

## TUTORIAL 3 CAD Modelling of Complex Shapes

In the first tutorial we were dealing primarily with parts that can be created mostly with extruding, revolving, and filleting. We saw that using combinations of features created by these tools can create some interesting part models, however you won't have to look far around you to see that most manufactured products have more complex shapes. Take for example a computer mouse, a drink bottle, or even the ubiquitous BIC pen. In this tutorial we will continue with the design of our model race car by modelling the body, and in doing so will explore some great tools that SolidWorks provides for creating complex shapes.

### Goals

- Understand the difference between solids and surfaces in CAD models
- Extent sketching skills, in particular with creating and modifying splines
- Introduction to a workflow for generating shapes from hand sketches or photos
- Learn about tools for working with surfaces and how to create solid parts from surfaces

### Understanding the job and establishing the modelling workflow

Perhaps even more so than when we made a plan for modelling the cylinder block in the previous tutorial, for a model such as the car body it is important to know how you are going to go about it before starting. Depending on the project there will be different starting points. They may be hand sketching with a pencil and paper, there may be existing renders created by an industrial designer, or it may be simple enough that you can start directly sketching in Solidworks and edit that sketch to get your desired result. For us we will take photos as a starting point, then create some hand sketches based upon these, and finally import these into SolidWorks to build the CAD model using these as a guide.

The inspiration for this car body is the 1934 Ford hot rod as seen in this photo. It's got some interesting shapes to model, and will fit out project perfectly.

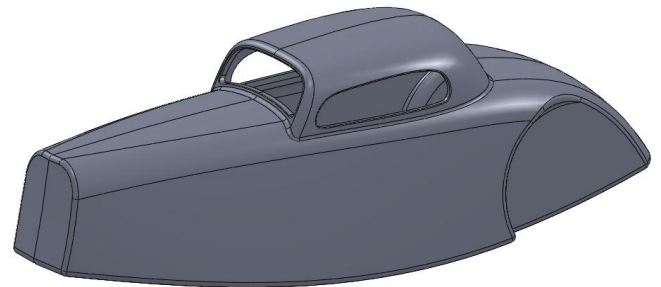


Existing products are a great source of inspiration when creating something new, so don't be afraid to spend time looking at photos on the internet to get ideas and direction on whatever your project is. In this case it is a retro look we are after, so the choice of shape is almost completely dictated by what was done in the past.

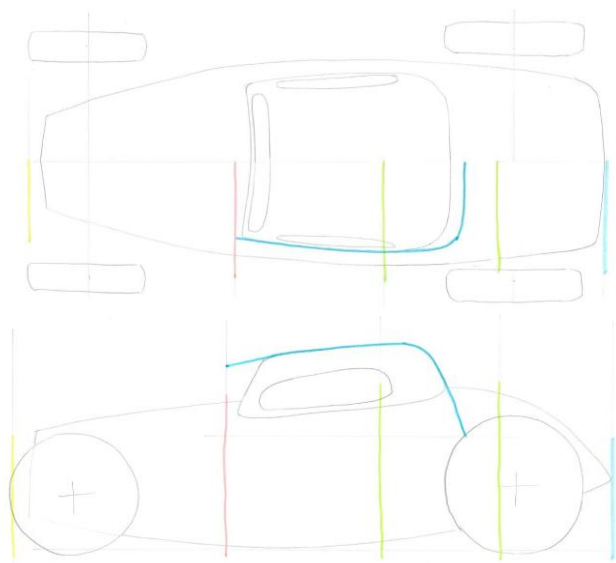
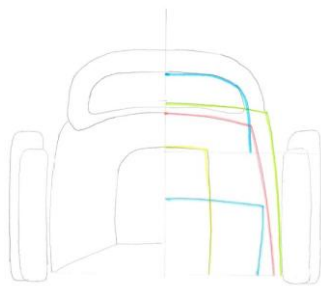
Ideally we would have liked to find a photo of this same car from the front and top, so that we could see more of its shape, but the best available was a front three quarter view. This gives us enough information for us to work with, and just means we will need to create the views ourselves. The nice thing about modelling something like this is that if it looks right then it is right! In the spirit of this, we will change the nose of the car a little to suit a different style (and to make it a little easier to model).



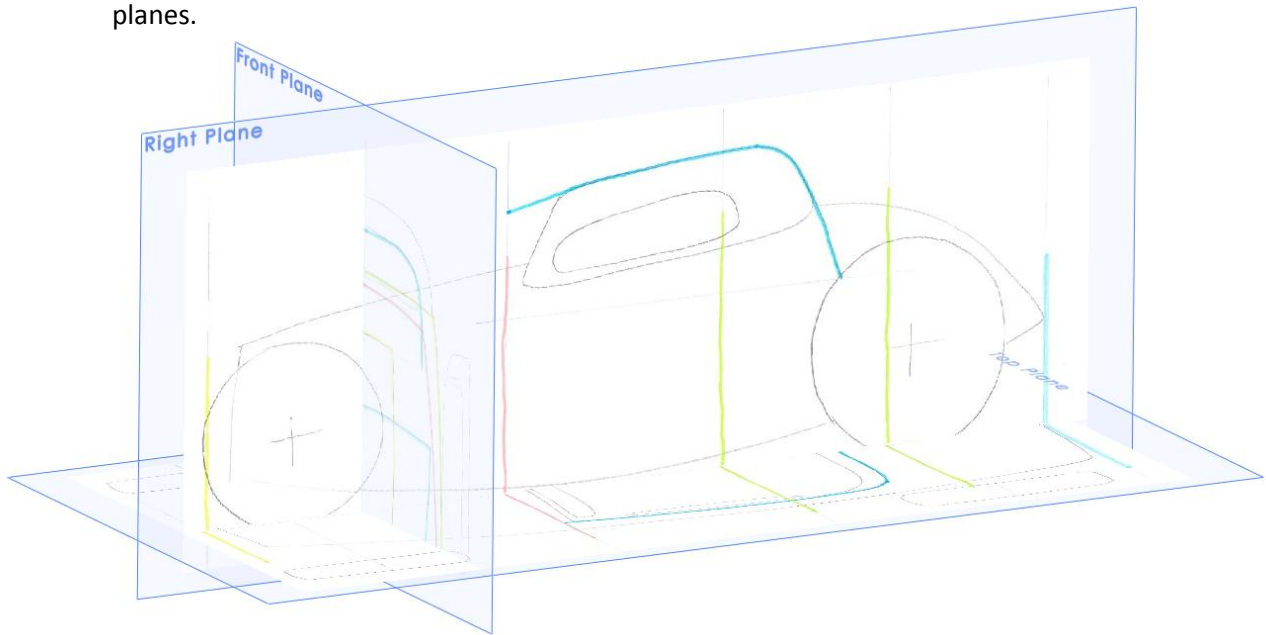
As a sneak peak of where we are heading, the finished model will look like this. We'll now go through a step by step plan of how to get to this point.



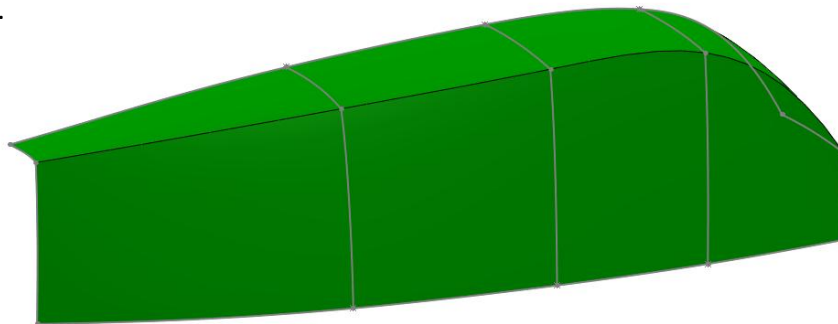
1. Using the photos as a guide, create three hand sketched views of the subject, with all of the important lines which define the shape. The colored lines are a guide that will help us later



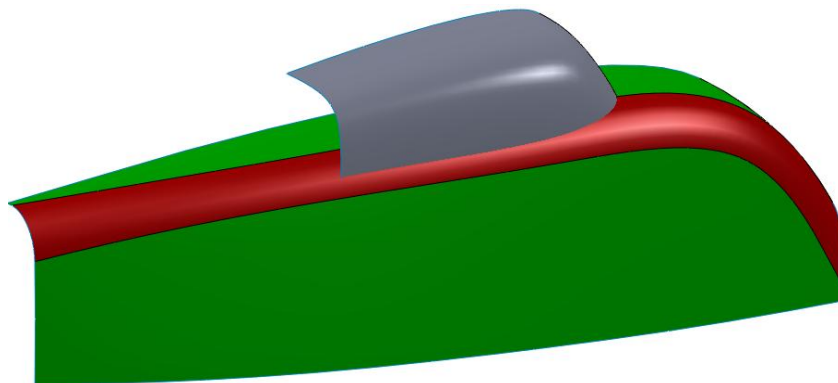
Insert the hand sketches into SolidWorks as sketch pictures on the top, front, and side planes.



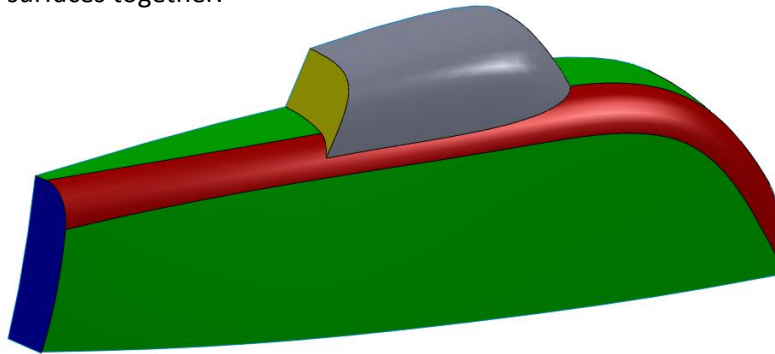
2. Create a series of planes and sketch profile along the length of the car, and then loft a surface to create the main panels of the car (like using these sketches as a frame to put a sheet over).



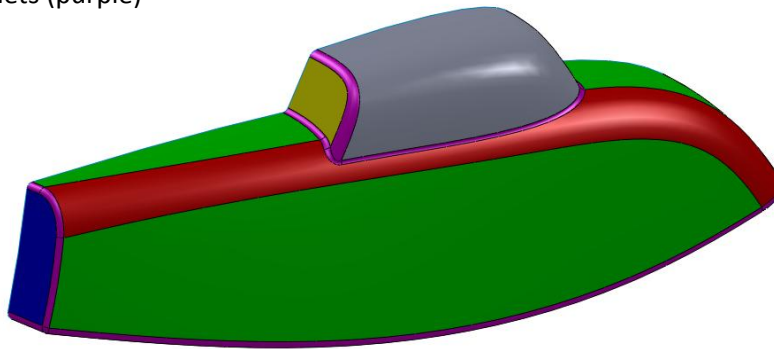
3. Fillet (red surface) along the length of the body, and then use another loft to create the roof section of the car (grey surface).



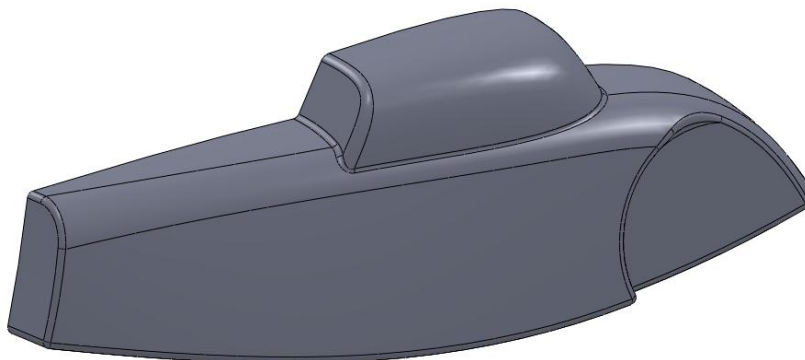
4. Extrude some surfaces to create the front of the car (blue) and widescreen (yellow) and join all of the surfaces together.



5. Create the shape of the underside of the car using an extruded surface, then apply some more fillets (purple)



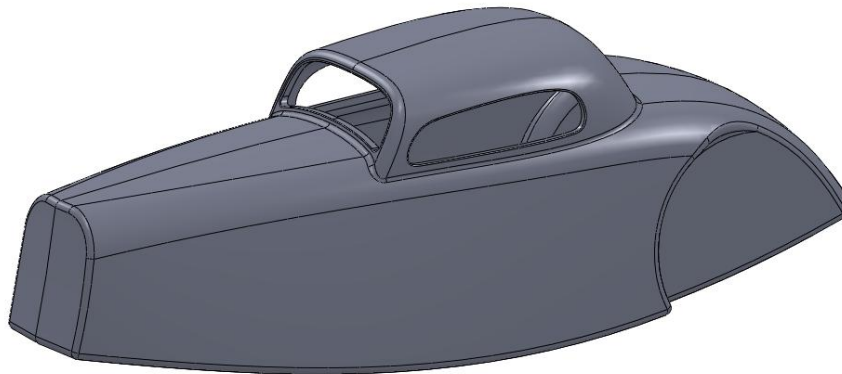
6. Change the surface model into a solid model and use a cut extrude to make the rear wheel cutout. Apply some fillets to this wheel cutout.



7. Perform a shell operation to turn the solid part into a thin shell (imagine this is half a watermelon, and you scoop out all the delicious inside, leaving only the shell). Use some extrude cuts to create the window and windshield holes in the shell.



8. Mirror the part to complete the body model

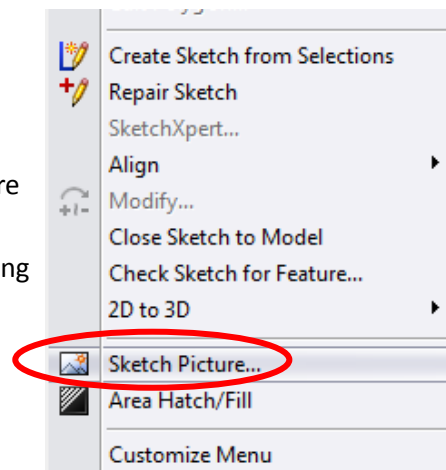


To complete this first part we'll complete step 1 of our plan, which is to get our sketches into a model. Because this is a little time consuming and repetitive a starting model is provided with this tutorial that has these sketches included already, but here we will go through the description of how to put in the side sketch as an example if you want to do it yourself.

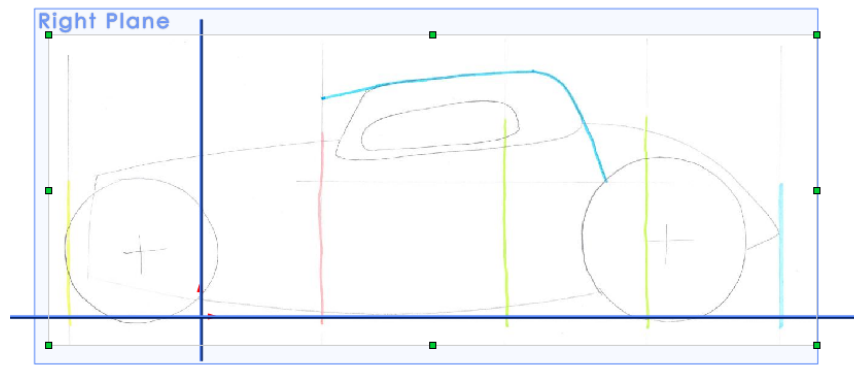
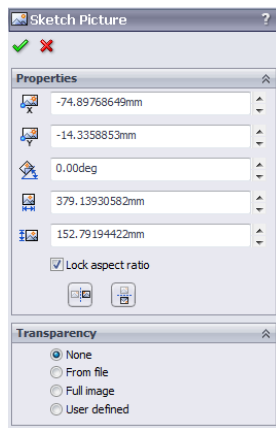
Select the Right (side) plane in the Graphics Area and then select the sketch tool to create a new sketch on this plane.

