

TUTORIAL 1 Introduction to CAD Modelling

After seeing how to start the application and create a part file, we will use some of the feature creation tools in SolidWorks to build a part model. We will see how the combination of a number of simple features results in a relatively complex part. In creating these features we will also begin to learn about 2D sketches, which are fundamental to nearly all CAD modelling.

Goals

- Become familiar with the SolidWorks user interface and file creation
- Create sketches with dimensions and sketch relations
- Use sketches to generate solid features
- Understand how the combination of solid features can create a complex model (Feature Tree)
- Gain an appreciation for the planning (design) that should go into creating a solid model
- See how design intent can be embedded into a modelled part to make subsequent model changes easy.

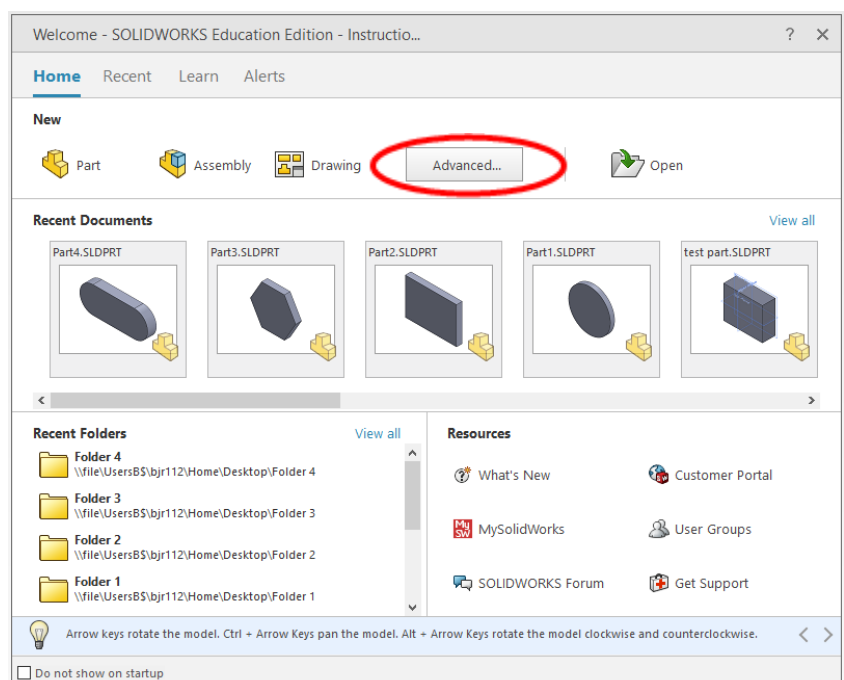
Opening the Application and Creating a File

SolidWorks is a Windows application and operates on similar principles to software you will already be familiar with, such as Microsoft Office products. To begin with we'll start the application, have a quick explore of the interface, and then create our first part file.

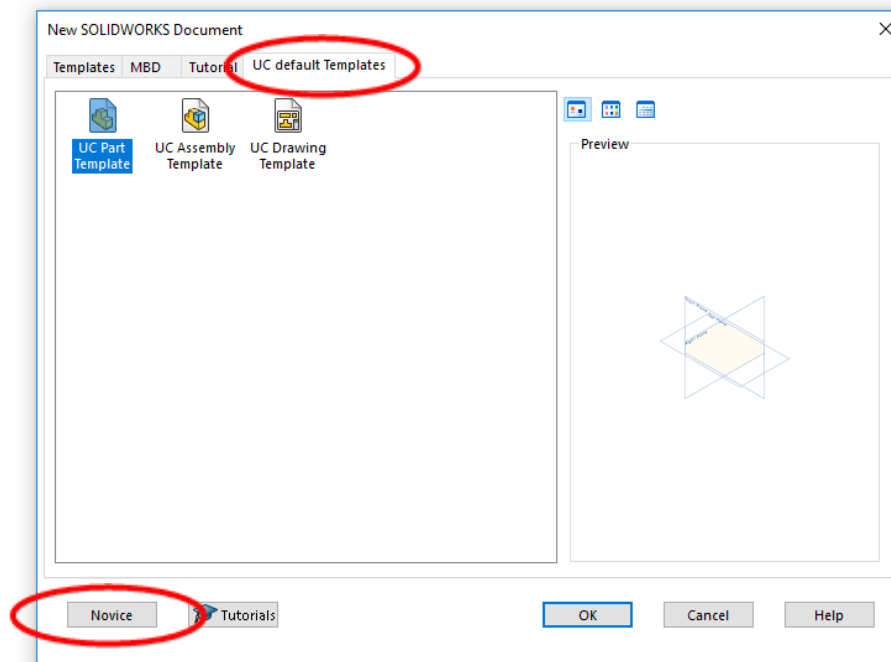
1. Search for Solidworks in the windows start menu and click on Solidworks 2018 Desktop app (note there are other Solidworks apps too, make sure you get the right one)



2. The first time you start SolidWorks the new document window will look like this... note the *Advanced* button around the centre of the screen. This actually means you are currently in the Novice setting, so you need to press that button to change to the *Advanced* user window.



3. A popup window appears that allows you to select from different templates and file types. The three types of files that you can create are Parts, Assemblies, and Drawings. At UC we have created templates for each of these file types, which set things such as fonts, dimension appearance and color schemes. Also these templates are set up so that custom properties, such as material type, name, part number etc, can be entered for the part and then have these automatically appear when the drawing is created of the part, or BOM (Bill of Materials) is generated. **These cannot be changed later, so be careful to always utilise the UC templates provided for the project you work on during your studies.**

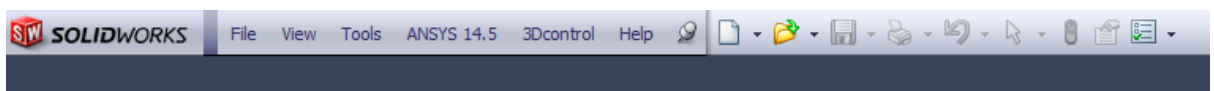


Note the tab which you open the file from is the **UC default Templates** tab.

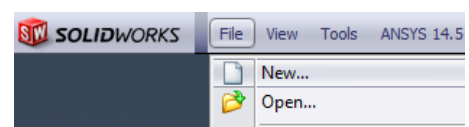
Also you can see the Novice button in the bottom left, which means you are now actually in the Advanced user interface!

With the **UC Part Template** icon highlighted, press the OK button.

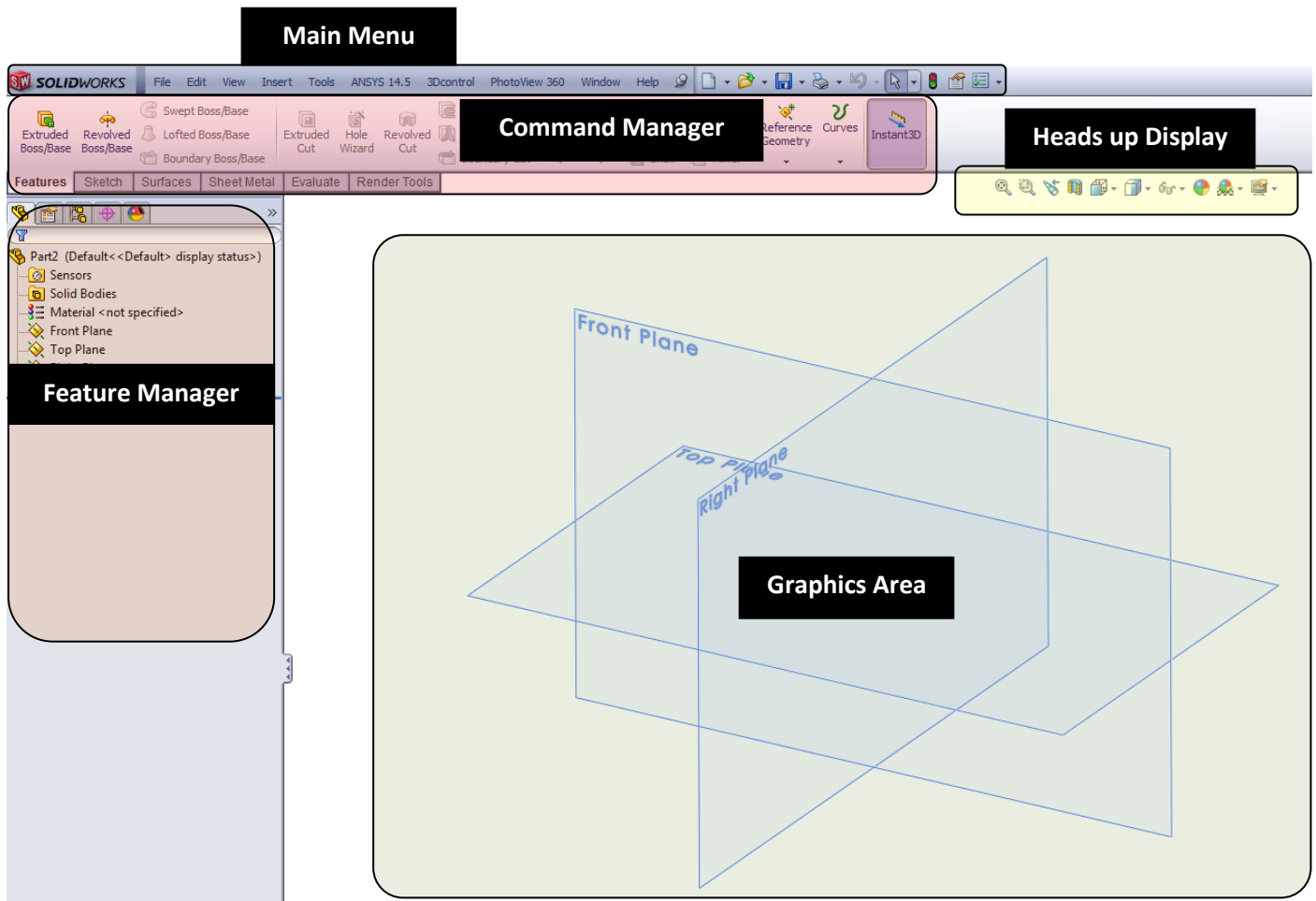
4. The top of the screen will look as in the image below. Click on the little arrow to the right of the SolidWorks logo and then click on the drawing pin icon to pin the popup menu in place. By doing this the main menu will always stay there so you can easily access it.



5. A new file has now been created, but if you wanted to create another one while Solidworks is already open, you can always select *New* under the *File* menu.



