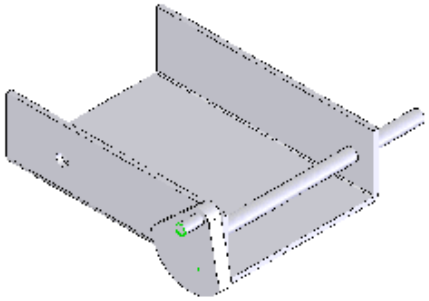
4. 

5. 

Click on the flat face of the frame with two holes; it should go green. Hold ctrl and click on the flat face of the crankshaft web with the longer shaft; this too should go green. With these two faces selected and both green like mine in the picture, click on the ***Mate*** button  and you should see the crankshaft swing around so the two selected faces are co-planar. So far, so good. But we really need a little clearance between the crankshaft and the frame if it's going to run freely. Edit the Property Manager pane so there is an offset distance of 0.5mm between the two selected faces. Click the green tick twice to confirm this mate condition.

6. Click on the long shaft of the crankshaft; it should go green. Hold ctrl and click on the inside face of one of the holes for the crankshaft which too should go green (you may need to zoom in a bit). With these two faces selected and both green,

click on the ***Mate*** button again. Notice that SolidWorks makes a most-likely guess about how to mate the selected surfaces together.

7. 

If yours looks like the picture everything's going well. Try dragging the crank-pin of the crankshaft (the small shaft) with the cursor and you should see it rotate. Save your work so far…

8. Adding the other parts is just as simple as the last step so you probably don't need much more description of how to do it. Sometimes a part gets placed the opposite way round to your intention. If this happens you need to delete the faulty mate in the ***Feature Manager Tree*** and perhaps try adding another mate first that gets things in the correct orientation. You could also try moving the part with the ***Move*** or ***Rotate Component*** buttons , which gives the program a better chance to guess your intentions.

9. Add the remaining component parts to give the assembly shown next. You'll see that I've set each part to have a different colour. To do this, select the part in the Feature Manager pane and then select its colour from the pallet . We're over half way to completing the engine shown at the start, that will be in the next tutorial as well as a look at creating 2D engineering drawings.