Select the Sketch Picture tool, which is found in the Sketch Tools submenu of the Tools menu in the Menu Bar at the top of the screen.

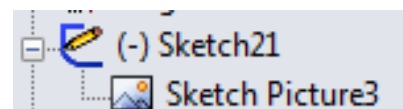
The Sketch Picture tool allows you to select from a variety of image file types (such as JPEG, BMP, etc). This image will then be inserted onto the sketching plane, where it can be resized, rotated, and have its transparency adjusted. Sketch Pictures are a really useful way of interfacing CAD models with the real world at the beginning of the modelling process, as we are doing here.







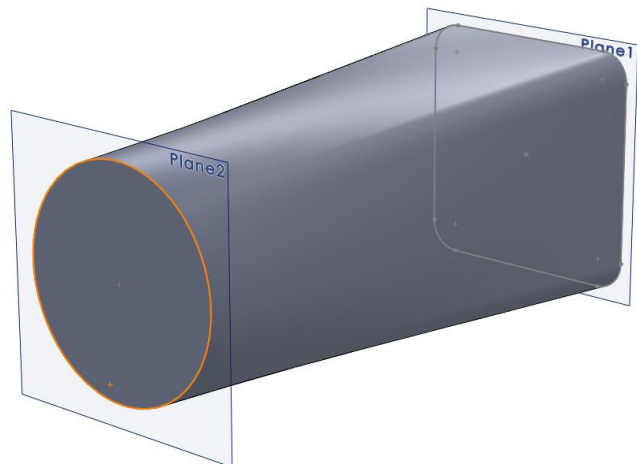
Notice that the Sketch Picture is incorporated into the Sketch (click on the + next to the sketch icon to expand it in the Feature Tree), and in fact if you want to you can do some SolidWorks sketching in this Sketch as well. In other words it is just a normal SolidWorks sketch, but it also contains a picture.



## Modelling the Car Body

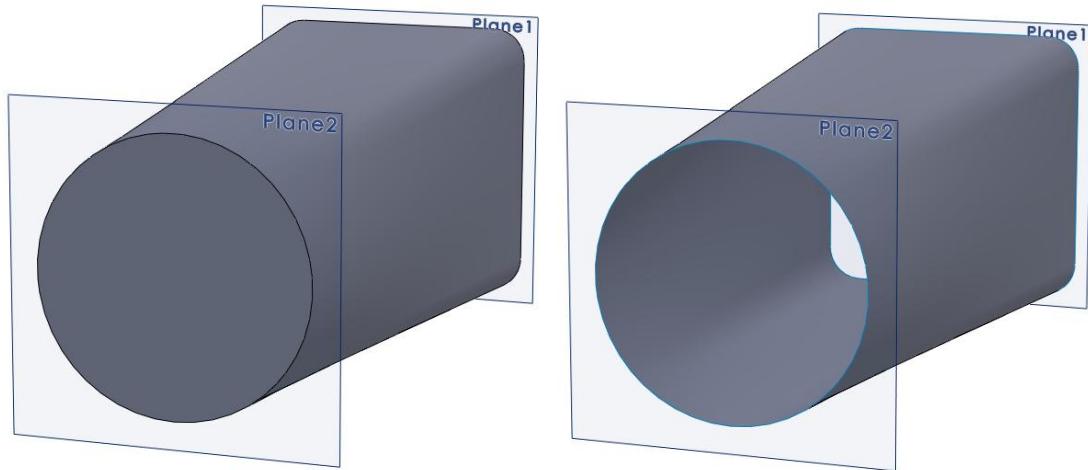
- I. The main part of the body will be created using a loft feature. A loft is a series of two or more profiles (sketches) separated by some distance, with a surface morphed between them.

The example here shows a lofted solid between a circle and a rounded square.



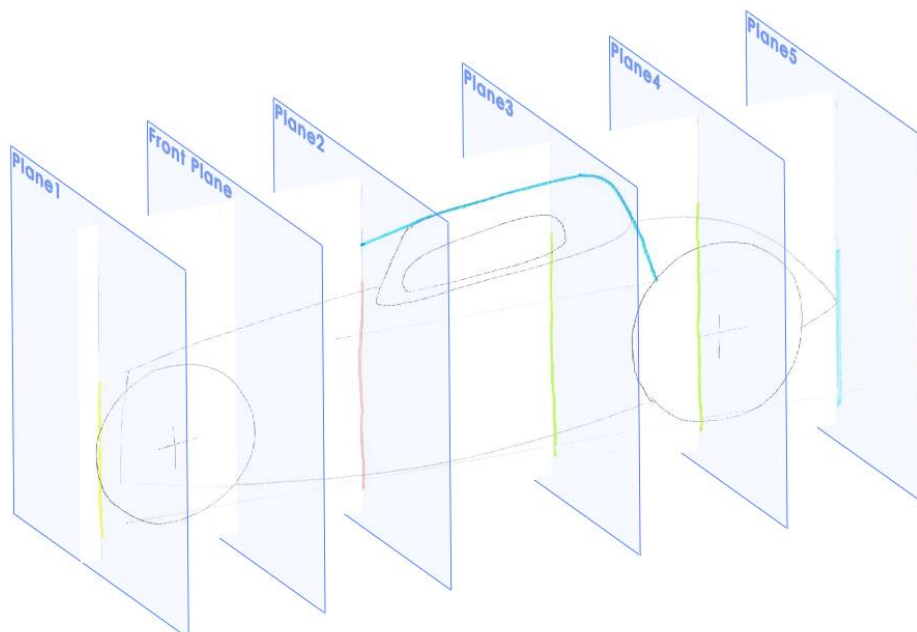
For our car we want to use the loft to create a surface feature rather than a solid feature. This is the first time we've come across the concept of a surface as opposed to a solid, so we'll look at the differences between them. A solid feature is one where we have a set of surfaces that are joined together to form a closed volume. Surface features are single or sets of surfaces that do not form a closed volume. The surface feature has no thickness. In solid modelling surfaces are useful to work with because they simplify the job of creating complex shapes. If a particular surface of a part is quite complex, we want to be able to focus all of our attention just on that particular surface. For example if you were designing the top surface of a car, you don't want to have to worry about the underneath of the car at the same time!

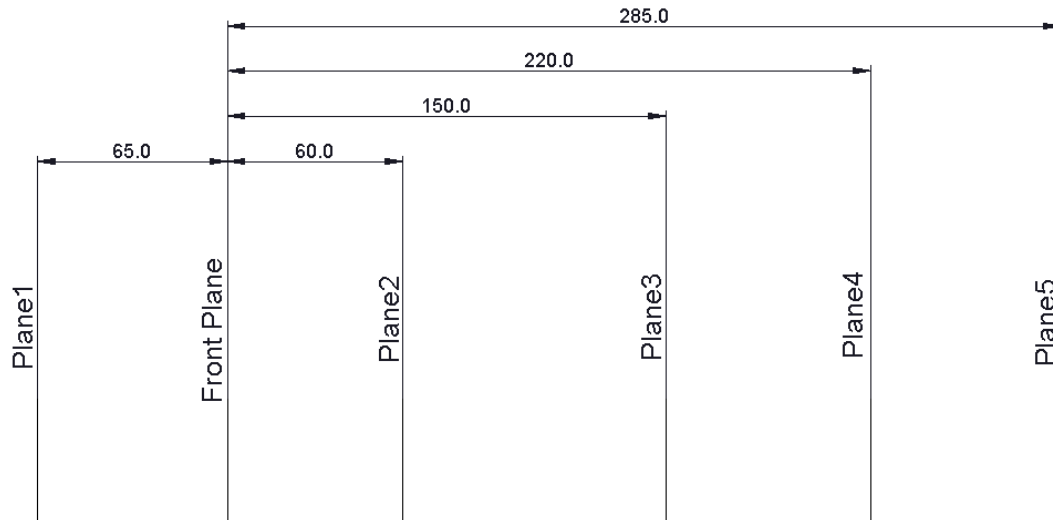
In the example below, our circle to rounded square example is performed using the standard loft feature on the left (creates a solid), and as a surface loft on the right (creates just the surface that lofts between the shapes, with no ends. If you look carefully you can see the surface loft has blue edges on the end of it, this signifies to you that this is the edge of a surface. The blue edge is called a *free edge*, because it is not joined to any other edges, as opposed to the black edges on the solid part, which are edges that belong to two surfaces that are joined together.



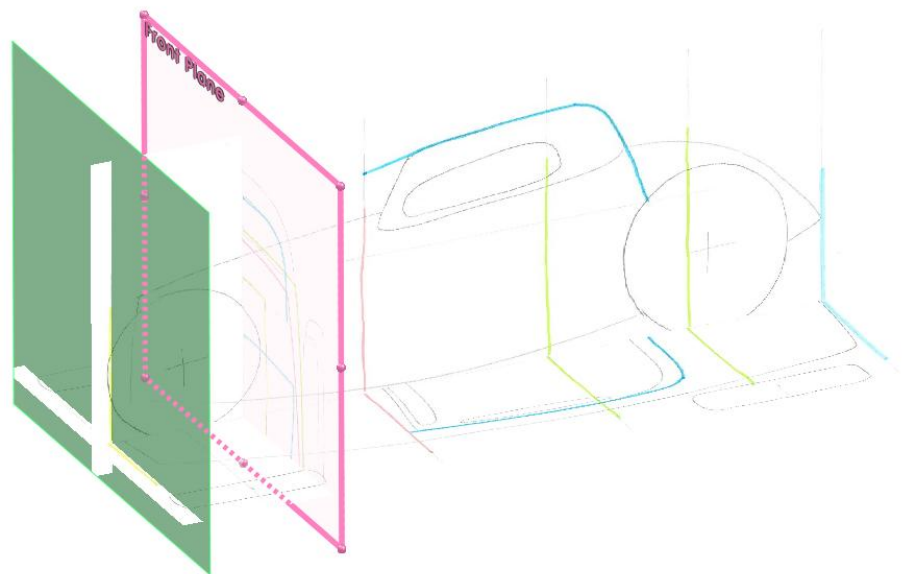
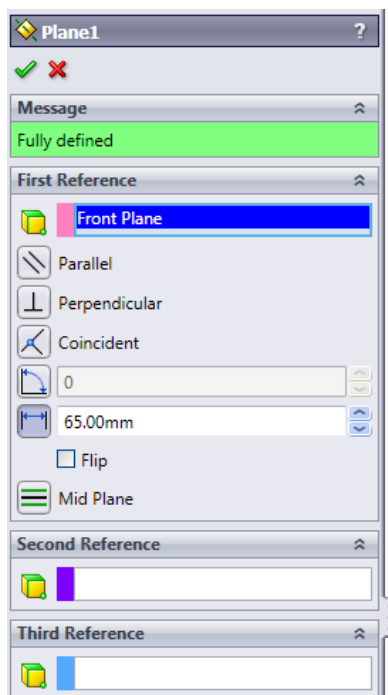
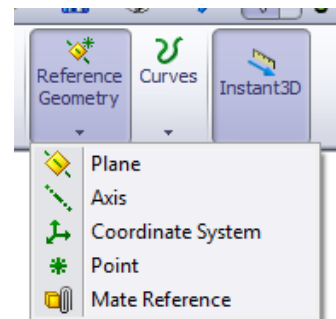
Surfaces can be made into solids by a number of ways (thicken tool, using the surface to cut from a solid block, or merging surfaces into an enclosed volume = becomes a solid). Any solid can be turned into surfaces just by deleting one or more of the faces of the object. The remaining surfaces no longer form an enclosed space, so they are no longer a solid. With experience you will be comfortable with these concepts, so now let's start to put them to use now.

Open the part file body 34 Ford.prt which already has the hand sketches embedded in it. Start setting up the loft by creating planes to do our profile sketches on. Place these as per the dimensions on the next page so it will look like the figure below.





Create the first plane by selecting the Plane tool from the Reference Geometry dropdown in the Features tab of the Command Manager. The Plane tool is used frequently as it allows you to sketch in places where there are currently no planes or flat surfaces to sketch on.



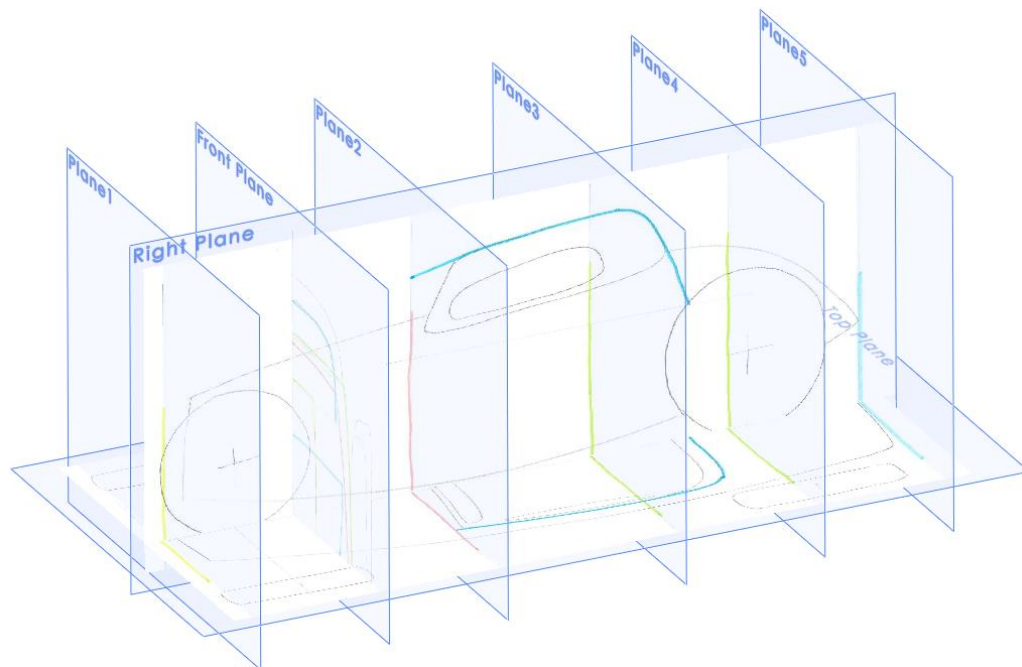
There are many ways of defining the position of a new plane by referencing existing geometry (planes, axis, surfaces). Once the tool is active SolidWorks will attempt to predict your intent as you select existing geometry. The most simple case for a plane is to create it offset from another plane. Select the Front Plane and the software will assume that this is what you want to do. Enter in a value of 65 in the dimension box and press Enter to generate the preview. If it looks OK then press the green tick at the top of the Plane dialogue



box. If the preview shows the plane has gone in the wrong direction (toward the back of the car body) click in the Flip box just below the dimension box and the direction will be reversed.