The main window will now look similar as shown below. The three planes that are visible are the principle planes which are automatically created and cannot be deleted



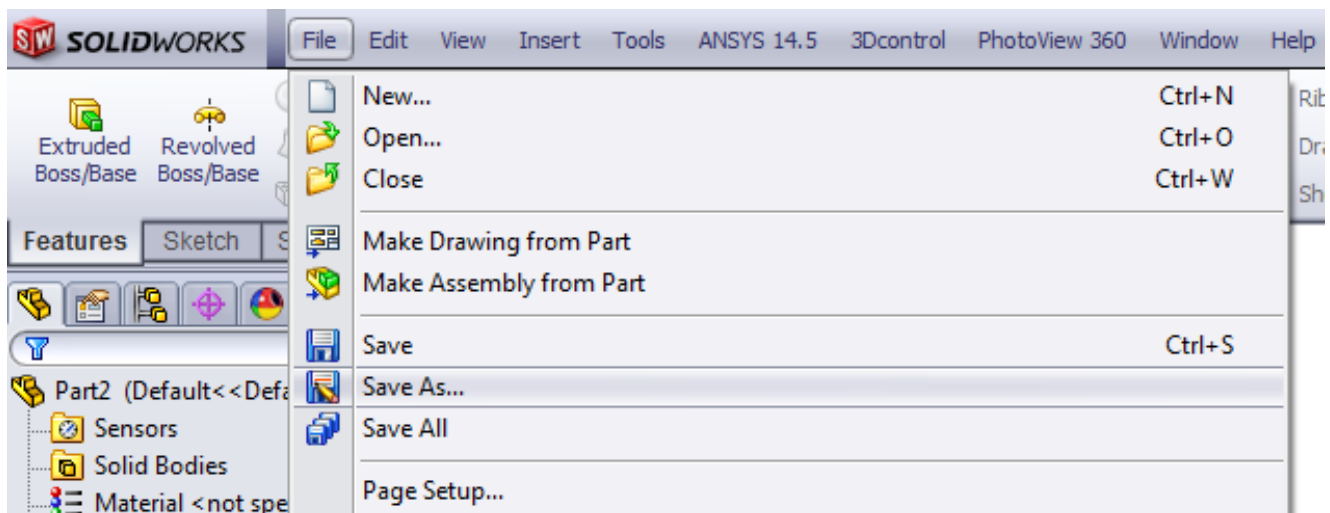
Some things to note on the blank part:

- The top of the window has a Menu Bar similar to other applications.
- Below the Menu Bar is the Command Manager, which is a tab based interface SolidWorks have developed to make it easier to find modelling tools quickly by grouping them together. All of these tools are also able to be found in the Menu Bar, but usually you will access them in the Command Manager.
- The left hand side of the screen is the Feature Manager which contains a list of all the features in your model and provides ways to edit them. All parts begin with three planes and an origin, which you can see in the Feature Manager. Sometimes the list of features is referred to as a Feature Tree.
- The central part of the screen is the graphics area where you can see your model. To begin with you will only see the three default planes because there are no other features. The origin is at the intersection of these planes, but is not visible by default in this template. Location of these planes and the origin is an arbitrary location in space, but provide an

important reference for your subsequent work.

6. By default the new part you have created is given a file name of PartX where X is a number which increments with the number of unsaved default parts that you have created in the current session. You can check the part name by looking to the right of the Main Menu. Note that if you have unsaved changes in the file there will be an asterisk (*) next to the file name.

Save the file with a file name before you start modelling on the part. This helps identify the part if you have multiple parts open that you are working on, and is a big help if the software crashes while you are working on it. Selecting either **Save** or **Save As** will open up a typical Windows file saving dialogue. Make sure to save somewhere appropriate so you can retain access to your files. Later on you will learn how files can reference each other, so it is best to keep all of your SolidWorks files in the same folder. Call this first part **engine block**



Planning and Modelling Parts

The general workflow for creating a part is to draw a 2D sketch or series of sketches and then use a 3D feature tool to create a solid body from these. This is repeated, with each subsequent feature merging with the previous (as long as they physically overlap), to create a 3D body that is as complex as we want. To illustrate this we will model an engine block for the single cylinder engine in our model car.

Before modelling any part its important have in your mind a plan for how you will go about doing it, or in other words what features will be combined to produce the final geometry. Even before this however you should have a good idea of what the finished geometry will look like. It's not important that you know all the dimensions, or detailed shapes (these are what CAD are great for working out

as you go), but it is a huge benefit if you at least know the basic form. Often it helps to sketch the part out by hand with pencil and paper, and then form a plan of how you will model it.

Usually we will move through this stage quite quickly, but for this first part we'll go a little slower to make the process clear.

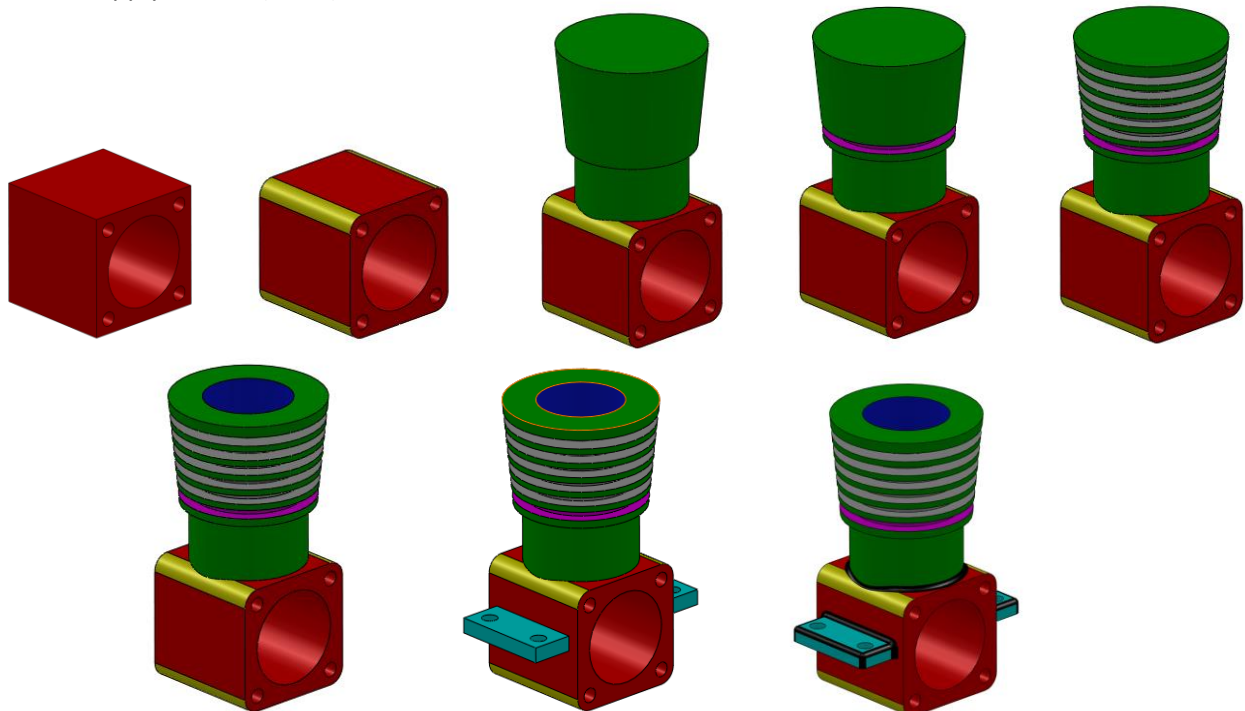
The part we want to model looks like this:



Think about the shape and try to break it down in our mind into elements that could be formed by extruding a profile, or revolving a profile around an axis. There are many other geometry creation tools, but these are all we need to think about for now.

The plan we will follow to generate the finished part is as follows:

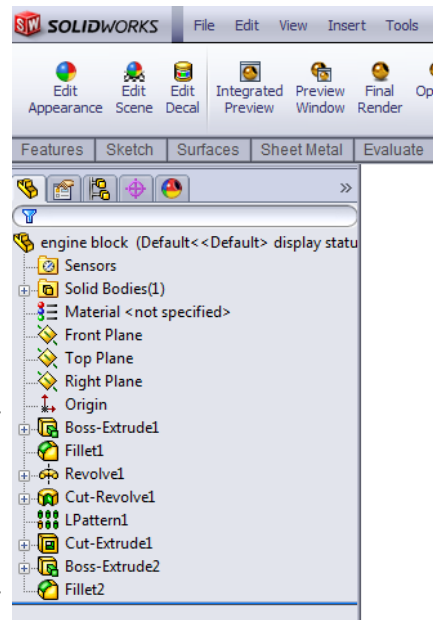
1. Extrude the red feature
2. Apply a round (in yellow) to the corners of the extrusion
3. Revolve the green feature
4. Revolve cut the pink feature
5. Pattern (grey feature) the pink feature to make more cooling fins
6. Extrude cut the blue feature
7. Extrude the light blue feature
8. Apply rounds (black) to blend between features



This is only a planning stage: read and follow the thought process but do not start modelling yet!

Study the Feature Tree in this screenshot to get a preview of what yours should look like when finished. There are eight features, each one implementing another step in our part plan.

Features generated one after another to produce final geometry



OK, you should start modelling from here...

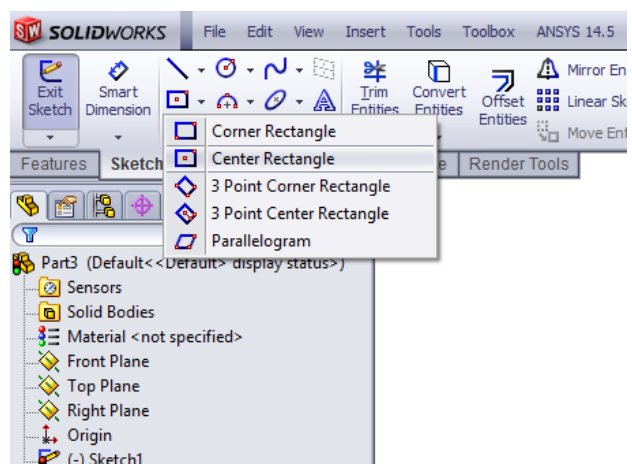
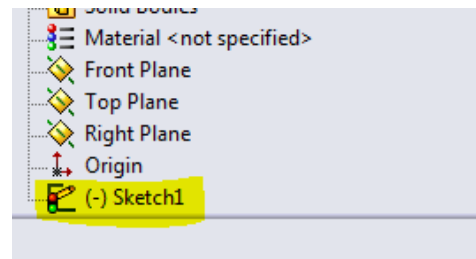
Based on this plan, we'll proceed one step at a time to create the model.

1. Select the front plane either in the Graphics Area or in the Feature Tree. On the Sketch tab of the Command Manager click on the Sketch icon. The planes in the Graphics Area automatically rotate so that the front plane (your current sketch plane) is parallel to screen. This is a default orientation for sketching, but you can rotate the model, either while sketching or looking at a 3D model, by pressing the middle mouse button and moving the mouse about (give it a try).

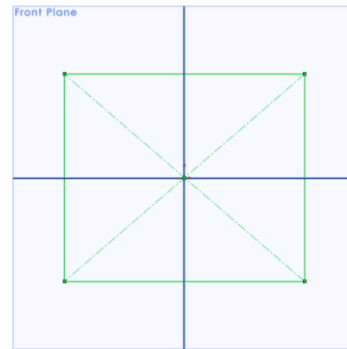
To zoom in or out, roll the mouse scroll wheel (try it now).

Notice how in the feature manager tree a Sketch1 has been created. The software will automatically apply default names to sketches and feature. It is possible to edit these names (slow double click on it, as in Windows Explorer), but usually it is not necessary or advisable to do so.

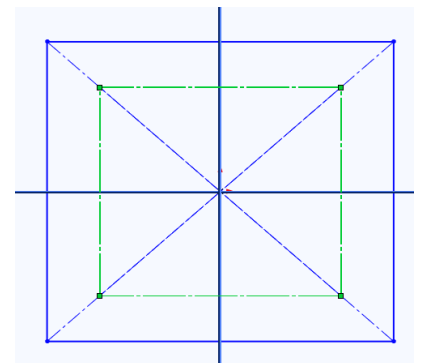
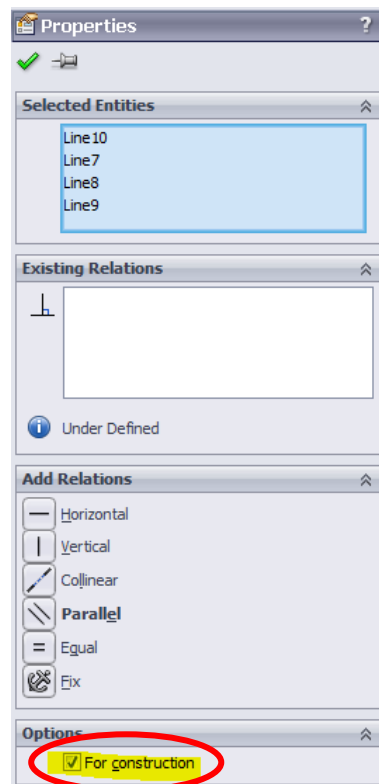
On the Sketch tab select the Centre Rectangle sketch tool. If you don't see it then click on the small drop down arrow to the right of whatever rectangle sketch tool you can see. This will give you all the options for different rectangle sketching.