Repeat this process for each of the other 4 planes, being careful that they are all offset from the Front Plane by the dimensions shown. These values will be different for each job you work on, and are determined by experimentation when modelling. The general principle is to try to work with as few sketches as possible in a loft, with the desired shape still being achieved. So start with only two or three, and then add more if necessary. For the purposes of the tutorial however, it's best we just stay with what we know will work as it can take a lot of time to work through this process on real jobs.

Your model should now look as follows, noting that the planes approximately line up with the colored horizontal and vertical lines on the top and side sketches of the car. We'll now go about making the sketches so the loft can be generated.

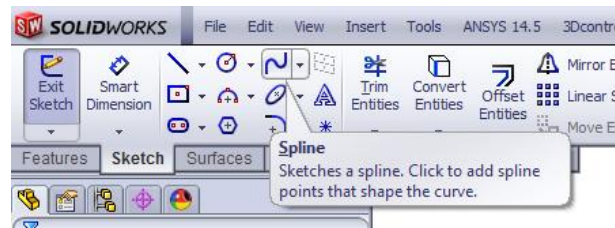


Sometimes a loft can be helped by sketches that go along the length of it to guide the surface as it lofts between the profiles. Unsurprisingly these sketches are called Guide Curves, and we will make two of these now, one on each of the Right and Top planes.

Start a new sketch on the Top Plane by selecting it in the Graphics Area and clicking on the Sketch tooling in the Sketch tab of the Command Manager. Align the sketch so it is 'flat' in the plane of the screen (hit space bar, double click Normal To on the popup window).

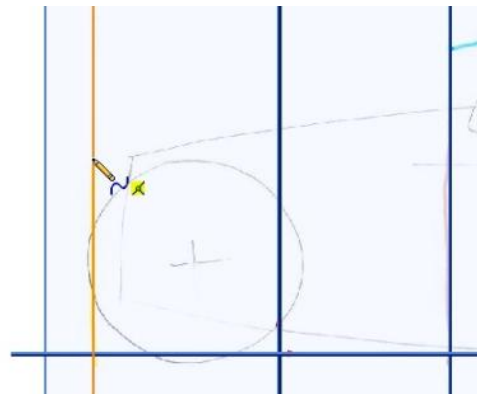
For the following few sketches we want to create curved sketch lines to match the shape of the object we are modelling. These lines are called splines, and are very commonly used in CAD modelling. They can take a little practice to get comfortable with, so perform each of the following steps exactly and hopefully you can create your first spline without any trouble. If you can do this OK, then you can manage any spline.

- I. Click on the Spline tool in the Sketch tab of the Command Manager.

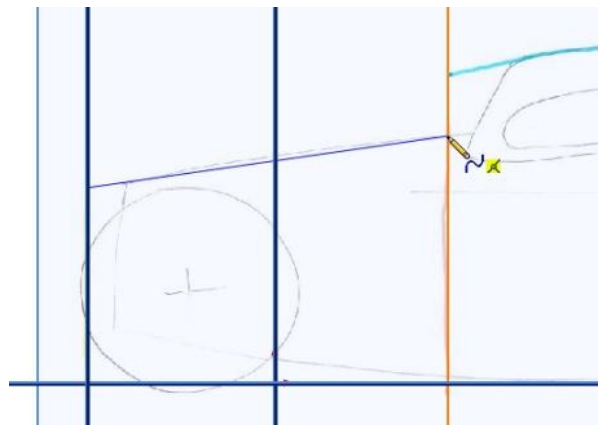


- II. A spline is defined by clicking in the graphics area, moving the mouse along, clicking again, moving the mouse, clicking again etc. Each time you click you create another point on the spline. You keep doing this until the length of the curve you want is as long as you want it, and then press the Esc key to finish the spline. The more points you define along the spline the more control you can have over the shape of it, but the harder it is to get it flowing smoothly. In our case we want to have the start of the spline on Plane1, the next point on the Front Plane, then on Planes 2,3,4,5.

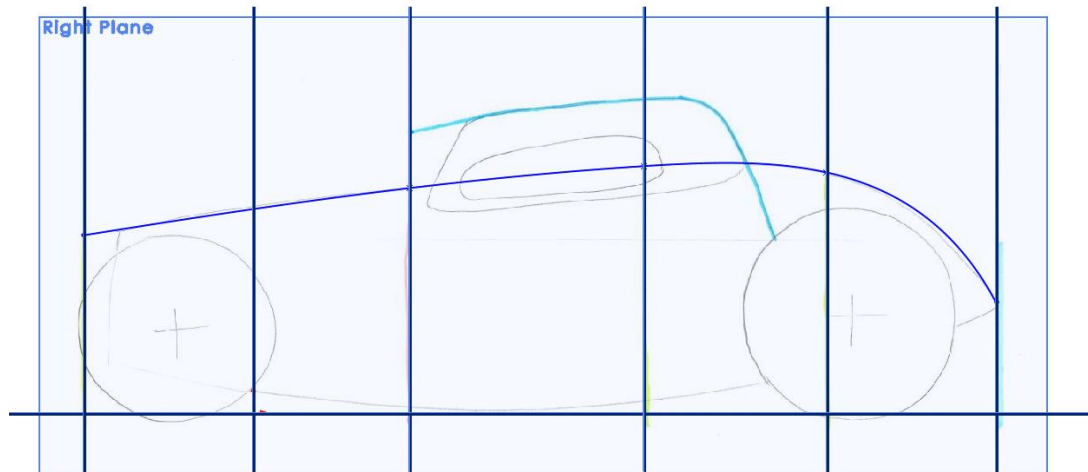
Do this by moving the mouse close to Plane1 approximately lined up with the where the top surface of the car would intersect it, as shown. As the spline tool pointer moves close to the plane you will see the plane being highlighted and the coincident sketch relation will appear. This indicates that clicking here will create a coincident relation between the spline point and Plane 1. This is what we want, so go ahead and click.



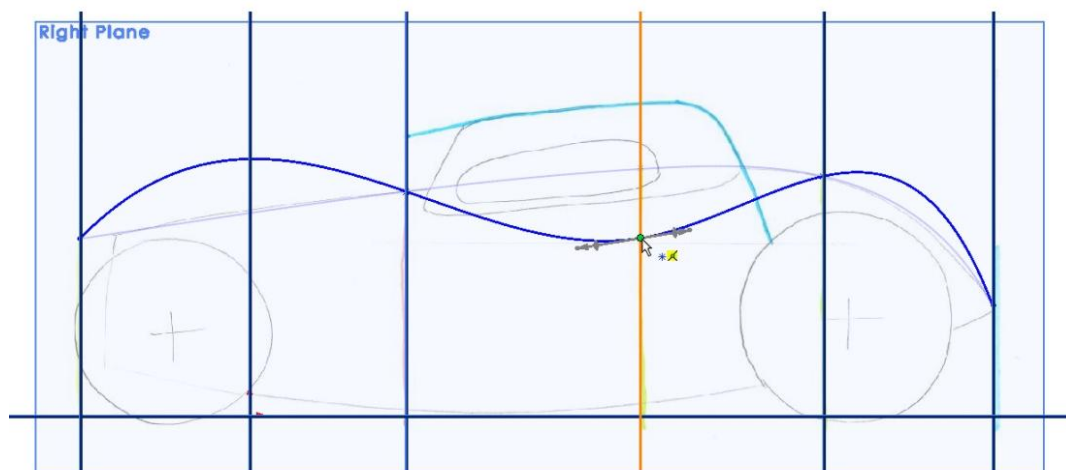
- III. Move the mouse across to the right and you will see a line being created between the first point and the pointer location. This is the spline, but it is straight until more points force it into a curve. Keep moving until the pointer highlights the edge of the Front Plane (and you can see the coincident sketch relation icon appear). Make sure the pointer is also at the location of the line on the top of the car bonnet (this is what we are 'tracing' over).



- IV. Carry on the same thing with planes 2,3,4 and 5. Move, click, move, click etc. After clicking on Plane 5 the spline is complete, so press the Esc key to end that. If you were to click again after pressing the Esc key then you will be starting a new spline, but we don't want to do that in this instance.



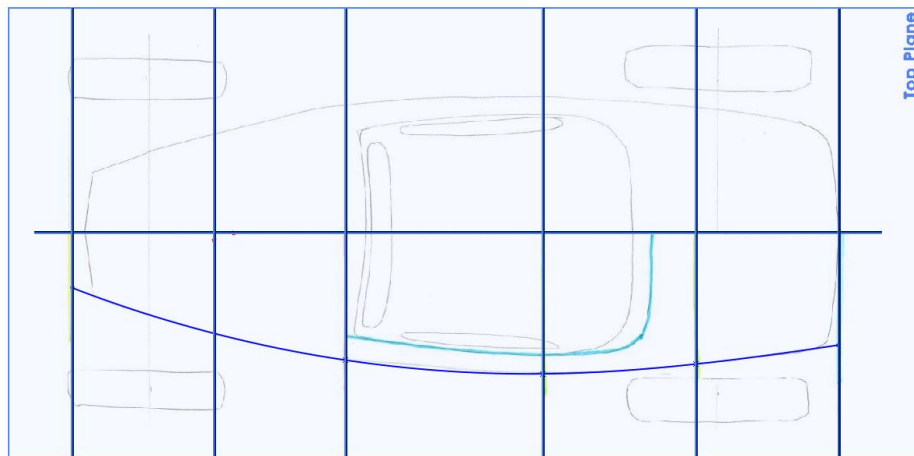
The finished spline should look similar to the image above, but if it doesn't it is easy to fix. Like anything in SolidWorks, editing is at least as big a part of the job as the initial creation and is what makes it such a powerful tool compared to the days of pencil and rubber. To test this, try clicking (and holding the mouse button down) on the spline point on Plane3, and dragging it. The first thing you will notice is that it should only move vertically due to the sketch relation making it stay on Plane3. The second thing you will notice is that moving one point changes the shape of the spline a lot!



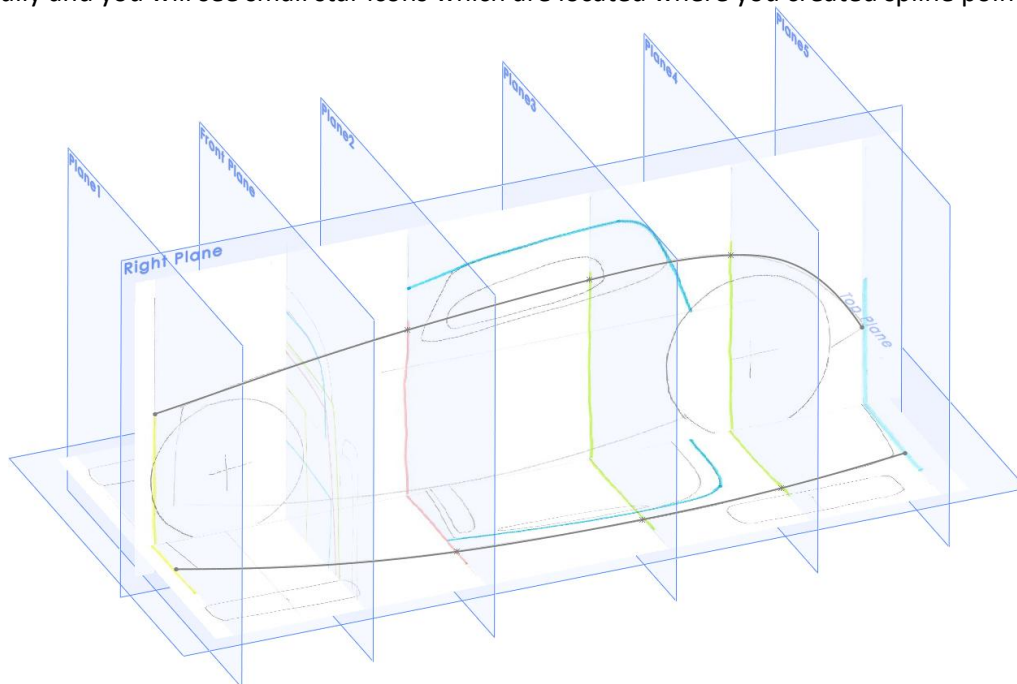
If you let go of the mouse button then that will be the new location for the spline point. Experiment with this a little bit and get a feeling for it. And then put the spline points back to where they should be, tracing over the top edge of the car hand sketch.

When you are satisfied that the spline looks good enough (it doesn't have to be precise), exit the sketch.

Now that you are able to create and edit a multiple point spline, go ahead and create another one on the Top Plane. This will be used as our second Guide Curve for the body loft. The process is exactly as you have just done on the on the Right Plane, starting the first point on Plane1, then clicking on planes 2,3,4 and 5. Make sure the spline points are snapping onto the planes with the coincident relation, and that the curve is approximately tracing over the hand sketch.



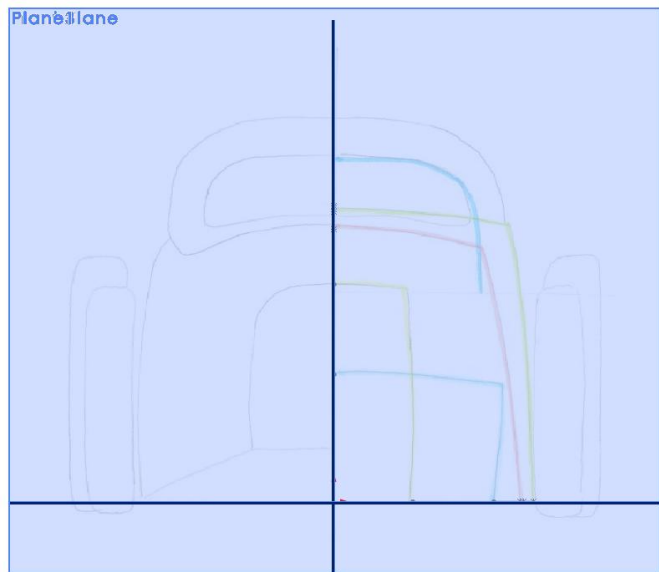
Once you are happy with the shape of this spline, exit the sketch. Rotate the model a little bit (middle mouse button held down and move the mouse) so you can see both of your spline sketches together. They are both grey because the sketches are not active (no in the sketching mode). Look carefully and you will see small star icons which are located where you created spline points.



As we will see soon, these guide curves will be useful for influencing the shape of the loft, but they are not essential. The essential sketches are the loft profiles which we will create now. We'll put on on each of the five vertical planes (excluding the Front plane). Notice on the hand sketches there are colored vertical lines on the Right Plane and colored horizontal lines on the Top Plane. These have been put there to guide you as to which of the profiles should be sketched on each plane. For example Plane1 has yellow lines on it, so we will sketch on here and copy the shape of the yellow sketch on the Front Plane hand sketch. Plane2 has a red sketch so we'll trace over the red one etc. The only trick is that planes 3 and 4 are both green, but this the sketches on both of these are near enough to identical, so we'll trace the same one on to both.

We'll start with the sketch on Plane1 and go through it in some detail. There are quite a few subtleties in what we are about to do, but work through it carefully and you will soon find it very fast and easy to work with splines.