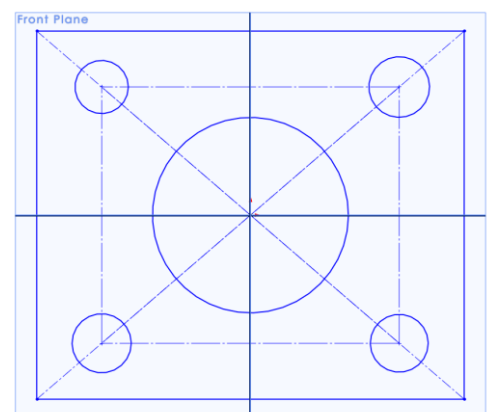
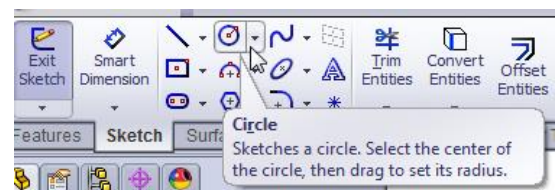
Click on the origin (intersection of the planes where the red arrows are) in the Graphics Area, and hold the mouse button down as you drag outwards. A rectangle will form, and when you have it a reasonable size then release the mouse button.



Create a second rectangle on the same centre, this time a little smaller but the same aspect ratio. This rectangle will be used as construction geometry to provide centre points for the four corner holes. Press Esc to finish with the rectangle tool. To change the lines to construction lines (which are ignored when using the sketch to create a solid), select each of them with the Ctrl key held down and then tick the *For Construction* box at the bottom of the Properties dialogue box in the Feature Manager area.

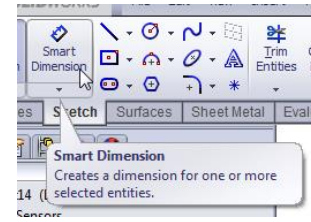


Click on the circle sketch tool on the Sketch tab of the Command Manager and then draw the centre circle and four smaller circles. Similarly to the rectangle, we click at the centre and hold the left mouse button down while dragging out the shape. Releasing the mouse finishes the sketch entity. Note that as you move around the sketch with the mouse pointer to create each circle, the pointer snaps to existing entities and icons appear next to the pointer as it does so. These are automatically offering to generate sketch relations such as concentricity or coincidence. Make sure when you select the centres that the pointer has picked up on your intended target (the dot will highlight orange) as this will ensure the relations are added and your design intent is embedded into the sketch.



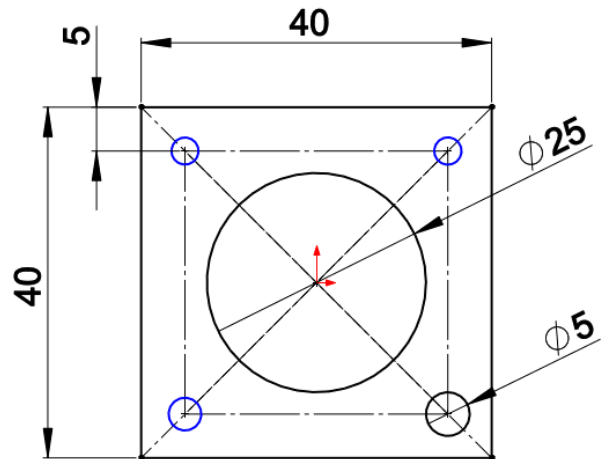
Note that we have created sketch geometry, but not applied any dimensions yet. This is a common workflow: create the form and content of the sketch, and then set the sizes.

To set the dimensions click on the Smart Dimension tool in the Sketch tab of the Command Manager.



This tool attempts to recognise what dimension you want, so for example clicking on a line will give you a linear dimension and clicking on a circle will give a diameter. Try this by clicking on the top line of our sketch and noticing that leader and dimension lines appear. To set the position of the dimension line move the mouse up a little bit and click again. Upon doing this a popup window appears asking for the value which you want to apply to this. Type in 40 and press enter.

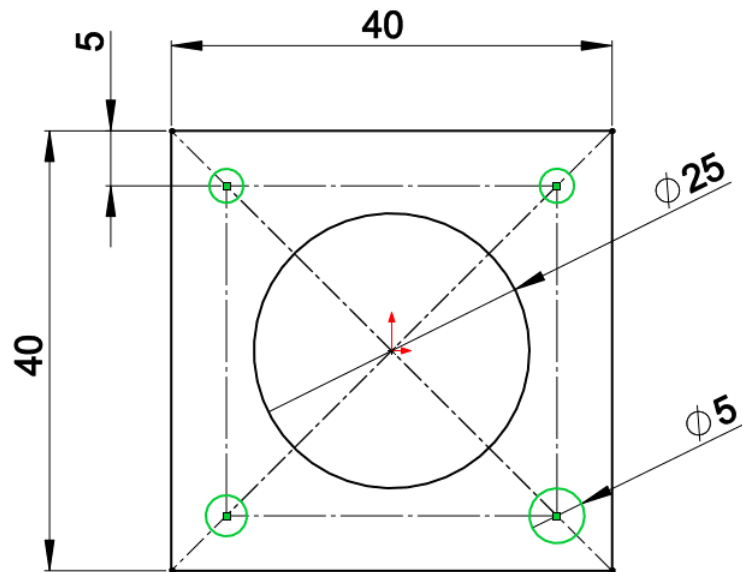
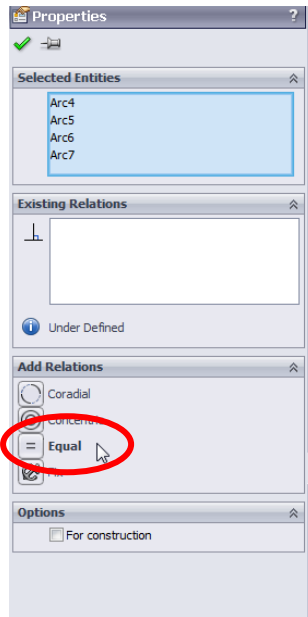
Repeat with dimensions for the height of the rectangle and diameters of the holes as shown.



For the 5mm dimension, which sets the distance between our construction rectangle and the outer rectangle, first select the top line of the outer rectangle and then the top line of the construction rectangle. The Smart Dimension tool recognises that because you have selected these two parallel lines then you want to dimension the distance between them.

Notice how as we apply more dimensions, more of the sketch geometry changes from blue lines into black. The blue indicates that the geometry is not fully defined (you can click on it and drag it around), and the black indicates it is fully defined by dimensions and relations. Once geometry is fully defined you can only move it by editing sketch relations or dimensions.

In this case we now have one of the four corner holes (the one with the dimension on it) in black, and the other three are blue. We could put dimensions on all four of them, but this would mean that editing the size of these would require four separate dimensions to be changed. By using a sketch relation we can simplify this and embed our design intent (that all of these holes should be the same diameter) in the sketch. Press Esc to end the Smart Dimension tool. Select all four of the circles while the Ctrl key is held down and then select the **Equal** relation in the Properties dialogue in the Feature Manager area.



This sketch relation will make all of these four holes the same diameter, and because one of them is already set to diameter 5mm then all of them will now be at this size.

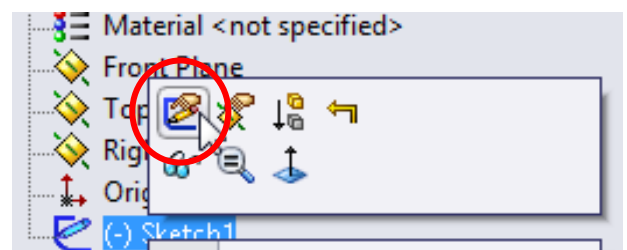
Click the green tick button on the top left of the Properties dialogue to finalise this and return to the sketch.

The sketch is now complete, so click on the Exit Sketch icon at the top right of the Graphics Area to exit out of the sketching mode.



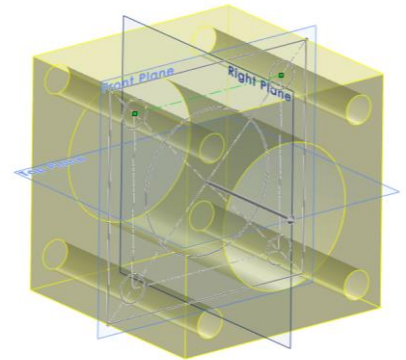
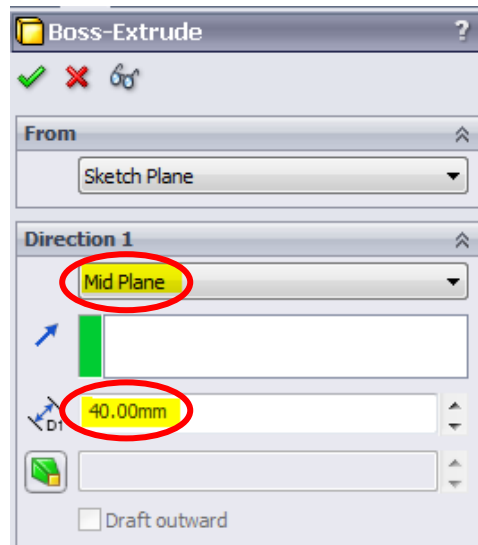
Notice that once out of the sketching mode the sketch color changes to grey. Attempting to apply dimensions to these grey lines will not work.

If you need to do add more dimensions, or make other changes to the sketch, then right mouse button click on the sketch in the Feature Tree and select the Edit Sketch icon which pops up (top left). This takes you back into the same sketch mode you were in when initially creating the sketch.



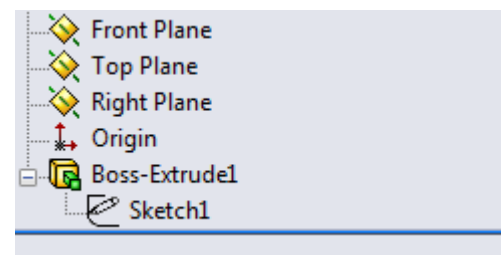
Now that we have the sketch, which defines the profile, we can use it to create the first solid body. Click on the Extrude Boss/Base tool in the Features tab of the Command Manager and then click on any entity within your sketch in the Graphics Area. Doing this indicates that this is the sketch that should be used as a profile to extrude along in a straight line.

A Boss-Extrude dialogue appears in the Feature Manager area which allows you to set various options which determine the behaviour of the extrude feature. In this case we want to select **Mid Plane** from the direction dropdown menu, and enter the value of 40 as the distance. This will cause the sketch profile to be extruded in 20mm in each direction away from the sketch plane.

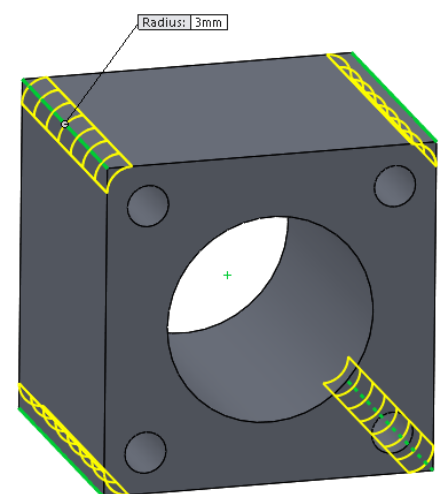
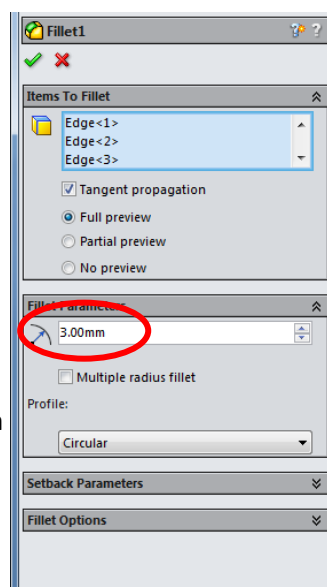


Once these are set you should see a yellow preview of the feature as shown. Click the green tick at the top of the Boss-Extrude dialogue, and the solid body is created.