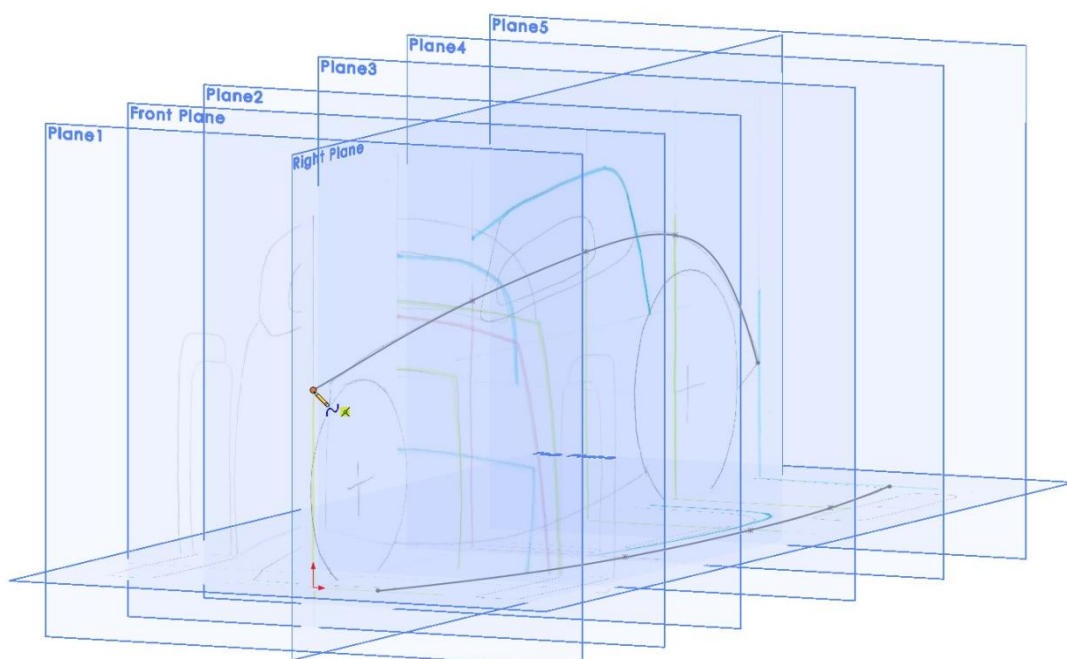
Select Plane1 in the Graphics Area and select the Sketch tool to open a sketch on this plane. Orient the plane so it is parallel to the screen. In this view you should see the colored profiles in the hand sketch. The yellow one is the one that correlates to Plane1, so we'll use the spline tool to trace over this one.



Splines create smooth curves, and because we have a sharp corner in the sketch we are tracing over, we'll make the sketch with two splines. The first will trace the more or less horizontal bit, and the second the more vertical one.

### Sketch the first Spline as follows...

Select the spline tool, and we want to create the first spline point on the left end of the yellow line, and importantly we want the point to form a coincident sketch relation with the end point of the guide curve we have created on the Right Plane. In order to be able to do this we need to move the model around on a bit of an angle (hold middle mouse button down and move the mouse). If we don't do this the spline pointer won't be able to 'see' the point we want it to snap onto because there are other model entities in the way. Rotate the model around as shown, and click when you see the guide **curve end point highlighted (orange)** and the coincident relation icon appear. This tells you that the new point will have a coincident relation created with the guide curve end point.

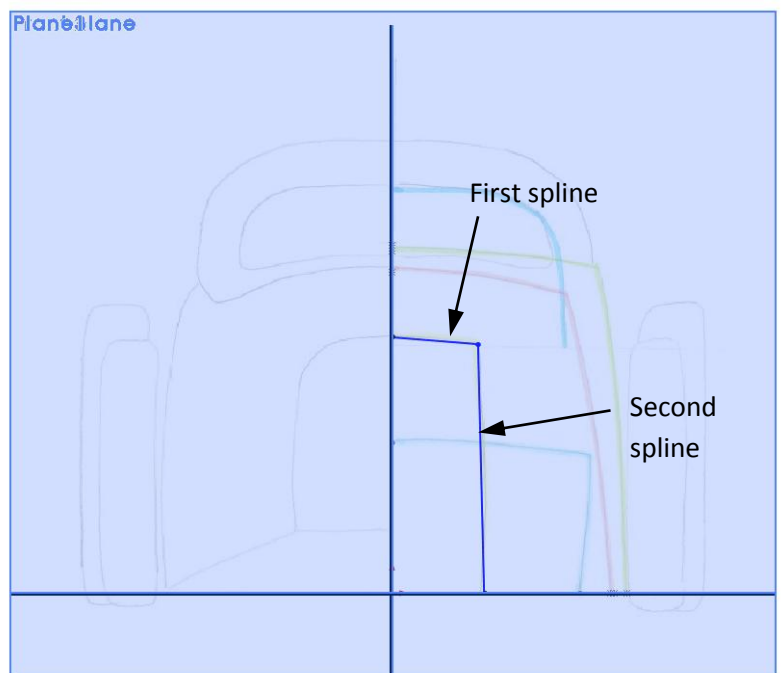




With the start point created, rotate the sketch back around so it is parallel to the screen. Do this either manually by holding the middle mouse button down and moving the mouse, or by pressing the space bar and selecting Normal To in the popup window. Even though you are part way through creating a spline, you can still perform these orientation operations, which are vital for situations such as these.

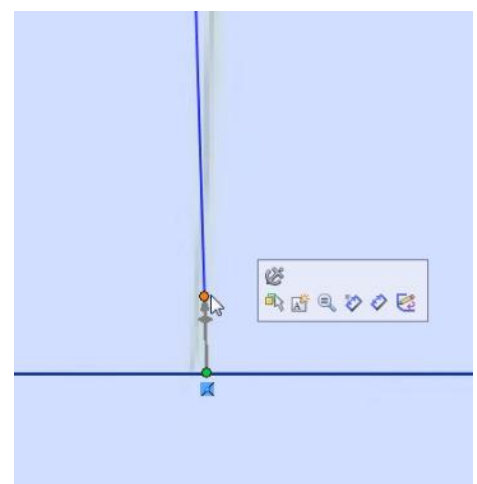
Move the mouse over and click the second point at the end of the (relatively) horizontal line. Press the Esc key to end this first spline. This is known as a two point spline, for obvious reasons. Initially it creates a straight line between the points, but soon we'll see that two point splines can still be edited into very useful curves.

**Now sketch the second spline (note, still in the same sketch)...** select the spline tool again and clicking on the last point of the first spline, and then rotating the model a little so you can select the end point of the guide curve on the Top Plane. This is just as we did when we snapped onto the end point of the Front Plane guide curve; make sure that the end point is highlighted orange and the coincident relation icon is visible before you click. If you can't get the point to highlight, try rotating the model a little bit more until the spline tool pointer can 'see' the point. After it is done press the Esc key to finish this second two point spline.

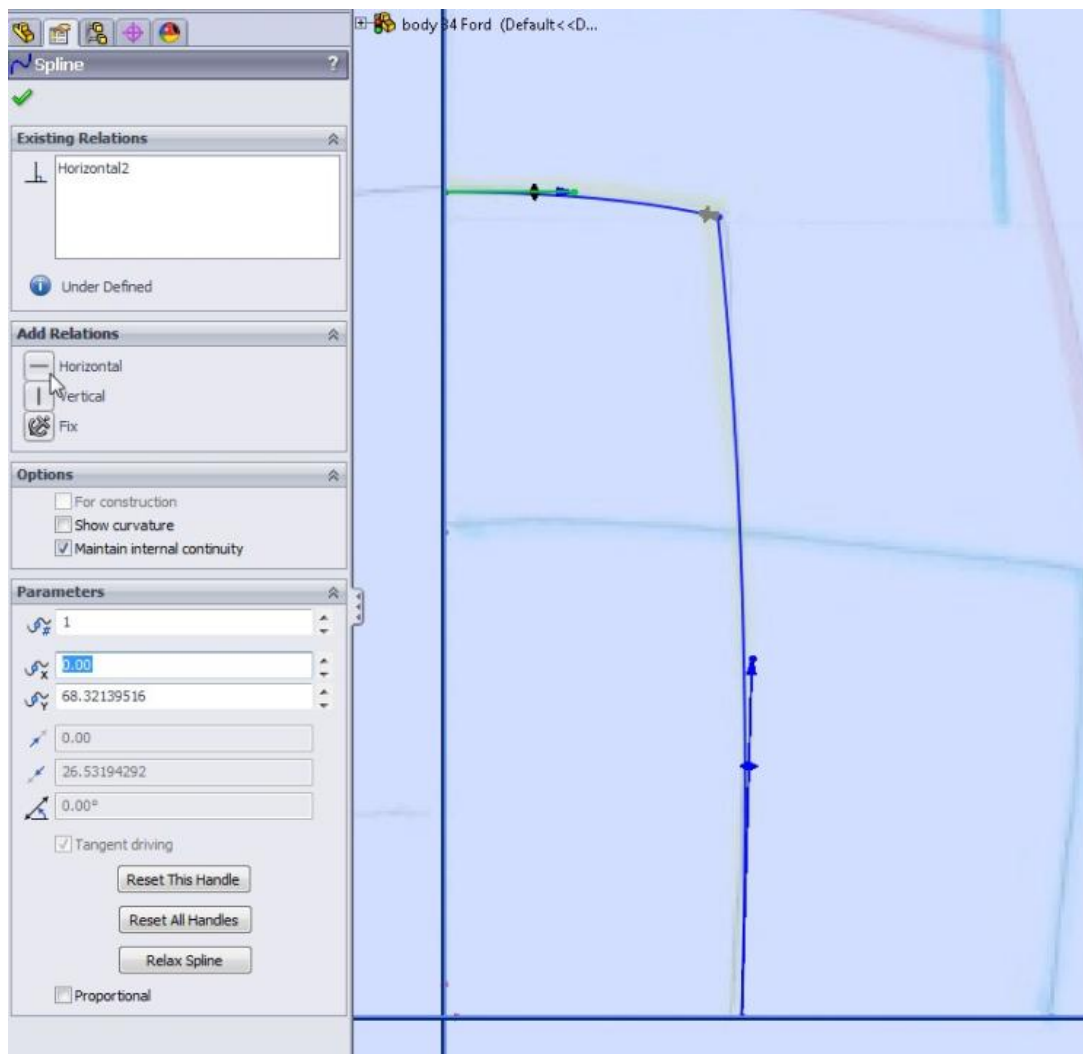


Your sketch should look similar to as above, but you may be disappointed that we went to the trouble of using the spline tool when we could have just used the line tool to create these straight lines. Now we'll see some more capability of splines that will reveal the sense in our choice.

Click on the point at the end of the more vertical spline and you will see a (very) small grey arrow appear originating from the end point. This arrow is a control handle for changing the direction of the spline at that point. Hover the pointer over the top end of the control handle and you will notice it highlights an orange dot. Click on this dot and hold the mouse button down. Move the mouse and observe what happens to the spline. By dragging the length and direction of the control handle you can change the shape of the whole spline. Move it to a location such that the spline closely traces the yellow sketch, and then let go of the mouse button to finish editing it.



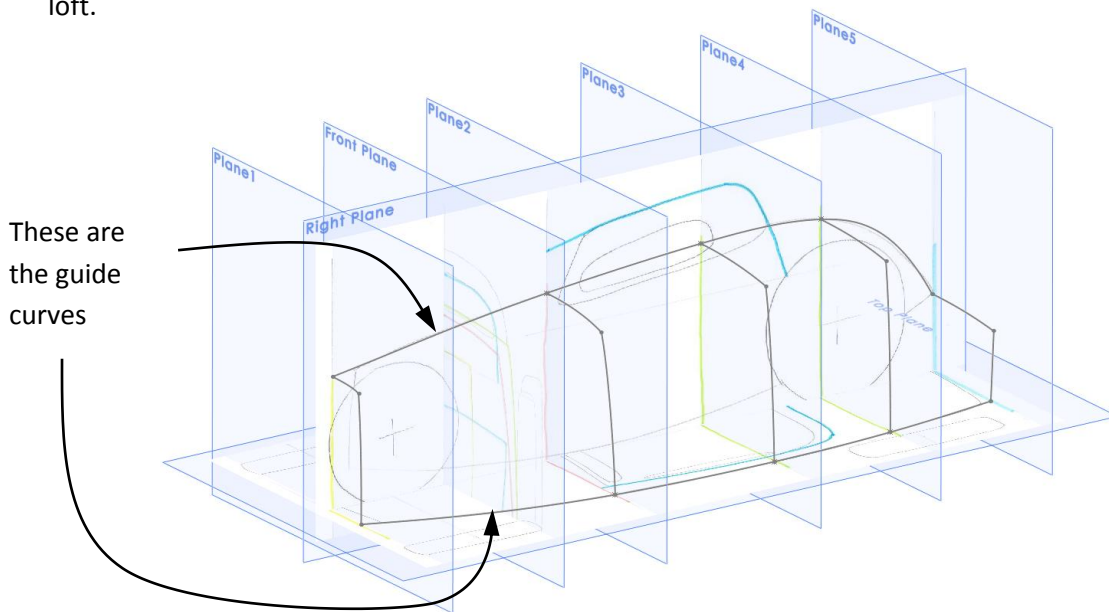
Do the same thing with the more horizontal spline by selecting the point at the left end of it. Because we are going to eventually mirror this part about its centre, we want to be sure that this end of the sketch is normal to the centre (Right) plane, otherwise there will be a dip or bump along the mirror line. We want it to be smooth. To make sure this is the case we need to make the left end of the sketch horizontal. We could do this by eye, but it wouldn't be perfect, so a better way is to add a sketch relation. Since we have already selected this point SolidWorks suspects we might like to do this, so has given us some options in the spline dialogue box in the Feature Manager area. Select the Horizontal relation and this will be applied, forcing the control handle arrow to move horizontal. You can still drag the length of the control handle, but no longer the direction of it.



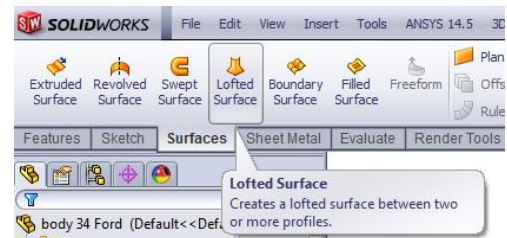
With the sketch now looking as we want it, with the end relations as they should be, and the curves close enough to tracing over the yellow hand sketch, exit the sketch to finish the first profile.

Each of the other four profiles are created in just the same way, so by the end of them you will be well practiced at creating and editing these very useful two point splines. Make sure all the end points have the correct relations by only clicking when the guide curve points are highlighted as

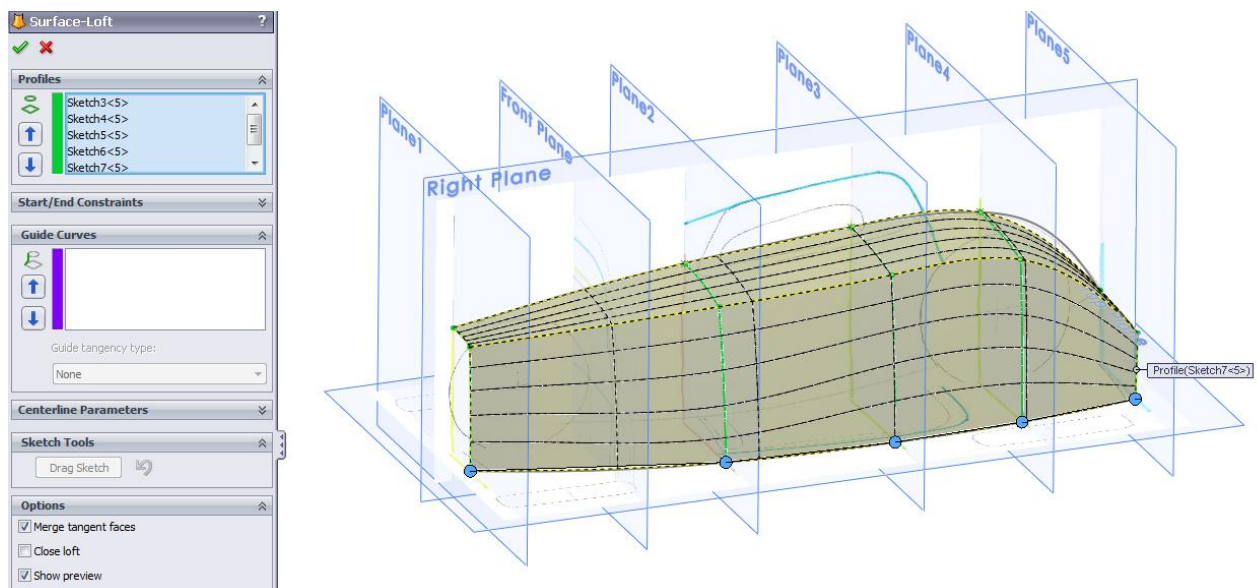
orange dots. Once completed the sketches should look as follows, and you are ready to create the loft.



Select the Lofted Surface tool from the Surfaces tab in the Command Manager. If the Surfaces tab isn't available, right click on any of the other tab names and select the Surfaces option to bring up this tab.



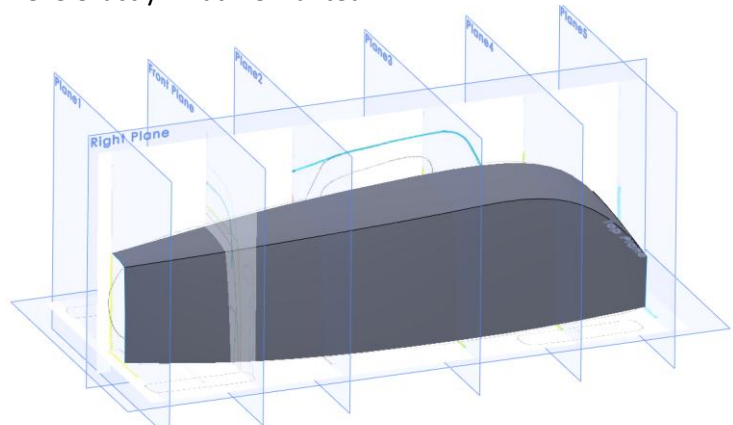
In the Graphics Area click on the Plane1 sketch, and then carry on clicking the other profile sketches one after another in order from the front of the car to the rear. Select the profiles by clicking them in (very) approximately the same location on each one, otherwise the tool may try to twist the loft. The preview should look as below.



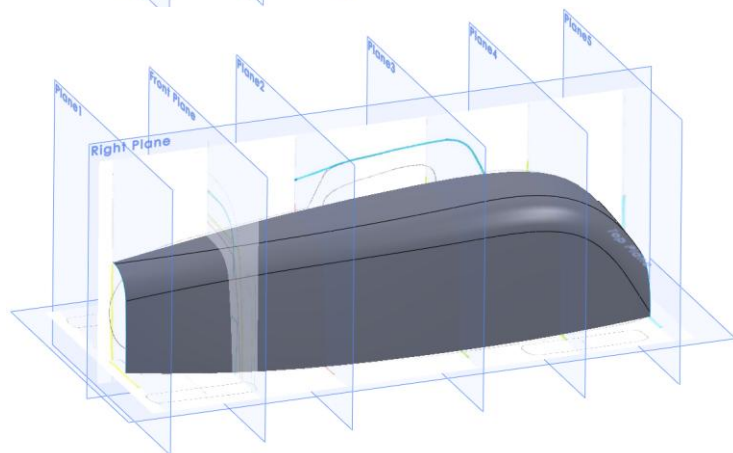
Note how the surface at the rear of the car droops below the guide curve on the Right Plane, which is where we would like it to be. This is where the guide curves come into use. Click in the Guide

Curves box in the Surface-Loft dialogue and then in the Graphics Area select both of the guide curves you created earlier. The shape of the preview changes so that it conforms with these curves, and thus we have been able to control it go achieve exactly what we wanted.

With the preview looking as you want, click on the green tick in the Surface-Loft dialogue to build the feature. It may not look like a car body yet, but we're on our way!

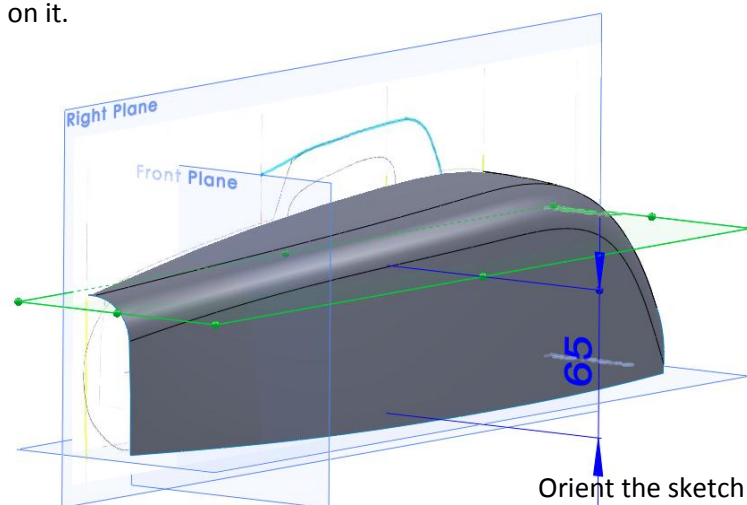


To blend between the top and side surfaces of the body, use the fillet tool to create a 20mm fillet down along the edge between them. You will notice that fillets can be created on surface features just like they can be on solids, there is absolutely no difference.



Next we'll see an example of the versatility of lofting by creating the car roof surface using the tool in a slightly different way.

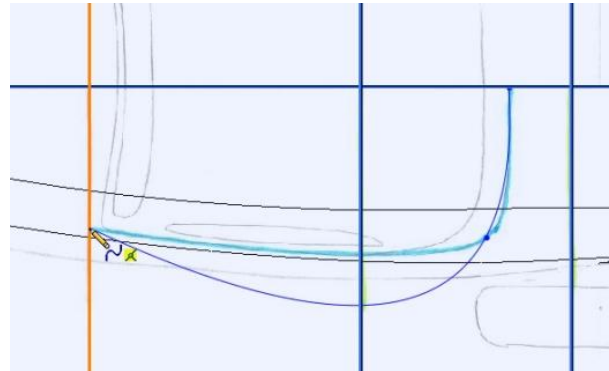
Begin by creating a plane offset 65mm up from the Top Plane. We are going to create a loft between two profiles, with a single guide curve between them. The first loft profile is on this new plane, so create a new sketch on it.



Orient the sketch so it is parallel to the screen. We want to do some tracing over a line in the hand sketch, but unfortunately we can't

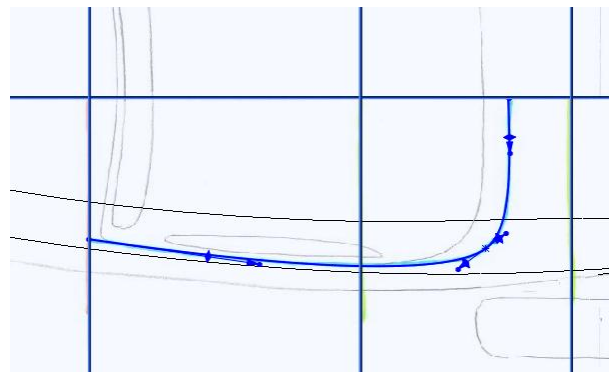
see it because our first loft surface is in the way. To fix this problem select wireframe view from the view dropdown in the heads up menu. This removes all shading and provides a sort of x-ray vision. The shaded view can be turned back on when you have finished with the x-ray vision.

Now that you can see the hand sketch on the Top Plane, select the spline tool from the Sketch tab in the Command Manager and trace over it. Begin with the point at the top, allowing it to automatically add a coincident relation to the Right Plane, then click one middle point about where shown, and finally allow the end point to snap a coincident relation to Plane2. Don't worry that the shape of the spline is a little crazy when your first create it.



**This is only one spline! Click for start, click for middle, click for end.**

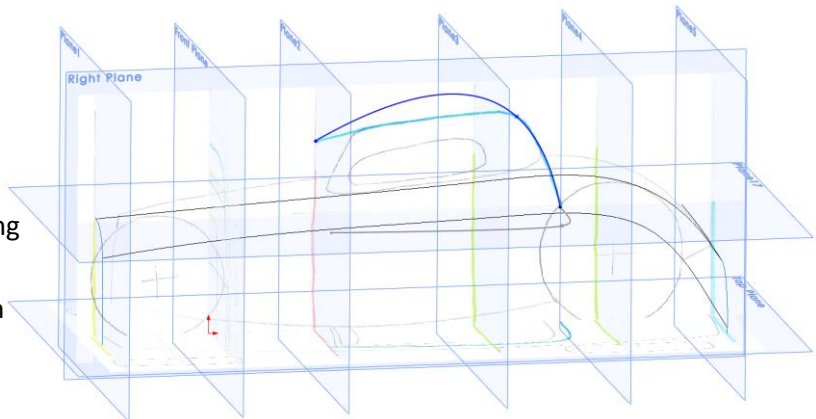
Just like we have done previously, select the end points of the spline and drag the control handles until you get the shape you want. Before we only did this with the end points of our two point splines, but in this case you can see that you can do it with spline points part way along a spline also. The technique is exactly the same.



Exit the sketch once you are happy with the shape.

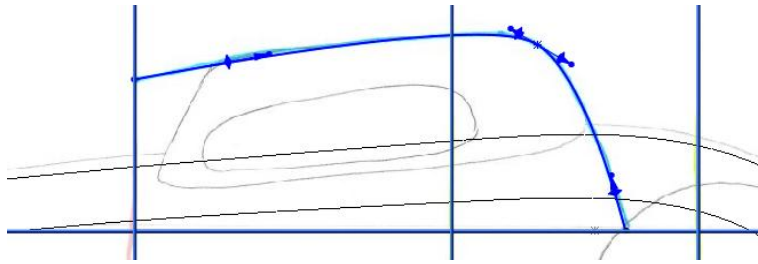
The second of the profiles for the roof loft will be created on the Right Plane, so select this in the Graphics area and create a new sketch on it. Use the spline tool to create another three point spline as you have just done. This time make sure you allow SolidWorks to create a coincident relation to the end of the first profile sketch.

As you have done before, you will need to tilt the model onto an angle to enable the spline pointer to 'see' the point you want to snap to. Remember if the point you are aiming for has not been highlighted with an orange dot, then the correct relation will not be made even though the pointer may be in right position.





Select the spline, or the points on the spline, to make the control handles appear. Drag each of these into so that the spline traces approximately over the blue line in the hand sketch which represents the top of the car roof. Once happy with the shape, exit the sketch.

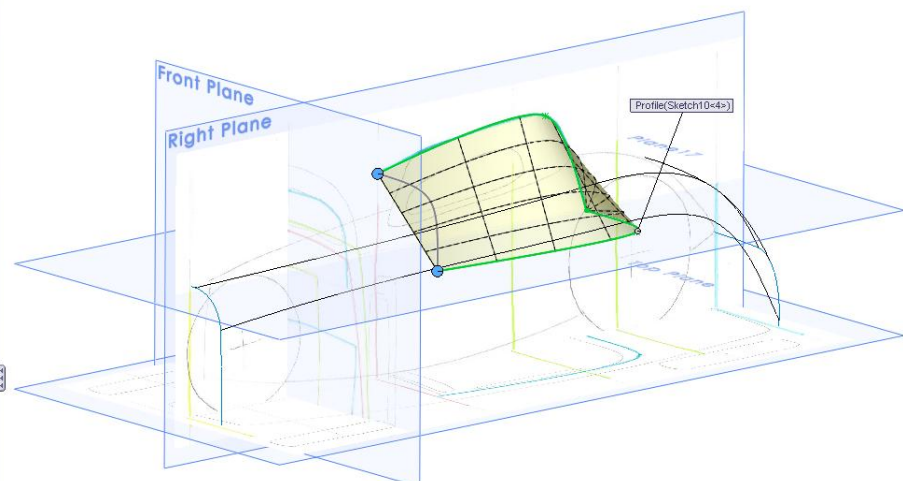
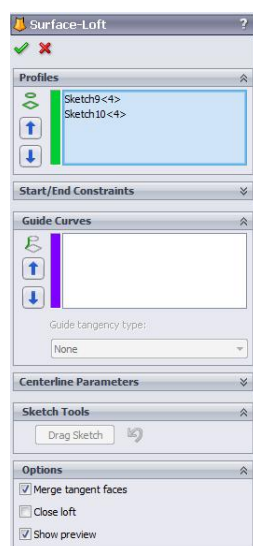


Traditionally lofts are thought of as being performed between profiles that are separated, but this is not necessary. The two sketches we have just created are the profiles that we will loft between to make our roof panel, even though they actually meet at one end. We could complete the loft now and it would work OK, but we will add one Guide Curve to assist the shape of the roof.

Select Plane2 in the Graphics Area and create a new sketch on it. Create a two point spline by just one click at the start and one click at the end, then press Esc to finish it. However make sure that you allow SolidWorks to create coincident relations with the end points of the two profile sketches (remember the orange highlight dot must be visible, and you will have to tilt the model around a little to let the pointer 'see' them). Drag the control handle pointers so that the spline traces the hand sketched front roof shape. To make sure the roof is smooth after it is mirrored, apply a horizontal sketch relation to the spline point on the centre line also. When happy with the shape, exit the sketch.

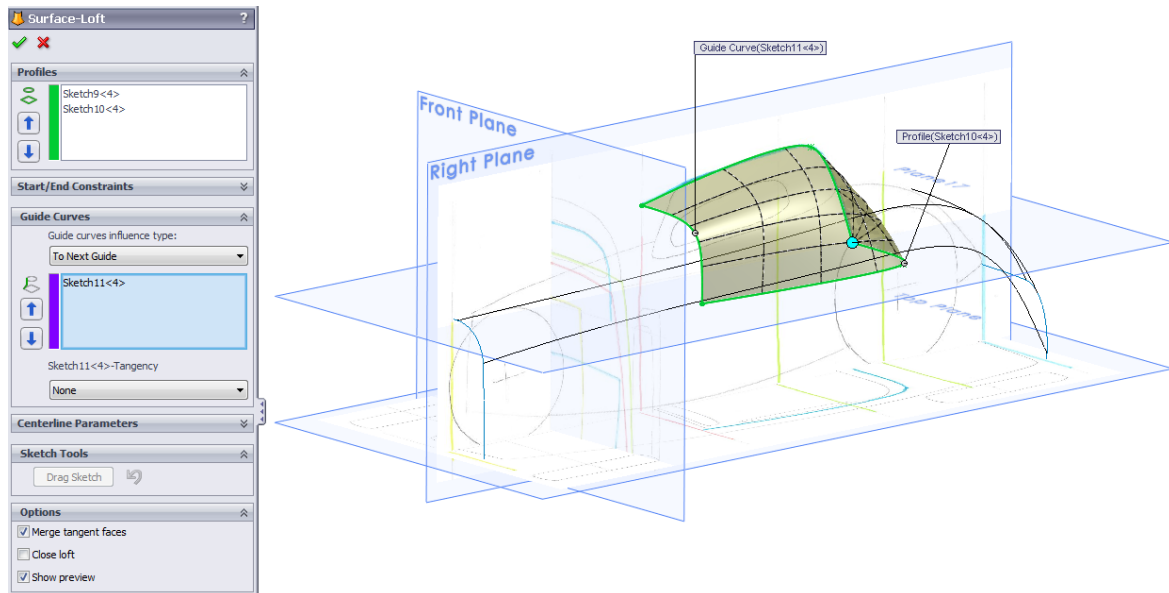


Now to use these sketches to create the roof surface, select the Lofted Surface tool from the Surfaces tab of the Command Manager. This time select the two profile sketches (one on the Right Plane and one on the Top plane) and you should see the yellow preview appear.