Notice that in the Feature Tree the sketch still exists, but it has been absorbed into the **Boss-Extrude1** feature. Click on the + next to the **Boss-Extrude1** feature and you can see the **Sketch1** is still there. This is important to realise as the sketch is still available for use in creating other features (not required here), and if the **Boss-Extrude1** feature was deleted the **Sketch1** would still be there. In a sense the extrude feature is borrowing the sketch, but never owns it.



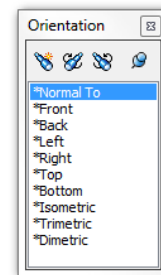
2. Apply the rounds (yellow feature in the part plan) by clicking on the Fillet tool on the Features tab of the Command Manager. Select each of the four edges as shown (rotate the part if necessary) and enter the radius value of 3 into the Fillet dialogue in the Feature Manager area. Click the green tick icon at the top of the Fillet dialogue and the rounds will change from the yellow preview lines to a solid feature.



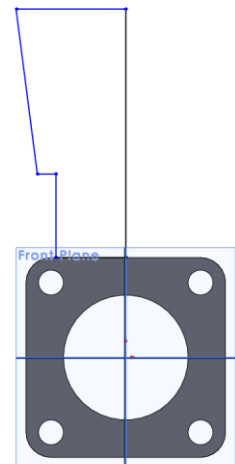
- We will create the cylinder (green in the plan) using a revolve feature. To do so we will first create a sketch in a similar way to when we created the extruded feature, but this time instead of extruding we will revolve the sketch around a centre line (axis) to create a solid feature.

Click on the Front plane in the Graphics Area and then click on the Sketch tool in the Sketch tab of the Command Manager. This will initiate a new sketch on the Front plane.

Tip: If the plane you are sketching on is not parallel to the screen, and you would like it to be (it's usually easier for sketching if it is) then press the space bar to bring up the orientation popup window. Click the pin symbol at the top of it so that it will stay on the screen. Double clicking on the **Normal To** option of the Orientation popup will move the geometry so that whatever plane you are currently sketching on, or have selected, will be orientated parallel to the screen.

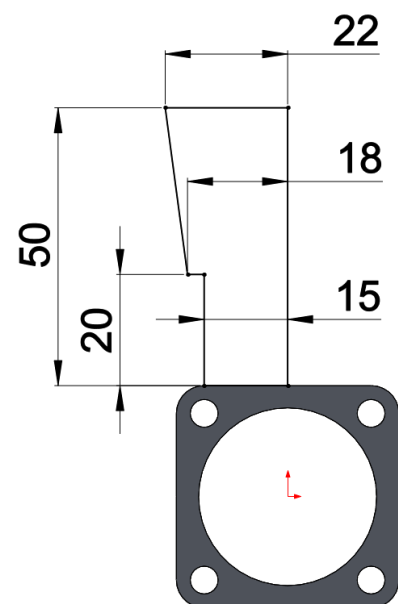


Select the Line sketch tool from the Sketch tab of the Command Manager and draw a shape approximately as shown. Do this by choosing any corner to start by clicking at that location in the Graphics Area, and then moving around the shape clicking at each corner. Once you have completed the loop back to the start point the line tool will automatically stop, or if you want it to stop before you have completed a loop then press the Esc key. The exact position doesn't matter because, as before, we are following a workflow of creating a shape and then setting its size.



Be careful when creating the bottom horizontal line that it snaps to a coincident relation with the top of the first extruded body. **If it is a little above then a gap will be modelled between the two features, which will prevent them from merging into one body as we want.**

Select the Smart Dimension tool in the Sketch tab of the Command Manager and apply dimensions as shown. Remember that dimensioning between two entities (eg a vertex to a line or a line to a line) is a three click operation: first click on one entity, then on the other, and finally click where you want the dimension line to be located. While the dimension tool is active you do not need to select it again, just keep clicking on sketch entities to create more dimensions.



Now that the sketch is completed and dimensioned, exit the sketching environment by clicking on the Exit Sketch icon in the top right corner of the Graphics Area. Notice again that after you do this the sketch lines turn to a grey color, indicating you are no longer in the sketching environment.

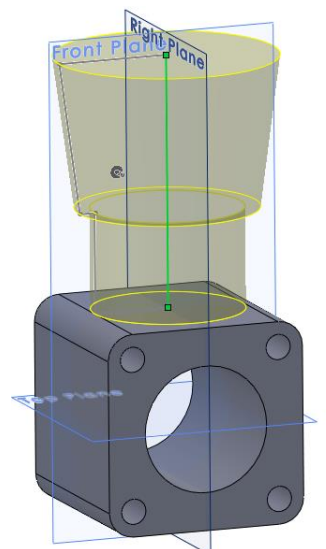
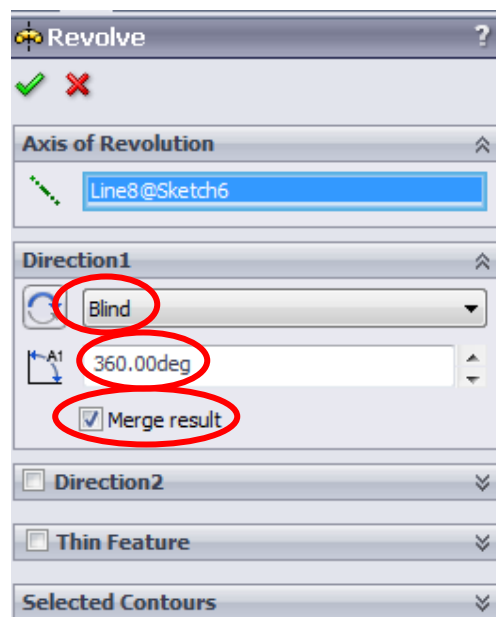


Now to use this sketch to create another solid feature, select the Revolve Boss/Base tool in the Features tab of the Command Manager. Click on the sketch, but this time be careful to choose the line in the sketch that we want to revolve the sketch around. Selecting the line has the dual function of selecting the sketch and the revolve axis. If you select the wrong line for the revolve axis it can easily be changed in the Revolve feature dialogue box, but it's easier to get it right first time.

The preview should look something like this. Notice the Axis of Revolution is already set by your first selection, and the default angle is 360 degrees, which is what we want in this case.



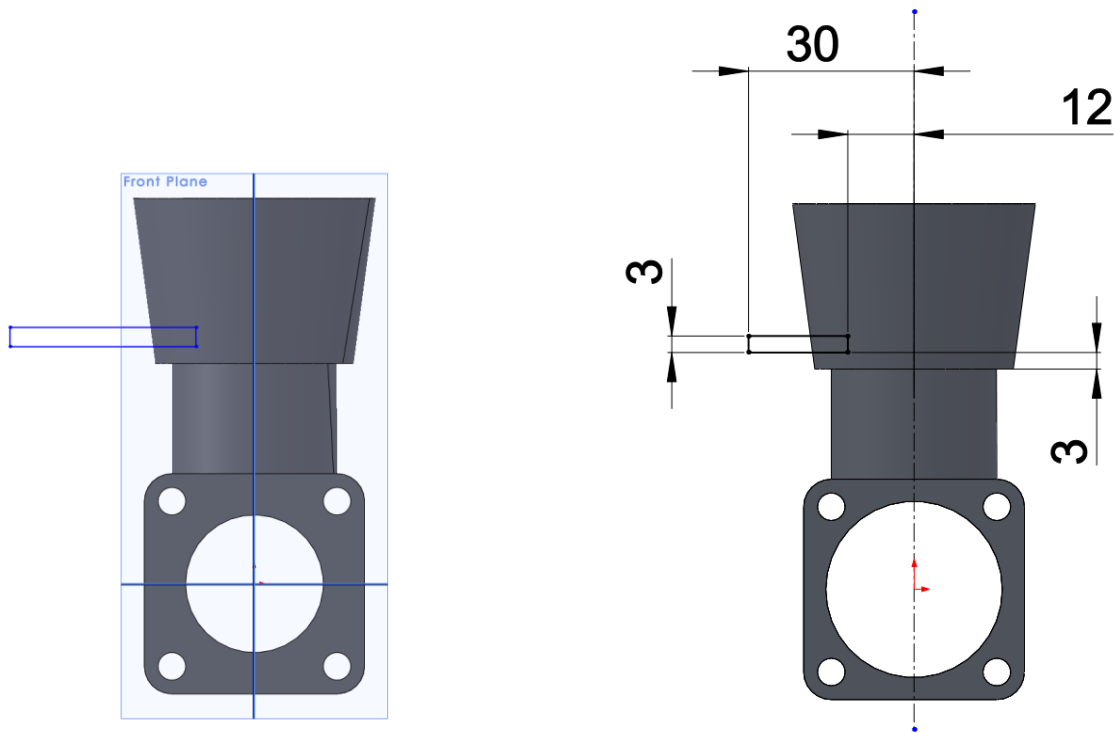
Also, importantly, note that the Merge Result tick box is checked. This means that the new feature will merge its body with any existing body that it contacts or intersects with. If this were not ticked then a separate body would be formed and this model would become a Multi Body Part. At times this is very useful, but in most cases such as this we do not want it.



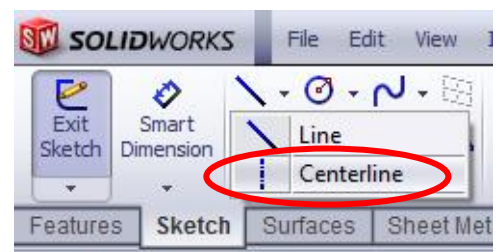
Click on the green tick at the top left of the Revolve feature dialogue box to accept these settings and generate the solid feature. You can tell when the solid feature is generated because the semi transparent yellow preview will change to a solid color.

4. The bottom cooling fin will be created by performing a Revolve Cut. This is similar to the Revolve Boss/Base, except when we revolve the sketch profile with this tool it will remove solid material rather than generating it.

Select the Front Plane by clicking on it in the Graphics Area, and then select the Revolve Cut tool in the Feature tab of the Command Manager. Using the line tool, sketch a rectangle approximately as shown, and then use the Smart Dimension tool to apply dimensions.



In the case of this revolve you will see that we do not happen to have a line entity that is on the rotation axis that we will want to use (centre of the part). This is easily resolved by selecting the **Centreline** tool, which is accessed by clicking on the small arrow to the right of the standard Line tool. Use this tool to draw a vertical line in the centre of the part as shown, allowing the automatic snapping to guide the line so that it is exactly on the centre.

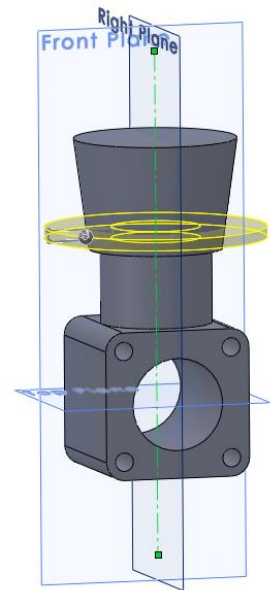
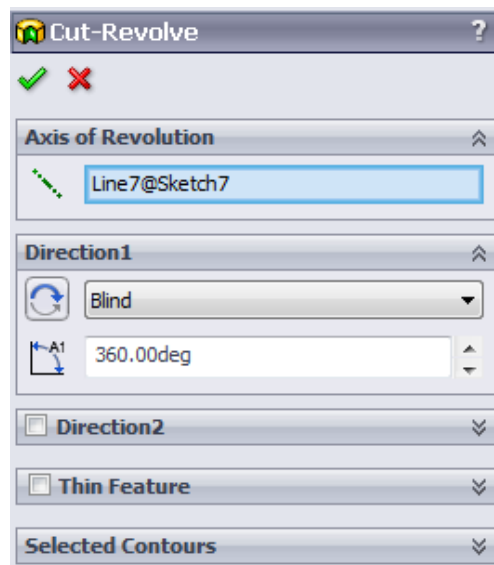


The Centreline tool makes great centrelines, but actually it is far more versatile than that because it is also used for construction geometry (lines that help you with a sketch, but do not form part of the profile when the sketch is turned into a solid feature). Remember when we changed some line types before into construction lines? That effectively turned them into Centrelines. Construction lines and Centrelines behave exactly the same.