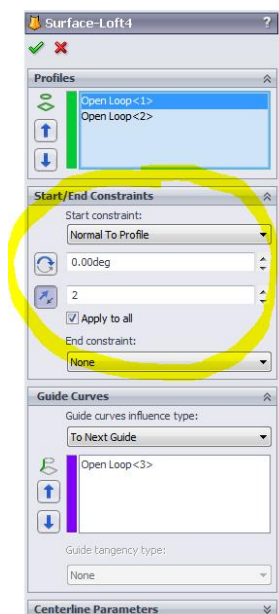
You can see the from the preview surface that the shape is very unlike a car roof. Fortunately we have anticipated this and have already created a guide curve to give the surface a 'lift'. If you hadn't done this already it would be no problem, it is very common to create more guide curves after performing a loft and then adding them into a loft to improve the shape.

Click in the Guide Curves box in the Surface-Loft dialogue and then select our Guide Curve in the Graphics Area. Look at the preview and you an see the shape has improved dramatically.

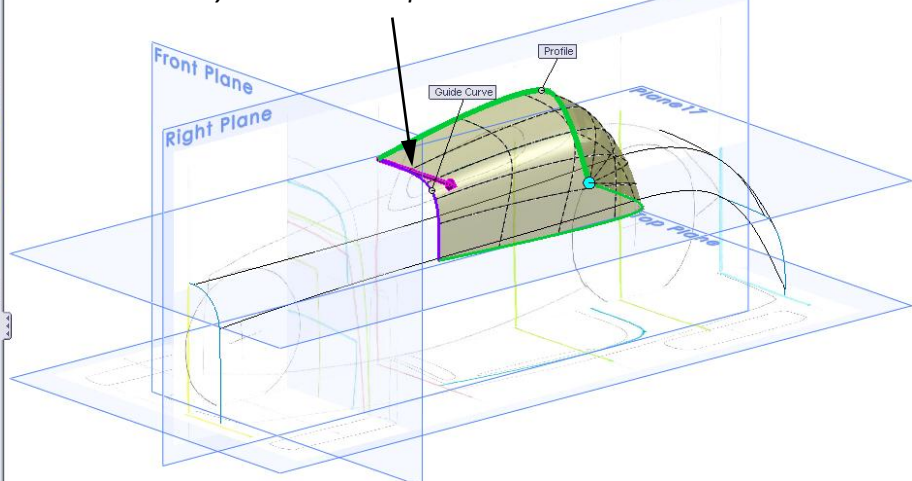


If we were to accept it as it is now we would still be unhappy with the result however. If you look along the edge that is on the Right Plane you can see that it is not all normal to the plane, so when it is mirrored it will have a 'peak' on it rather than being smooth. At the very front it is normal, because of the sketch relation we put in place, but this did not ensure that condition was carries all along the edge, only at that particular point. Also the back end of the roof is not as 'puffed up' as we would like. The good news is that SolidWorks provides easy and powerful capability to fix these issues.

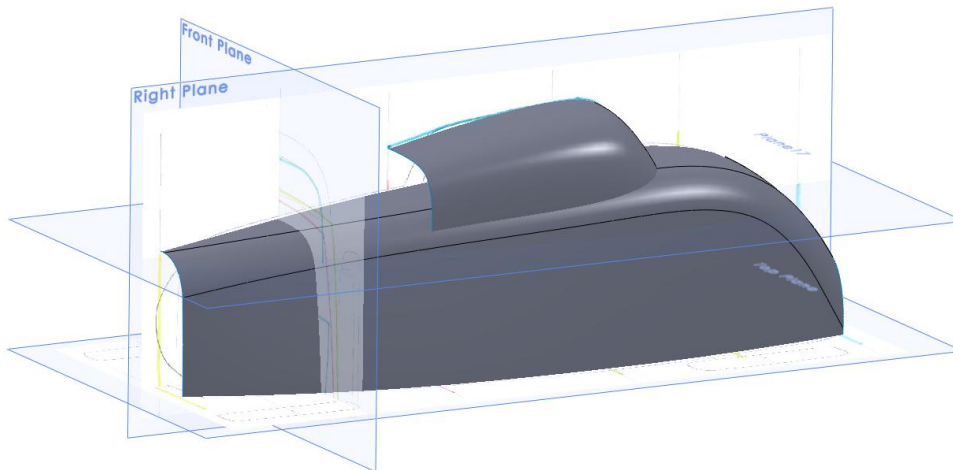
Click on the Start/End Constraints bar just under the Profiles box in the Surface-Loft dialogue to open the dropdown that contains these additional controls.



*See paragraph on next page... if this vector arrow is vertical rather than horizontal you need to swap the start and end conditions around.*

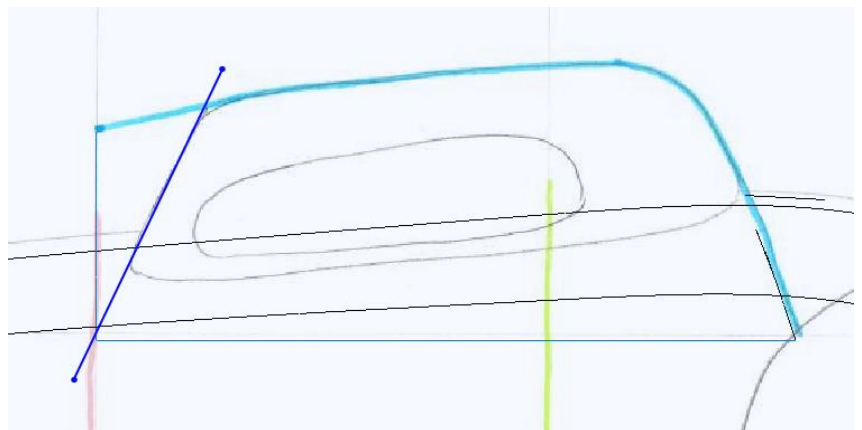


**The start refers to the conditions for the first loft profile you selected and the end refers to the condition for the last loft profile you selected.** Assuming you chose the sketch on the Right Plane as the start sketch as I did, then select the condition Normal To Profile instead of the default None. Also enter 2 in the box next to the arrows. The number determines the 'strength' of the normal condition; experiment with different values to see the effect. If you selected the profile sketches the other way around to that which I've done, just enter the Normal to Profile and influence value of 2 for the End Condition rather than the Start condition. Either way will give the same result. The preview should now look as shown, with the pink arrow indicating the Normal to Profile control on the surface. Notice how the roof now looks nicely puffed up now. Click the green tick to create the new surface and then use the heads up view control to turn the surfaces back into shaded mode.

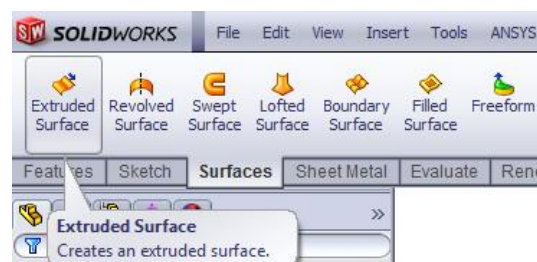


The basic shape of the body and roof are looking good now, so its time to add the nose (radiator grill) and windscreen of the car. Both of these will be created using the Extruded Surface tool. This works similarly to the normal extrude too, except rather than extruding a closed profile to create a solid body, it extrudes an open profile to create a surface body.

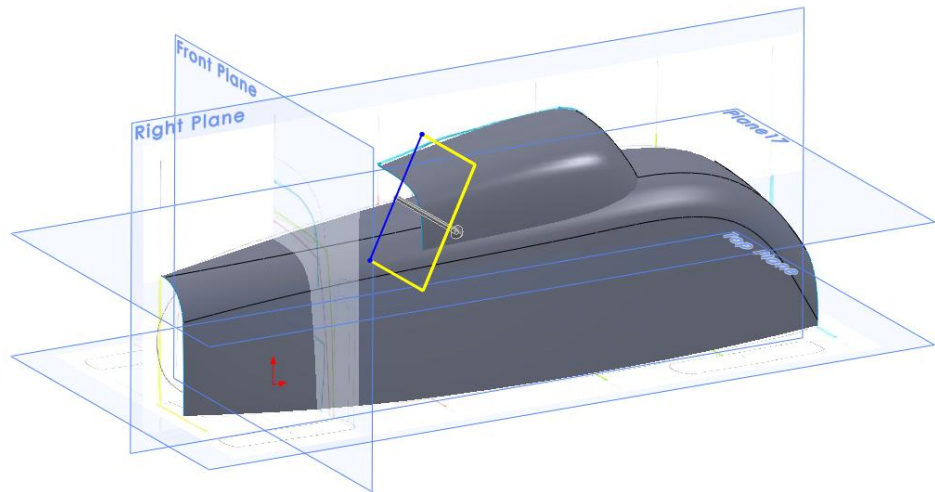
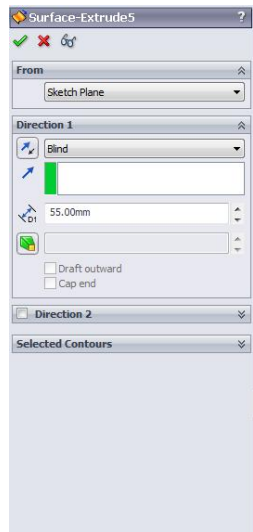
Begin by selecting the Right Plane in the Graphics Area and creating a new sketch. Use the Line tool and create a line that traces approximately over the side view of the windshield. Remember you can change the model view to wireframe enable your x-ray vision. Carry the end of the lines out a little further as shown, but the exact location of the end points doesn't matter. Exit the sketch.



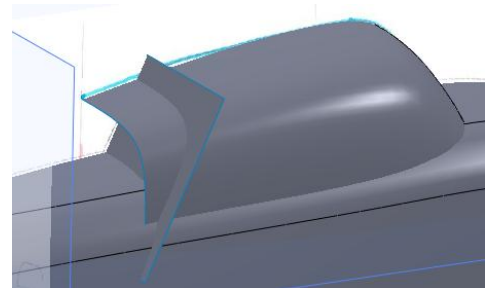
Select the Extruded Surface tool from the Surfaces tab of the Command Manager and then select your line sketch from the Graphics Area.



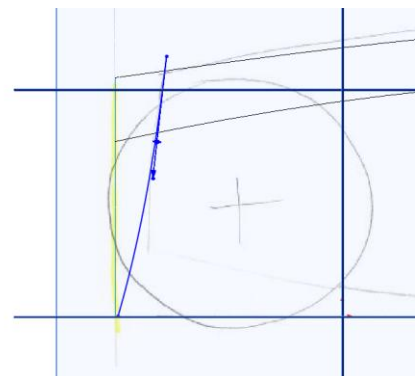
Enter an extrude length of 55 and check that the preview shows the extrude is going in the correct direction. If it looks OK, click the green arrow to create the feature.



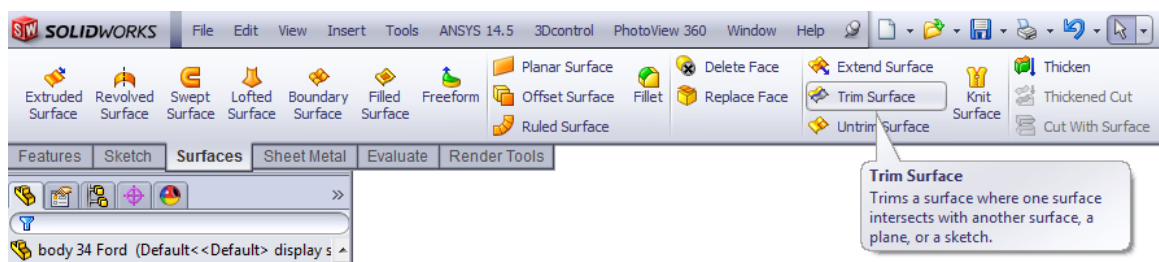
Don't worry that the new surface you have created stick out through the previous surfaces, this is as it should be for now, and we will resolve that very soon. It is a very common workflow to create surfaces that overlap each other and then trim them back to each other. The method is called *over modelling*.



Repeat this process over again to create another surface that will be the radiator grill at the front of the car. This time use a two point spline so that you can put a little bit of curvature into the shape. Begin one end of the spline on the Top Plane and then the other end some point above the bonnet of the car. Don't worry about the exact shape, just make it look something like shown. Use the Extruded Surface tool to extrude the sketch a distance of 40.

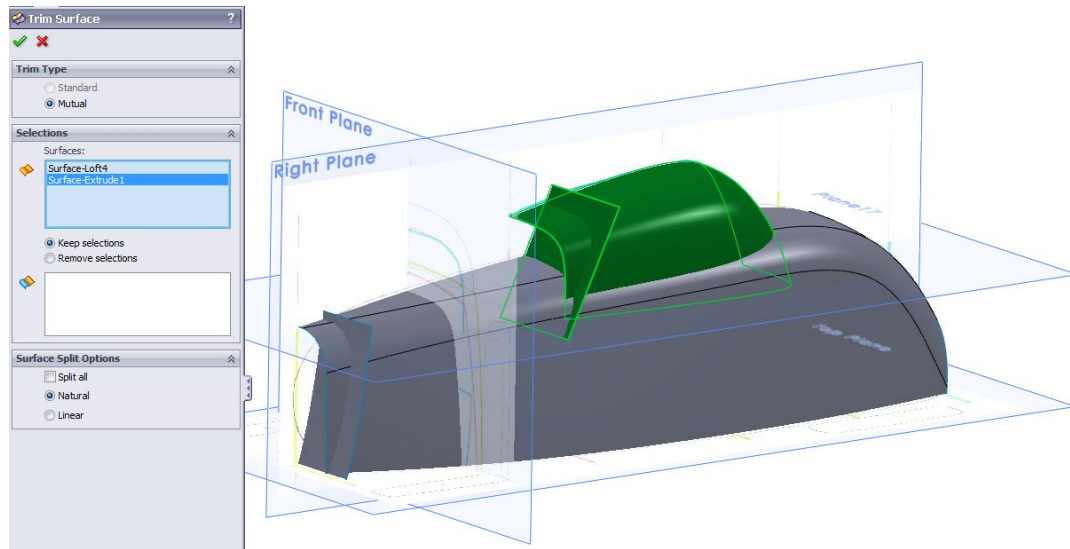


Now to tidy up our various surfaces, and join them together into a single surface, we will use the Trim Surface tool. Select this tool from the Surfaces tab of the Command Manager.

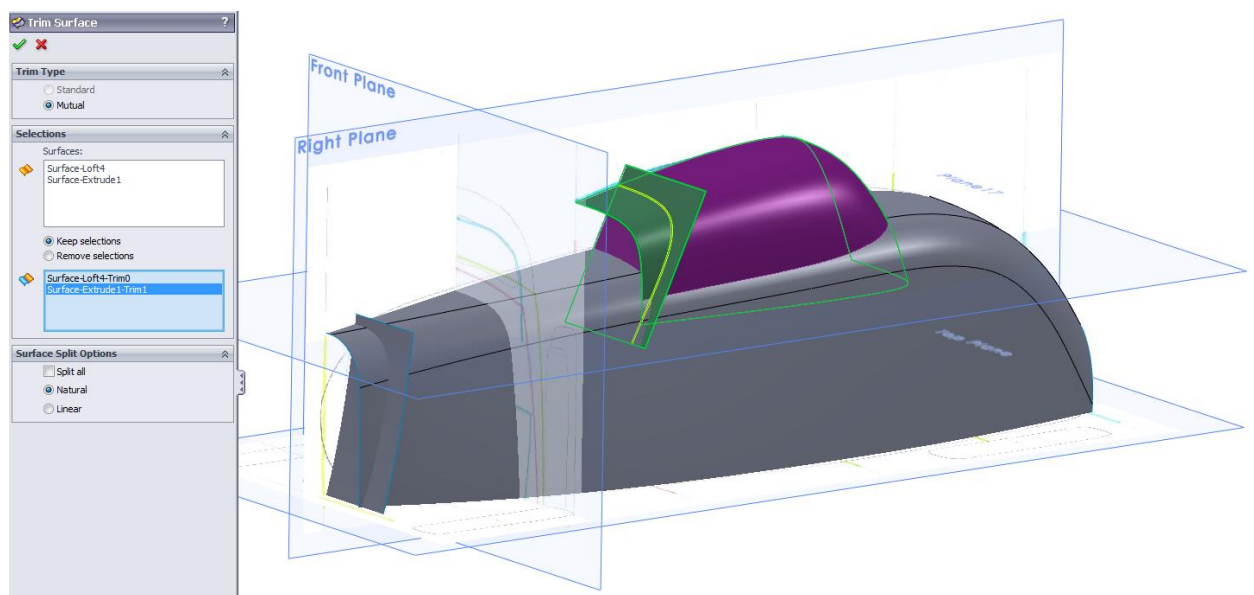




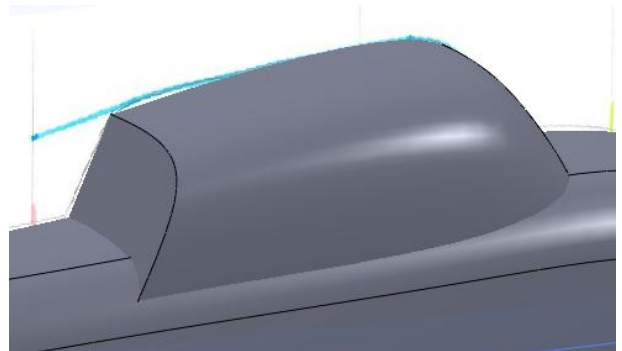
The Trim Surface tool is very simple, but very powerful. Select the Mutual radio button at the top of the Trim Surface dialogue box and then click on each of the roof surface and the windscreen surface in the Graphics Area. Your screen should now look similar to as shown.



Click in the white box below the Keep Selections/Remove Selections radio buttons, and then click on the roof and windscreen surfaces again, but this time make sure you click in a location on the surfaces which will be part of the piece you want to keep. When you select these surfaces they will change to a purple color to signify that this is the part that will be kept.

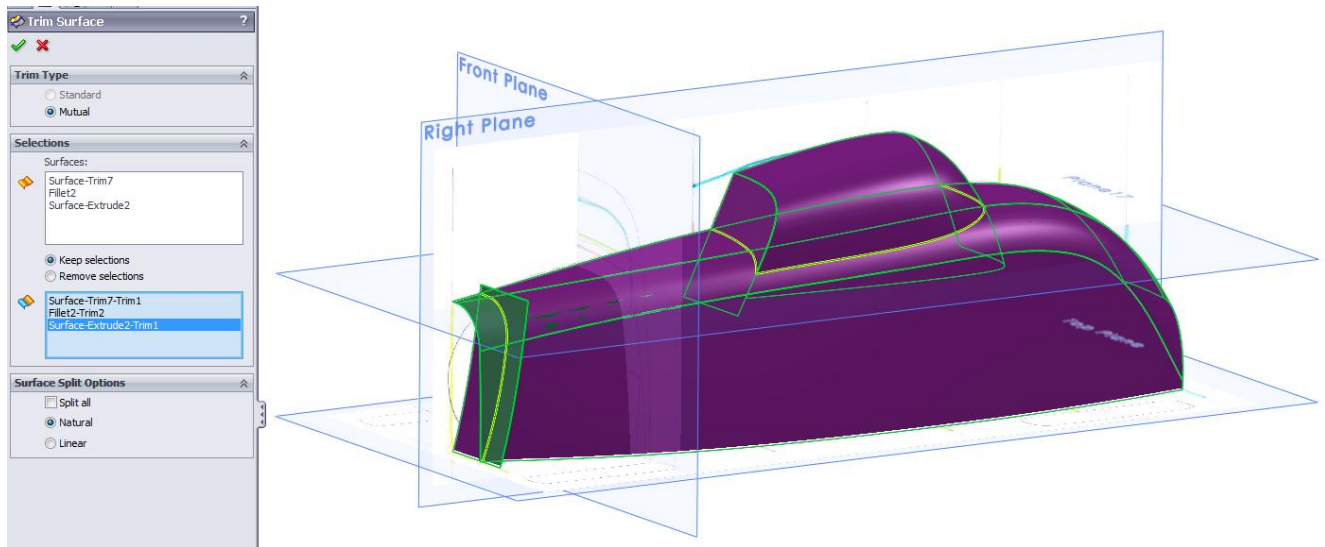


Click on the green tick and you can see that the roof and windscreen trim each other (that's why it was called a Mutual trim) and have joined together into one surface (note the black line where they are joined as opposed to the previous blue lines where the edges are not joined to any other edges).

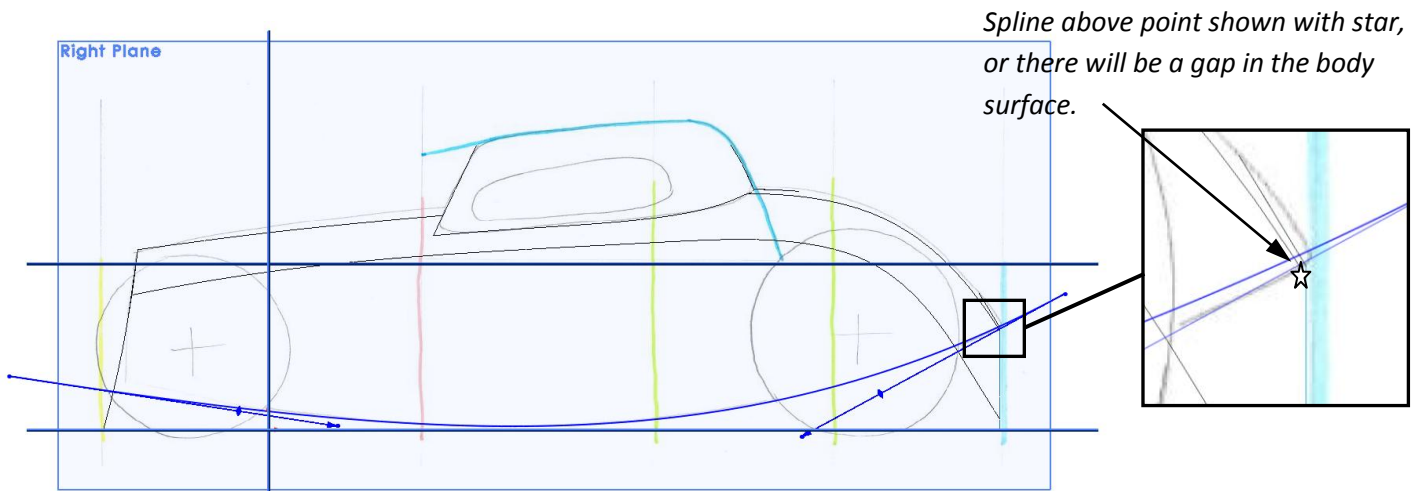




Perform another Trim Surface operation, but this time select the roof/windscreen surface, the main body surface, and the radiator grill surface. Make sure it is set to Mutual, and then select the bit of each you want to keep. The result should now look somewhat more like a car.

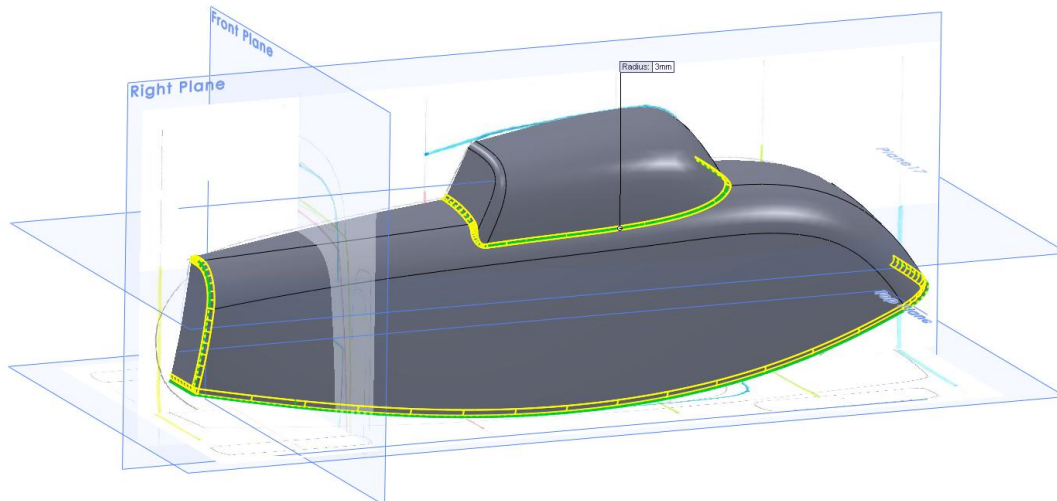


Now as a revision of the last few steps (sketch, extruded surface, trim), let's take care of the bottom shape of the car. In a unique hotrod fashion it is a big curve in side profile, so create new sketch on the Right Plane and use a two point spline to create a long curve as shown. Make sure it doesn't go down below the Top Plane, and make sure that it is high enough at the back that once it is extruded it will intersect the other body surfaces (otherwise the trim won't work).



Use the Extruded Surface tool to extrude the sketch 80mm. It should now overlap with the main body surface, so use the Trim Surface tool set to Mutual to trim them both back to each other and keep the upper part of the main body surface, and piece of the new extruded sketch that forms the bottom of the car.

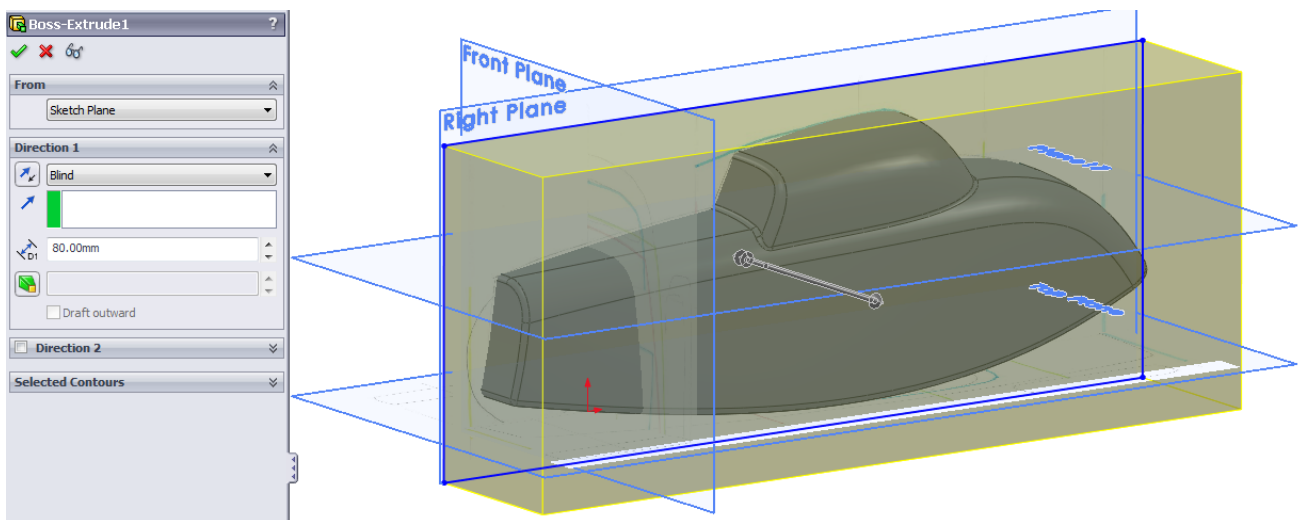
Use the Fillet tool located in the Features tab of the Command Manager to apply a 5mm fillet on the edge between the roof and the windshield. Next use the same tool to apply a 3mm fillet around all of the remaining edges on the model.



We've now completed as much of this model using surfaces as we need to. Remember that some parts of modelling are easier using surfaces and some are easier using solids. We can quite easily convert a model either way, and in fact in more complex models the model may transfer back and forth a number of times.

There are a number of ways to convert this surface body into a solid body, so we'll choose one that demonstrates a useful property of surfaces – that they can be used to cut solids.

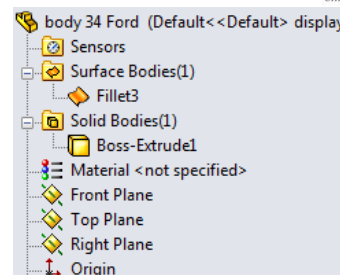
Use the Extrude tool in the Features tab of the Command Manager (note this is a standard Extrude, not the Extruded Surface tool in the Surfaces tab), to extrude large rectangular prism that engulfs the whole car body.



Click the green tick to generate this feature and it appears that the surface which we spent so much trouble creating is gone, but actually it is not, it is still sitting exactly where it was, but the solid is sharing the same space.

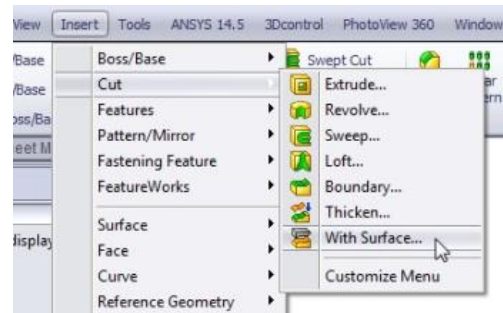
Take a look toward the top of the Feature manager area and you can see that there are two folders, called Surface Bodies and Solid Bodies, with + next to them. Click on each of the + icons to expand

these folders and you will see in each there should be one item. The Surface Bodies folder contains the car body surface we have constructed (click on it to highlight it in the Graphics Area), and the Solids Folder contains the solid rectangular prism we just created (Click on it to highlight it in the Graphics Area).