Exit the sketch now to go out of the sketching environment (sketch turns grey and dimensions become hidden).

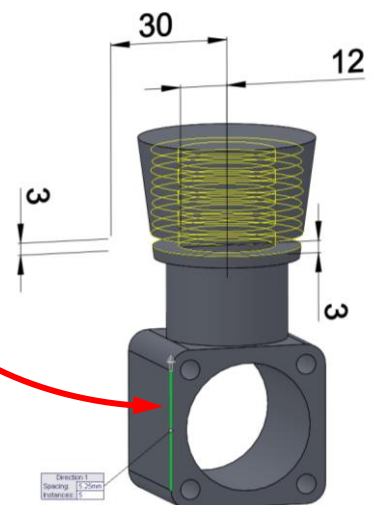
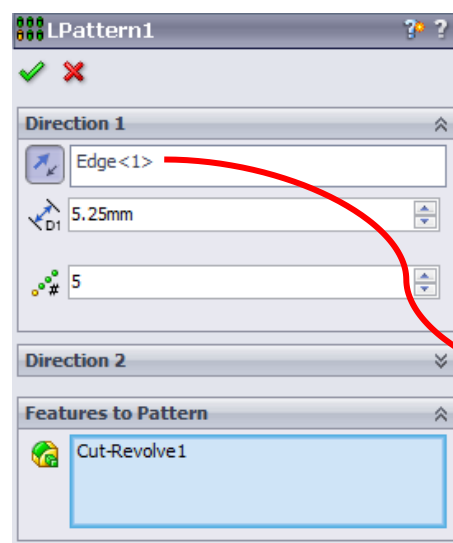
Select the Revolve Cut tool from the Features tab of the Command Manager, and then click on the Centreline that you created in the sketch. This automatically selects the profile that was included in that sketch, and the Centreline as the axis of revolution.

Click the green tick at the top of the Revolve Cut dialogue box and a groove (bottom cooling fin) will be cut in the cylinder.



- We could make the rest of the cooling fins using a similar method as the first, but because our design intent is for them all to be the same it makes sense to embed that intent by using a pattern feature to copy the first one.

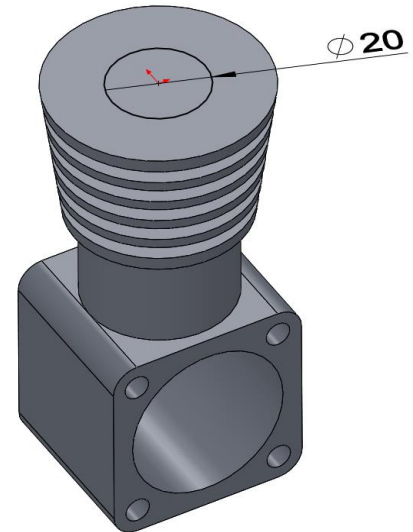
To do this, select the Linear Pattern tool from the Features tab of the Command Manager. Enter the values as shown. The Edge<1> defines the direction that the pattern will be made in, and can be any vertical edge or sketch that you can click on in the Graphics Area which is in the direction you want. In this case notice it is the green line selected on the part. To select the Features to Pattern, click on any surface that belongs to the groove which we want to pattern.



Once the yellow preview of the pattern looks as it should, click on the green tick icon in the top left corner of the **LPattern** dialogue box to generate the solid geometry.

6. The cylinder needs a bore down its centre for the piston to slide up and down in. To create this we will use the **Extrude Cut** tool. This works just the same as the **Extrude** tool, except the extruded profile removes material rather than creating it.

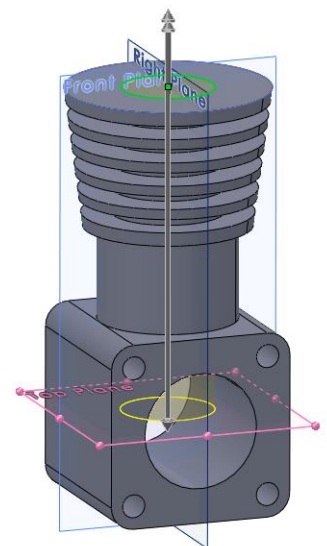
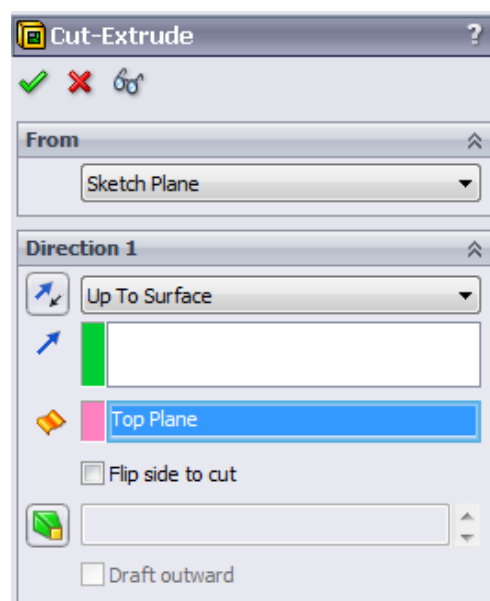
This time instead of creating a sketch on a plane we will do so on a planar surface of the part we are modelling. Select the top circular surface of the part in the Graphics Area, and then click on the Sketch tool in the Sketch tab of the Command Manager. Be careful when you select the surface that you are selecting the surface itself and not the edge around it. Experiment by selecting one and then the other; selecting the edge changes the color of the edge only and selecting the surface changes the color of the whole surface. Anything that is selected in the Graphics Area can be deselected either by selecting something else (without the Ctrl key being held, or it will multi select), pressing the Esc key, or clicking in a blank area of the Graphics Area.



Now that you are in the sketching environment, select the Circle tool in the Sketch tab and draw a circle in the middle of the cylinder, taking care that the circle centre snaps to the central sketch origin. Using the Smart Dimension tool apply a diameter of 20mm.

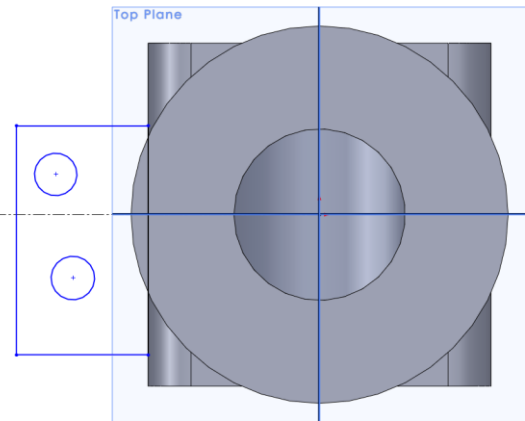
Exit the sketching environment by clicking the Exit Sketch icon in the top right corner of the Graphics Area.

Select the Extruded Cut tool from the Features tab of the Command Manager, and then click on the circle sketch in the Graphics Area. The default extrusion setting is **Blind** which will extrude a specified distance. Our design intent is that, regardless if we later doubled the length of the cylinder, the bore would go all the way through into the crankcase. To embed this intent we'll select the **Up To Surface** option and set that target surface as the Top Plane.



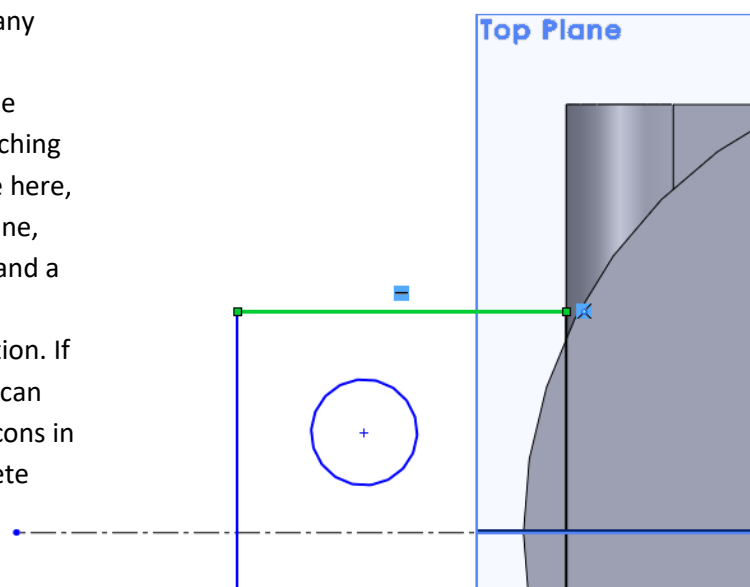
Once these are set and the yellow preview looks as it should, click on the green tick at the top left of the Cut-Extrude dialogue box to generate the geometry.

7. The mounting lugs are modelled next, using an extruded sketch profile. It is our intent that the mounting surface on the underside of the lugs should be on the centreline of the crankcase, so we will sketch on the Top plane and then extrude upwards. Click on the Top plane in the Graphics Area, and then select the Sketch tool in the Sketch tab of the Command Manager. Use the Line and Circle tools on the Sketch tab to draw a sketch as shown. **Note there are 4 lines** which make a closed rectangle surrounding the two circles.

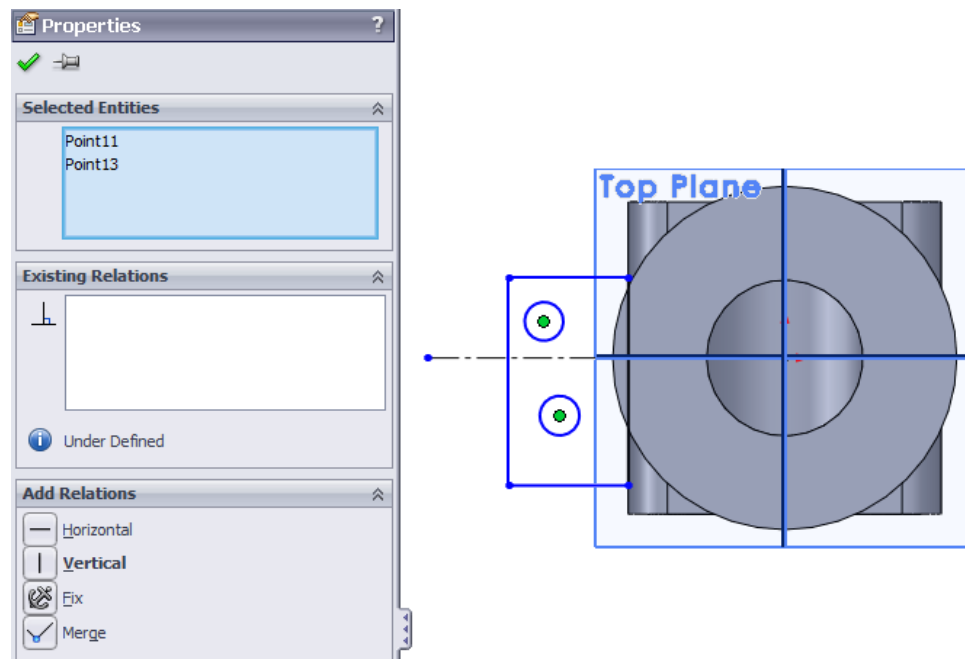


Notice that the example sketch is drawn purposefully in a fairly careless way. This is just to illustrate again the point which has been shown previously, that the dimensions of the sketch, when first done, do not matter. All that is important is to get the form and intent of it correct, and applying the dimensions and any additional sketch relations afterwards will make it exactly as we want. In this case, additional to the dimension requirements, our design intent is that the holes and lug should be symmetric about the horizontal Centreline (which you need to draw using the Centreline tool, or as a standard Line and then convert it to a Construction Line). Also the two holes should be vertically in line, and they should be the same diameter. To embed this intent we use sketch relations. We have already been using these quite a bit without realising it, as the software automatically applies them where it seems to make sense. For example if we draw a line close to horizontal the software will make it perfectly horizontal and add a horizontal sketch relation to force it to stay there. Another automatic relation example is when two line endpoints meet, at which point there is a coincident relation applied so that they remain connected even if one of the lines is moved.