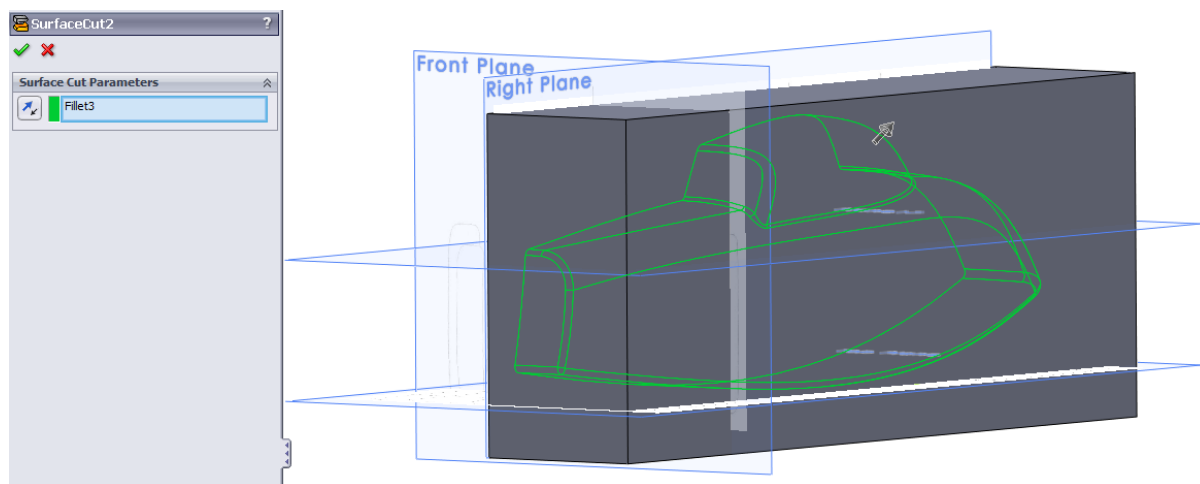


A great capability of SolidWorks is that you can have as many surface and solid bodies in a part as you like, and they can be interacted with each other to achieve useful outcomes. In this case we will use the surface to cut the solid block.

Select the With Surface tool from the Cut dropdown in the Insert main menu. This tool works when a surface passes full through a solid and effectively divides the solid into two parts, one on either side of the surface. You choose which side you want to keep and the other half disappears.



The SurfaceCut dialogue is only asking for one input, and that is the surface to be used for cutting. Hover you point over the solid block in the Graphics Area and you will see it highlight the surface body that is lying within it. Because the feature is looking for a surface body it ignores the solid. Click when you see the surface highlighted and it will be selected. The arrow in the preview shows the direction of the material that will be removed, so we want this pointing outwards. If it is pointing in the wrong direction then click the arrow in the Graphics Area, or the button in the dialogue box with the arrows on it, and the direction will be reversed.



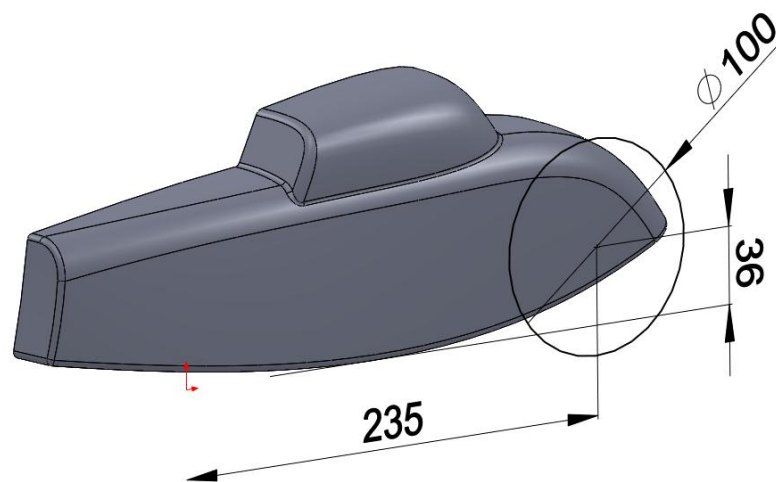
Click the green tick to perform the cut, and you should see the car shape again, with the rest of the solid block cut away. Take a look in the Surface Bodies and Solid Bodies folders and you will see that there is still one item in each. The surface is still there unaltered, and the solid is still there, but it is now the shape of half a hotrod. When you look at the car in the Graphics Area you are actually looking at both the surface and solid body as they are right on top of each other. This may cause some confusion later, and because we have finished with the surface in this model, right click on the

surface body in the Surface Body folder and select the Hide icon so that this body will be invisible. You will not see any change in the Graphics Area, but it will stop the surface getting in our way in the next few operations.

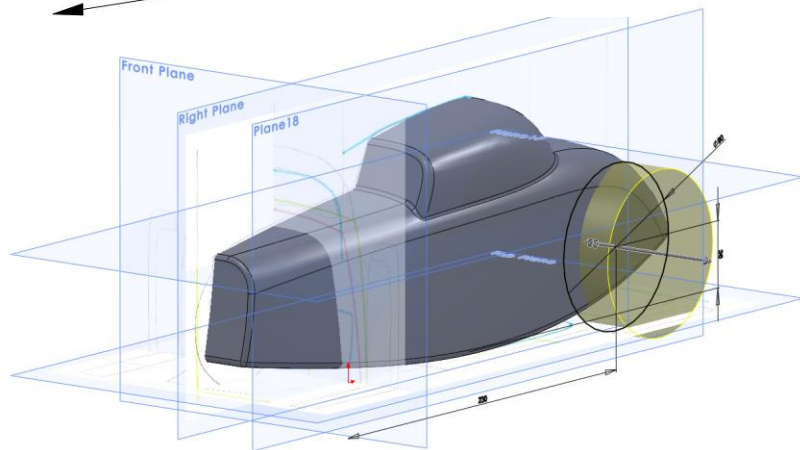


The rear of our hotrod has some large cutouts in the side of the body to give clearance around the rear wheel. We'll create these with a simple Extrude Cut, and then put some fillets around the edges to smooth it off.

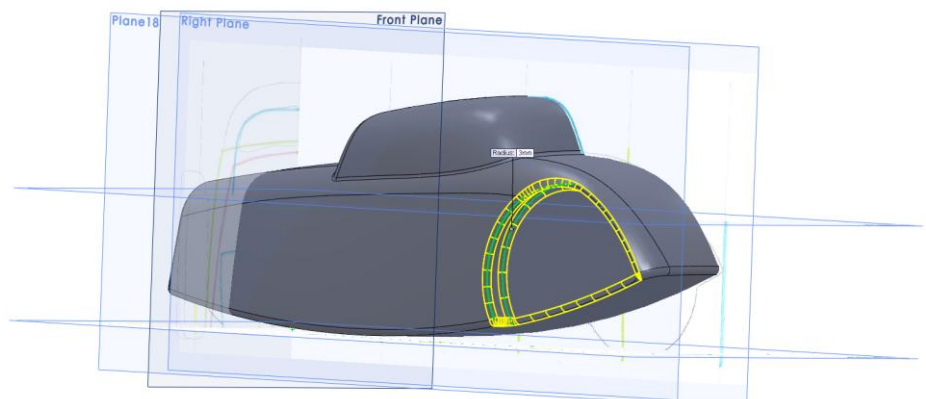
Create a plane offset from the Right Plane by 50mm and start a new sketch on it. Use the Circle tool in the Sketch tab of the Command Manager to draw and dimension a circle as shown.



Exit the sketch and use the Extruded Cut tool in the Features tab of the Command Manager to extrude a cut outwards.

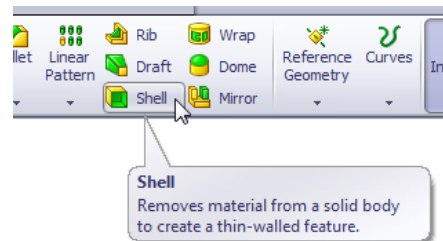


Use the Fillet tool and apply a 3mm radius to all of the edges around the cutout you have just made.



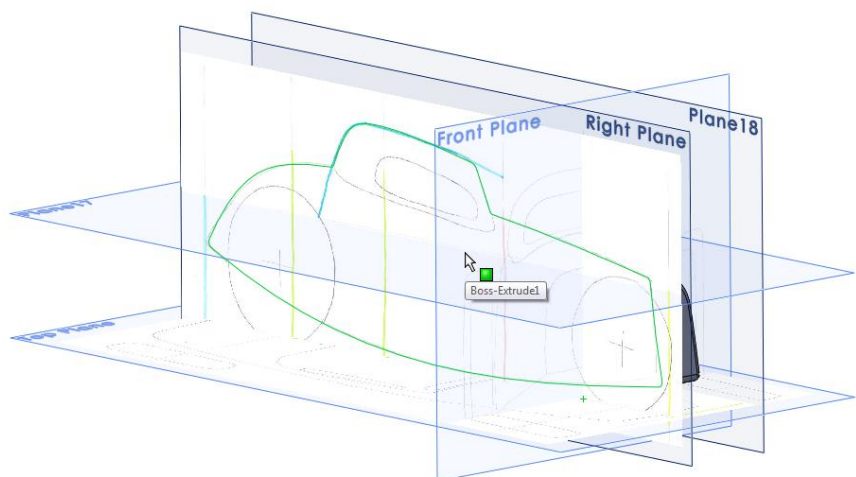
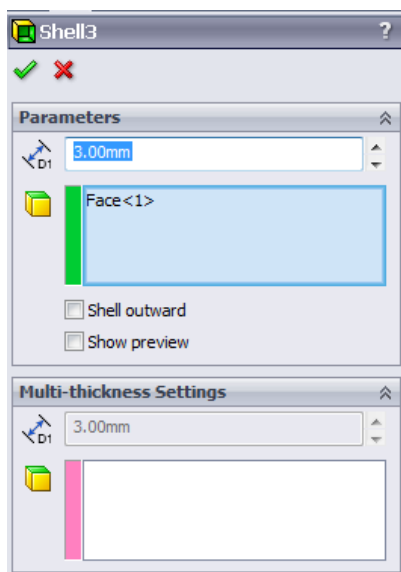
Because we want to put some things inside the car body, ie the engine and fuel tank etc, we need it to be 'hollow'. In CAD modelling this is called a shell operation because it removes the material from the inside of a solid and leaves only a shell of specified thickness. It is a very useful and powerful functionality.

Select the Shell tool from the Feature tab of the Command Manager.



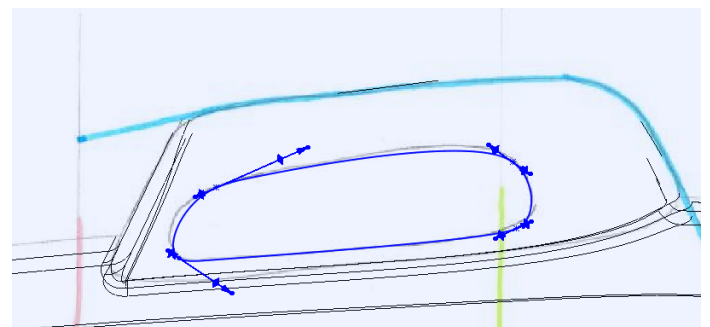
The Shell dialogue primarily wants to know what thickness the shell should be, and also if you want to remove any faces of the solid in the shelling process. In our case we do want to remove the face on the Right Plane of the part otherwise there would be a 'wall' down the middle of the car.

Enter a value of 3mm and then select the face on the side of the solid body that is on the Right Plane. You can't see it because the hand sketch picture is in the way, but the face will select anyway; you will see the edges highlight when you hover over the face. Click the green tick at the top of the dialogue box and the shell feature will build.



The car needs cutouts for the side window and windscreen, so we'll use Extruded Cuts to create these.

Create a sketch on the right plane using a closed spline as shown. A closed spline is one where the last point you click is on top of the spline start point. Unlike open splines you don't need to press Esc to finish the spline, SolidWorks knows you are done once you click back on the start point. Hold down the Ctrl key as you select the spline points to avoid any unwanted sketch relations from generating.



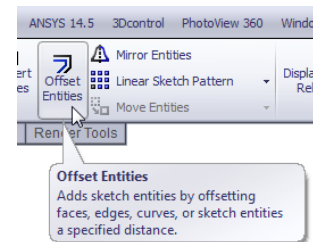


Note how there are four points on the spline and these are roughly in the corner locations. This was chosen because it is generally necessary, for sufficient control of the spline shape, to have a spline point wherever there is a significant change in shape or direction. After first creating the spline you'll need to do a bit of playing around with the control handles to get the shape right.

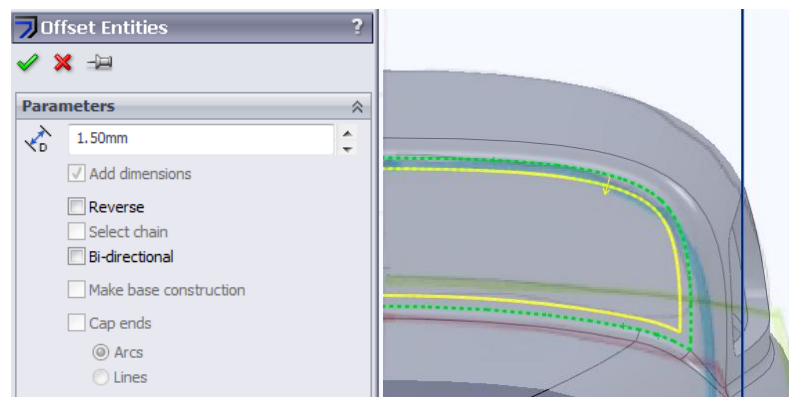
Use the Extruded Cut tool to extrude this sketch outwards and cut a hole in the door of the car to make the side window.

The hole for the windscreen is done in a similar way, but start the sketch on the Front Plane.

Because we want the windscreen shape to match the shape of the existing flat surface we'll use the Offset Entities tool, located in the sketch tab of the Command Manager, to create a copy of the existing edges and offset them into our sketch.



Select the tool and then click on the edges (highlighted in green) that you want to offset. Enter a value of 1.5mm. If the yellow preview line goes in the wrong direction then click on the check box next to Reverse, otherwise click the green tick to generate the sketch lines offset from the edges.



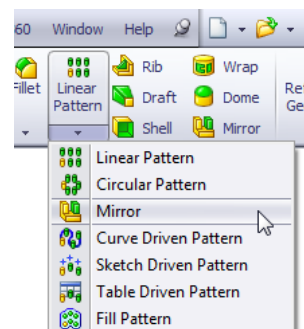
Finish the sketch by drawing a vertical line to join the ends of the offset edges. This will make the sketch a closed profile, which is necessary for performing a cut operation.

Use the Extruded Cut tool with this sketch to cut the hole for the windscreen.

With the body now looking quite like a car, and having done as much modelling on it for now as we intend to do, the final step is to apply a Mirror feature to make the other side of the car.

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

Select the Right Plane as the Mirror Face/Plane. There are three options as to what can be mirrored, Features, Faces, and Bodies. Each of these has a different selection box which can be expanded or collapsed by clicking on the arrow next to the text description in the dialogue box.



It is a common mistake to select Features for mirroring, when Bodies are usually more appropriate. Mirroring the body collects all of the geometry of the features that have been used to create it and mirrors them as one piece. This has the advantage that if you go back and add any features you do not have to go back and add that feature into the mirrored feature list, as it will automatically be picked up as part of the body geometry.