Sketch relations that belong to any sketch entity can be viewed by selecting that sketch entity in the Graphics Area (while in the sketching mode). For example we can see here, by selecting the top horizontal line, that it has a horizontal relation and a coincident (with the edge of the crankcase extrude feature) relation. If desired the automatic relations can be deleted by clicking on their icons in the sketch and pressing the delete key.

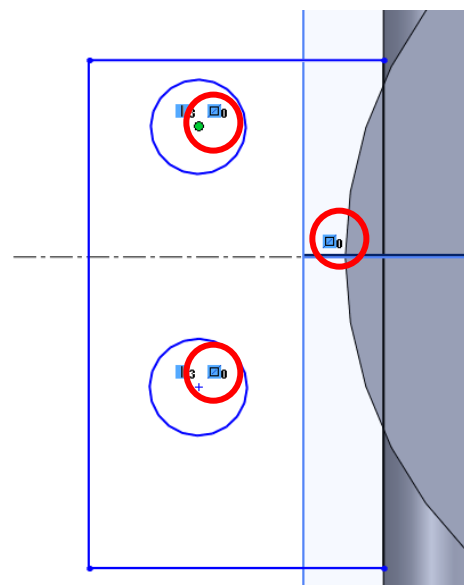


For now we will add some new ones. Start by clicking on both of the circle centres with the Ctrl key held down. A Properties dialogue box appears in the Feature Manager area which gives some options for all the possible sketch relations that you could generate with these sketch entities selected. The one we want is **Vertical**, so that one will always be vertically above the other (as oriented in the default sketch view). Click on the **Vertical** icon and you should see one of the circles move over so it is vertically aligned with the other. Click on the green tick at the top left corner of the Properties dialogue to accept the creation of this sketch relation.



Now to create the symmetry of the holes about the horizontal Centreline, select the two circle **centres** (not the actual circle) and the horizontal Centreline, all while the Ctrl key is held down so they will multi select. This time the property dialogue that appears has fewer options available, but the one we want, symmetry, is possible to do so it is there. Click on the Symmetry icon and one of the holes will move. Click the green tick to accept this application of this sketch relation.

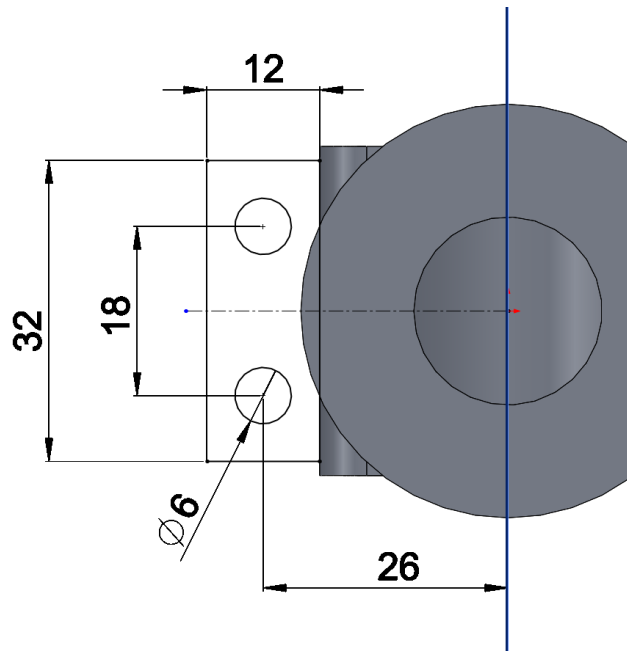
To confirm that all is as you expect now, click on the **centre of either circle** (not the actual circle) and you will see the relations that you have created. The Vertical between the two holes, and the Symmetry between the two hole centres about the Centreline. Also experiment by dragging either circle around by clicking and, without releasing the mouse button, moving the mouse. You will see that the other hole moves too, according to the demands of the relations in place.



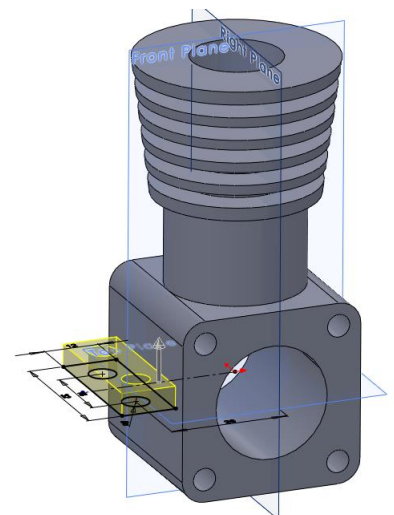
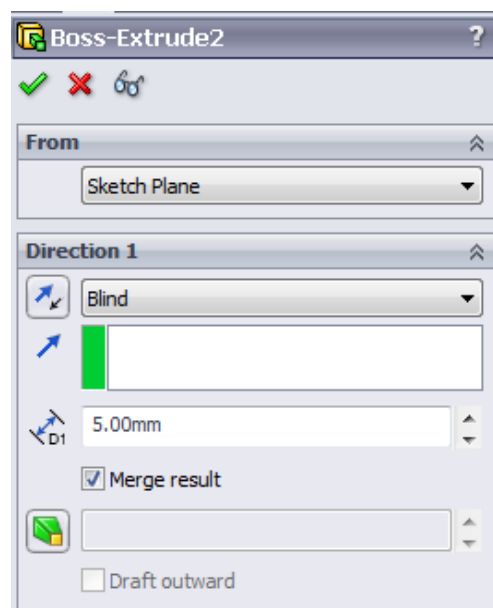
The diameters of the two circles can be made the same using an **Equal** relation, the same as we did on our very first sketch in this model. Select both circles with the Ctrl key held down and then select the **Equal** sketch relation.

The final sketch relation required is to make the top and bottom horizontal lines of the lug symmetric about the horizontal Centreline. Do this exactly in the same way as you did the circles, except selecting the lines and the Centreline rather than the circle centres and the Centreline.

Now that the content, form, and design intent is all included in this sketch, we can add dimensions to it. Do this, using the **Smart Dimension** tool found in the Sketch tab of the Command Manager, with the values shown.



Exit the sketch by clicking on the Exit Sketch icon in the top right corner of the Graphics area. To create the solid lug feature, use the Extrude tool and the sketch which we have just created. Make sure the settings are as shown, with an Extrude type as **Blind**, the direction pointing upward, and the extrude length at 5mm. Once this is all correct and the preview looks right, click on the green tick to generate the solid feature.

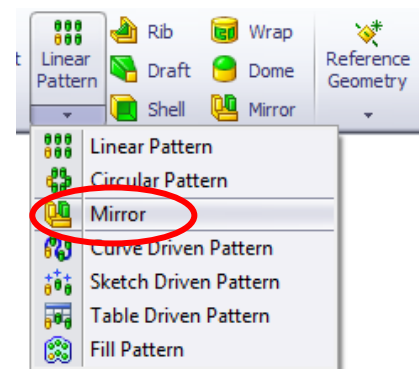


At this point we have some options for creating the boss on the other side of the engine block. We could create another sketch and another extrude feature, or we could sketch the second lug in the first sketch we did so both are created in the same extrude. Either of these will work, but are not good choices. The first option will not embed our design intent (that the left and right lugs should be identical and symmetrical about the central plane of the engine), thus a change to the lug dimensions will require two separate sketches to be modified and manually synchronised. The second option of sketching more entities in the first sketch, or using the sketch mirror functionality within the sketch, will overcome the design intent problem, but will result in a more complex sketch than it needs to be. As a general and very important principle, sketches should be kept as simple as you can make them. Complex sketches can become a nightmare to make changes to, so use whatever methods you can to avoid them! **It is nearly always better to create a simple sketch and then pattern or mirror the resulting feature, rather than perform patterning and mirroring within a sketch.**

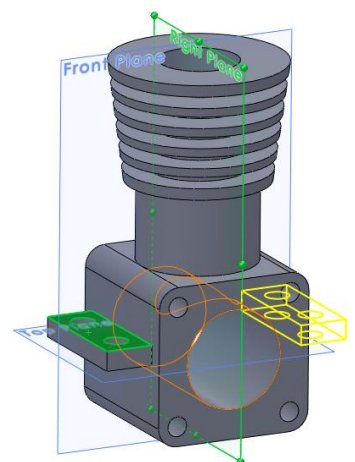
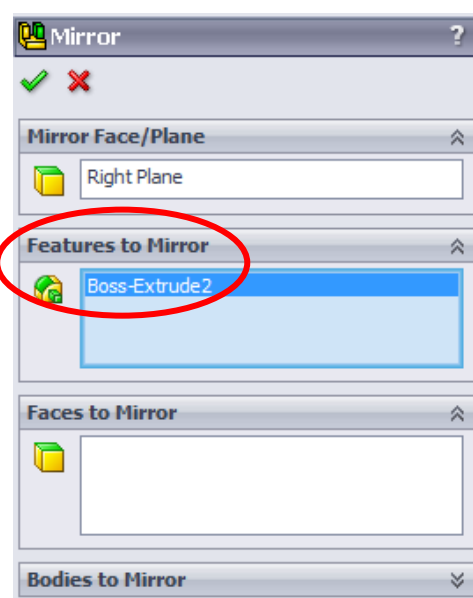


A better option is to use the **Mirror** tool to create a mirrored copy of the existing lug feature. Doing so has the advantage of embedding our design intent for the lugs (that the left and right lugs should be identical and symmetrical about the central plane of the engine), and keeps the model simple, robust, and easy to modify.

Select the **Mirror** tool from the dropdown menu under the Linear Pattern tool in the Features tab of the Command Manager.

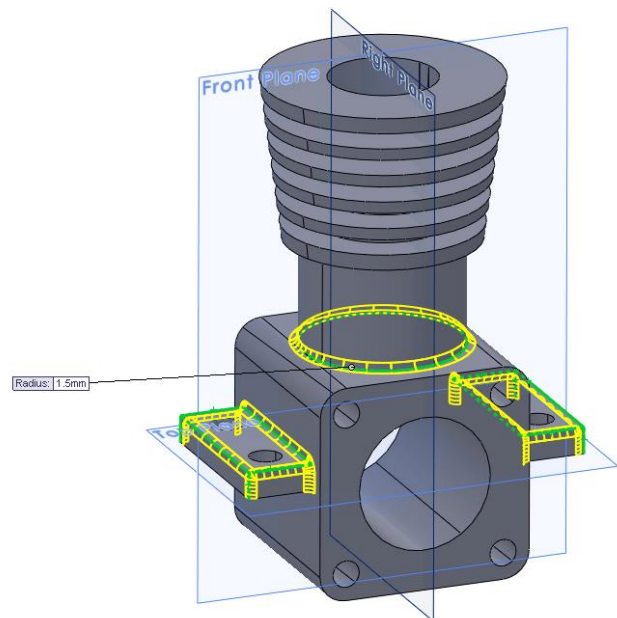
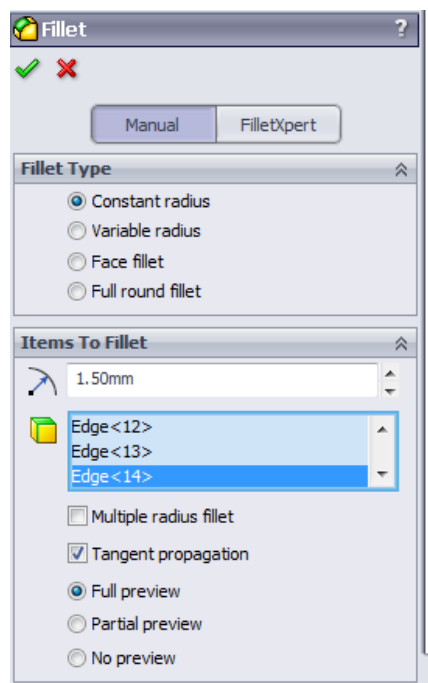


Select the Right Plane in the Graphics Area as the Mirror Face/Plane. Note how the mirror tool will work on either Features, Faces, or Bodies. There are three separate areas in the Mirror dialogue for selecting any of these. In this case we want to mirror a Feature, so select any surface that belongs to the first lug feature as the Feature to Mirror. The preview should look as shown, and clicking the green tick will accept this and generate the solid.



8. The last step we have planned for this part is to add some rounds (convex edges) and fillets (concave edges). In SolidWorks, for simplicity, these are both called fillets and the **Fillet** tool will create either of them. Fillets create smooth blends between features and remove sharp edges on them, so they make parts look nice and are practically important for manufacturing.

Select the **Fillet** tool on the Features tab of the Command Manager and enter a radius value of 1.5mm. Begin selecting edges on the model that you want to apply this fillet to. If you select something by mistake or change your mind, click on it again to deselect it. If you select a face on the model, all of the edges that belong to that face are selected. This can sometimes greatly speed up the selection process.



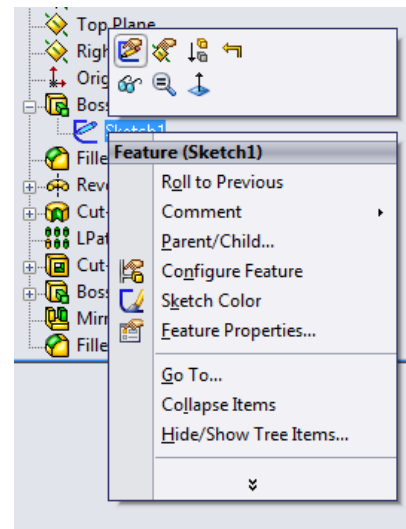
Once you have made all of the selections and are happy with the preview, click on the green tick and generate the fillets.

Editing sketches and features

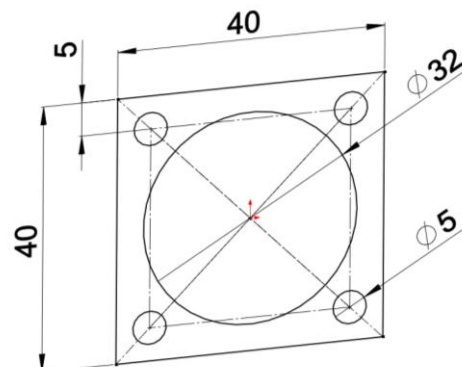
Very quickly, perhaps already it has happened for you, you will realise that changes are necessary after sketches and features are created. In fact making such changes is one of the fundamental virtues of CAD modelling. It is what enables it to be a tool for designing, rather than merely a tool for recording a design. In this exercise we'll take two examples of changes we want to make to the part at this stage, firstly we decide that the large hole through the centre of the crankcase needs to be bigger, and secondly we want to make the lugs a little thicker to increase their strength.

1. As discussed earlier, sketches appear to be absorbed into the first feature that uses them to create a solid feature, but they are still there and are merely being borrowed by that feature. To change the diameter of the hole through the crankcase we will need to go back and change the dimension which we put on the large circle in the sketch that was used to generate that feature. Click on the + symbol next to the **Boss-Extrude<1>** feature in the Feature Tree, to reveal the sketch which it is using.

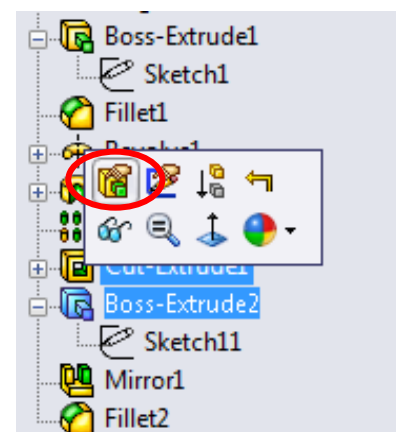
Right click on the **Sketch1** text and select the Edit Sketch icon in the top left corner of the popup. This will take you back into the sketch environment, just exactly as everything was when you first created this sketch and were about to accept/exit it.



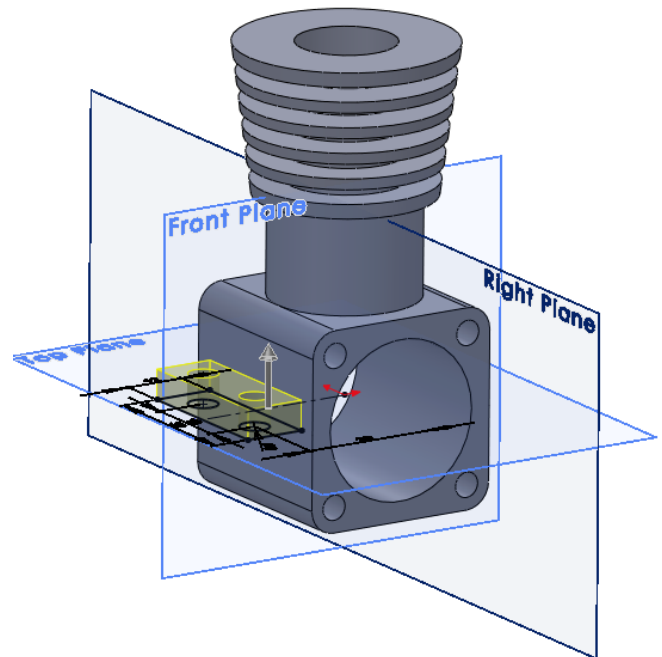
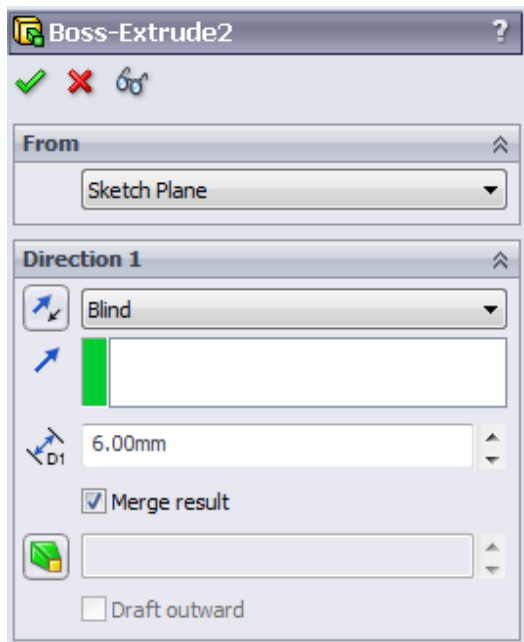
Double click on the diameter 25 dimension in the Graphics Area and enter in the new value of 32. Click the green tick in the **Modify** popup window, or alternately press the Enter key to accept the change. The sketch is now modified as we want it, so click on the Exit Sketch icon at the top right of the Graphics Area. The model now rebuilds through all of the features and the new solid has the larger hole through it. Any other feature that uses a sketch can be modified in the same way.



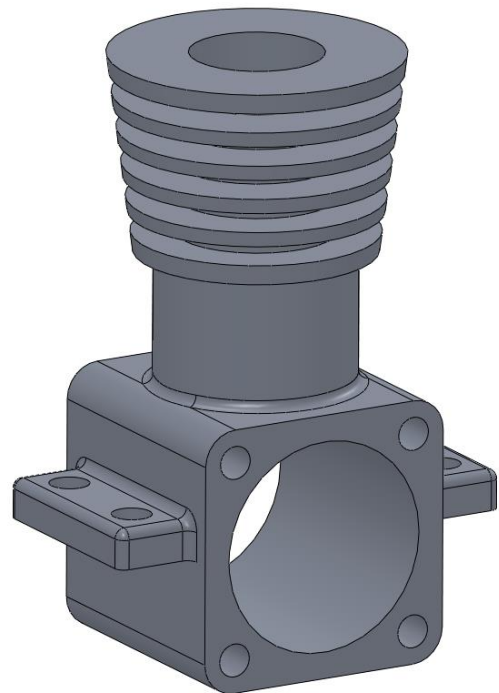
2. To modify the thickness of the mounting lugs is different to what we have just done because the thickness is set in the feature parameters rather than as a dimension in the sketch. In other words it is the distance the profile is extruded rather than how big the profile is. However the process is quite similar to modifying the sketch. This time right click on the feature itself in the Feature tree, rather than its sketch, and select the top left (Edit Feature) icon in the popup menu. This, same as the sketch editing, works like a time machine and takes you back to exactly as things were when you first created this feature and were about to accept/exit it.



Change the **Direction 1** value to 6 and press the Enter key. Note that the preview of the lug increases in thickness slightly. This indicates that all is going as intended, so click on the green tick at the top of the **Boss-Extrude2** dialogue to accept this and generate the new geometry. The whole model rebuilds, with any subsequent features (such as the fillets on these lugs) rebuilding in a slightly different position.



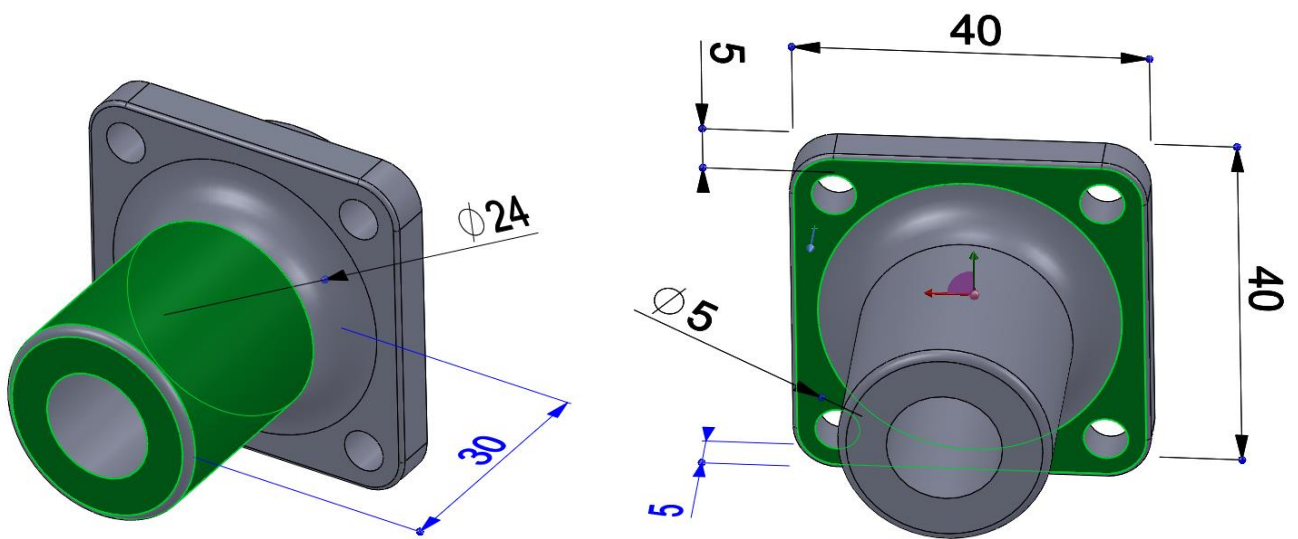
The model is now as we want it for now, so save the file. If you would like to admire the finished part without the planes in the Graphics Area, click on the **Planes** item in the View menu in the top Menu Bar. All of the different visual entities listed in the menu can be shown or hidden by toggling them in this manner. Also it is possible to turn all of them off and on with one selection of Hide All Types, which is often convenient.



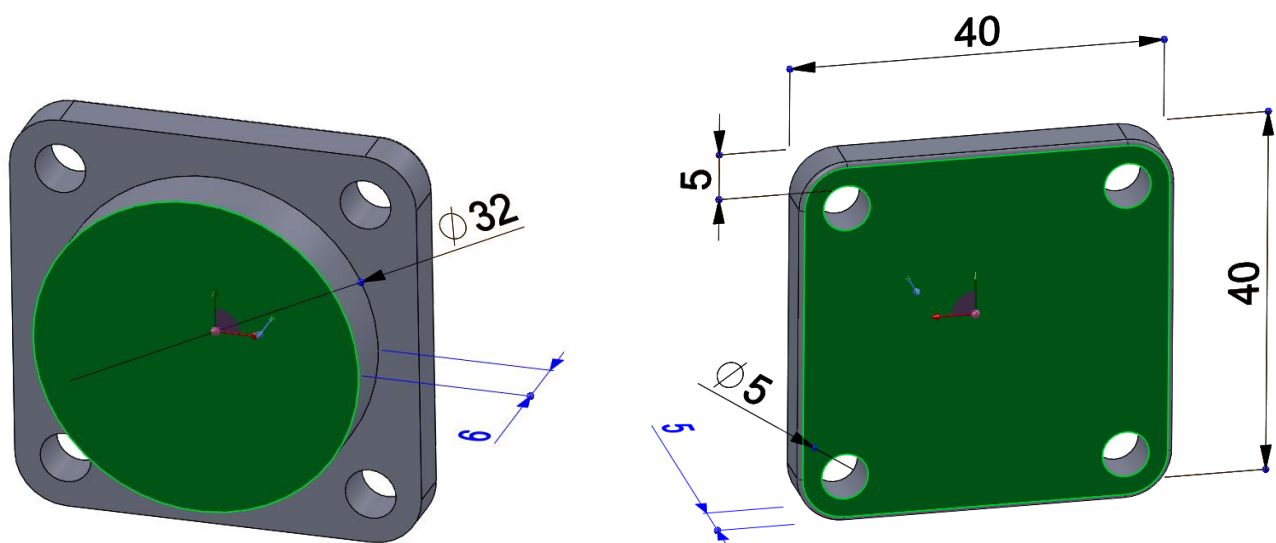
Exercise 4: Modelling some more parts

You have now had an exposure to the basic elements of part modelling, and should be able to independently model some more engine components. Complete these parts using the same workflow as you have used on the engine block. Most of the dimensions are given, but if any are missing you should use your judgement to make these up. The actual dimensions are not nearly as important as getting practice at creating the parts, sketches, and features, so don't spend too much time concerning yourself with them.

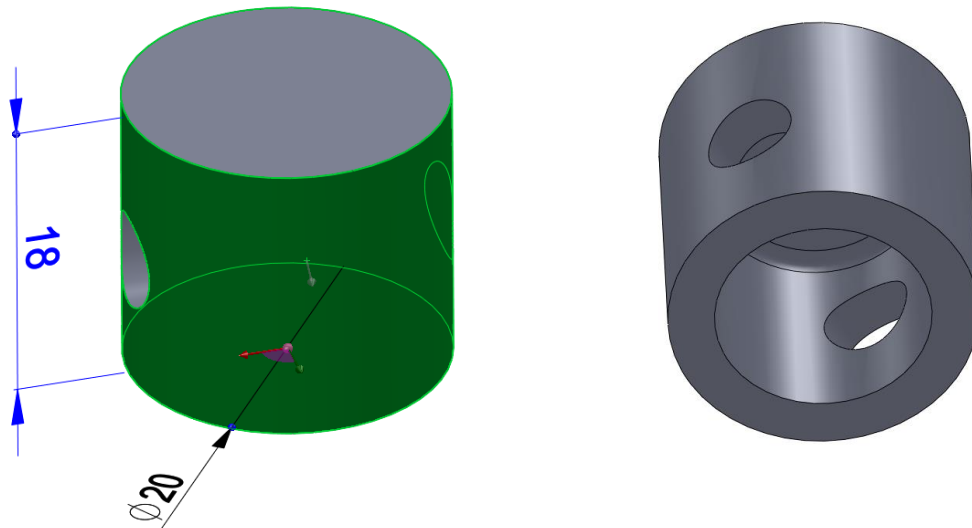
1. Front cover



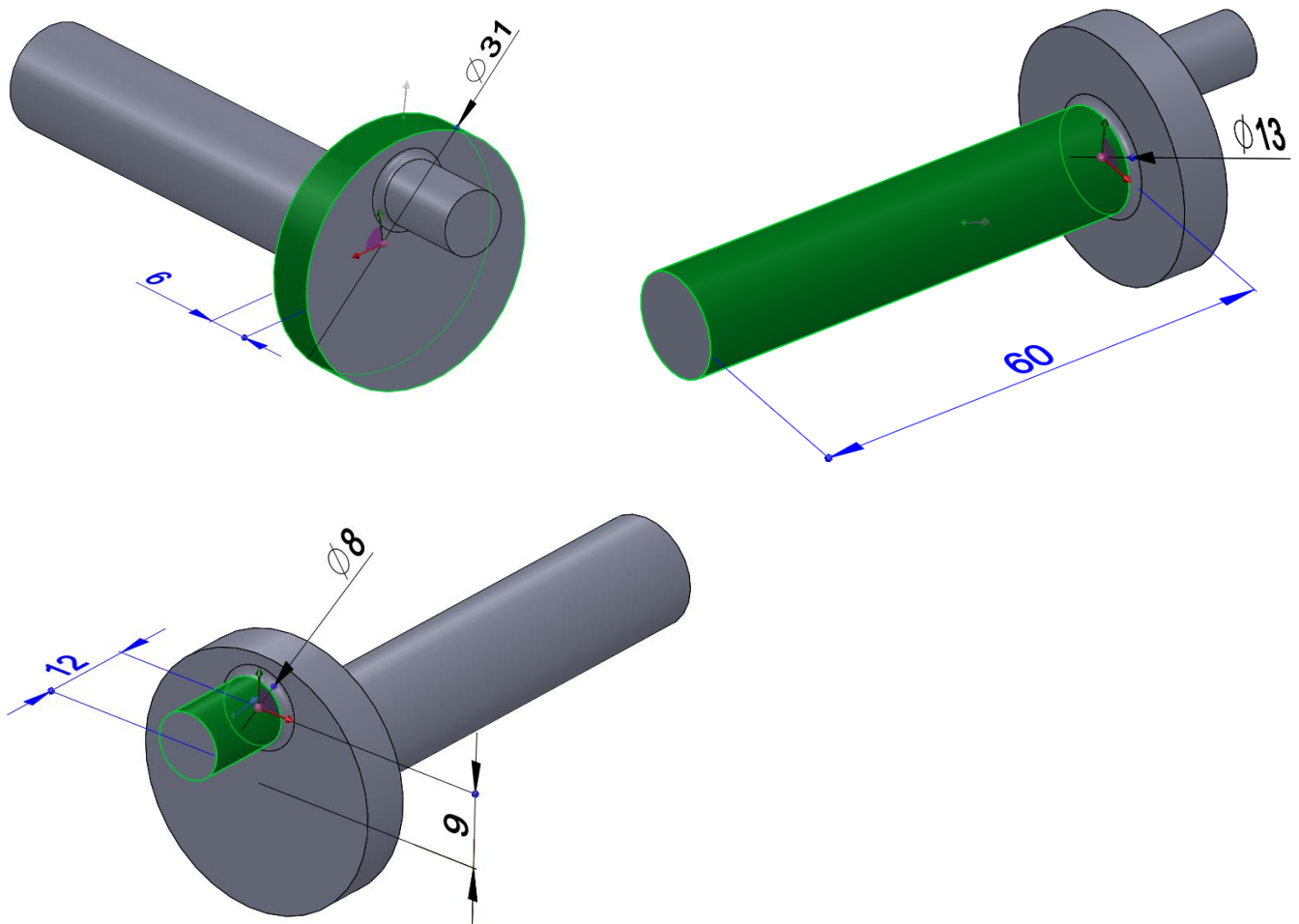
2. Rear cover



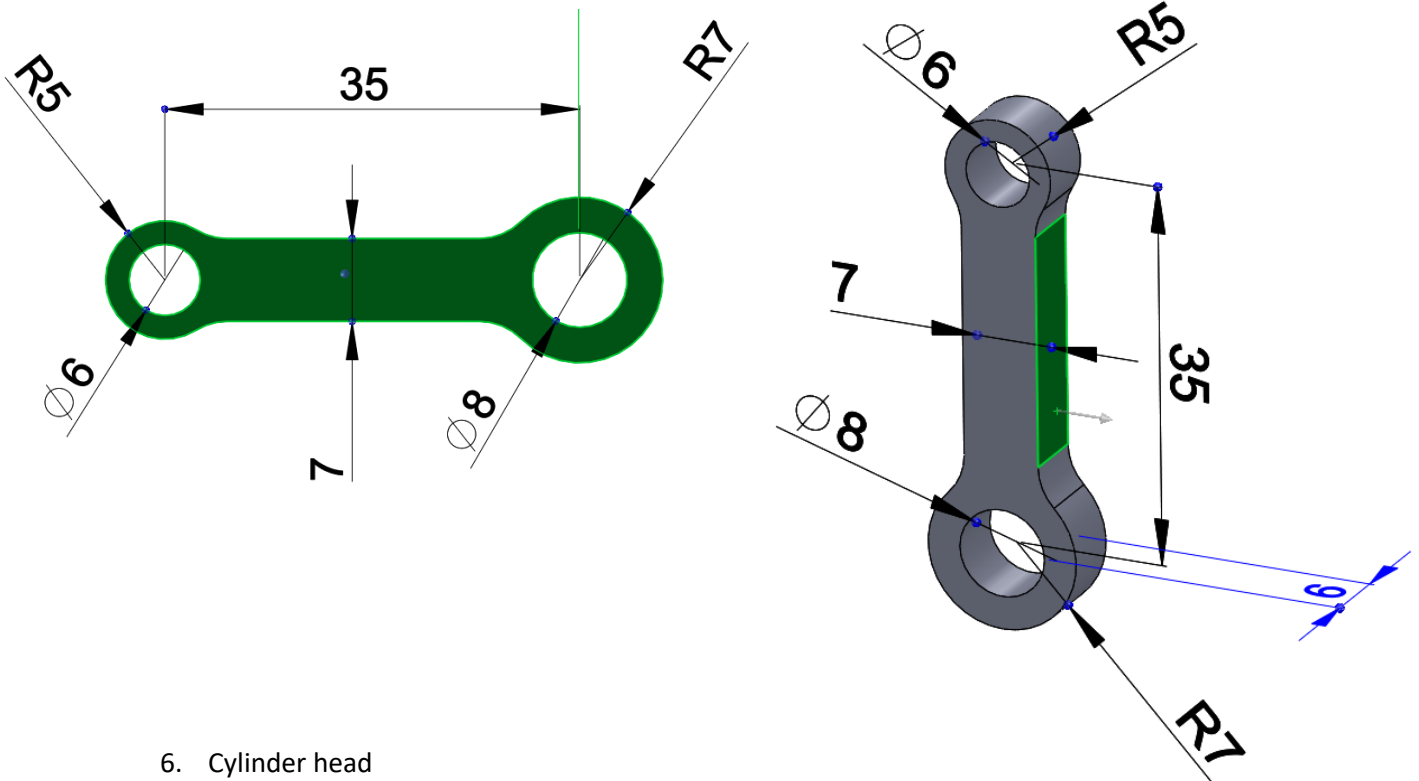
3. Piston



4. Crankshaft



5. Conrod



6. Cylinder head

(Model shape as shown, dimension to suit other parts)

