Where to Start?

One thing to notice about this part is that the edges are really what is of primary importance here. The 3D shape of the faces is actually secondary.

The point from which your modeling starts is partially dependent upon what type of information you are starting with. In this case, we start with a set of digital pictures, and we are essentially reverse engineering the model from the pictures. In some parts, we will start from scanned hand sketches.

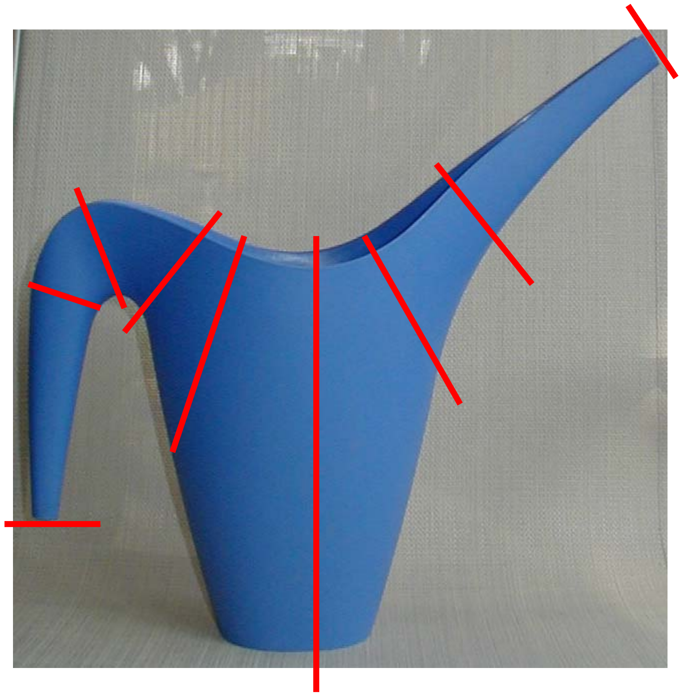
Planning the Work

The first thing to look for is to see if you can do the part, or at least the major surface of the part, in a single feature. Remember that the smoothest connections between faces are ones that don't exist, so reducing the number of faces (and therefore reducing the number of features) in the model, where practical, is a good thing. That "where practical" is a bit of a cover-all statement, but personal judgment is an important part of the process, a single formula cannot be written which automatically makes correct decisions for you in every circumstance. Using this concept, you will often *overbuild* model faces, that is, make them bigger than they need to be and then trim them back. This is a frequently used technique in surface modeling.

> **Note:** You should be aware that not everything you try in SolidWorks is going to work on the first shot. In fact, there will be times when you give up on one method and try another. Keeping a toolbox full of workarounds or simply different ways of accomplishing something is a necessity with complex modeling. This is not a complaint about the software. it's just the reality of the situation.

In trying to make the main shape in a single surface feature, the next step is to plan out *how* exactly to accomplish that. Whether you do this in your head, on paper, in some other software or in SolidWorks initially doesn't really matter. I will lay out the task visually to show you what I mean.

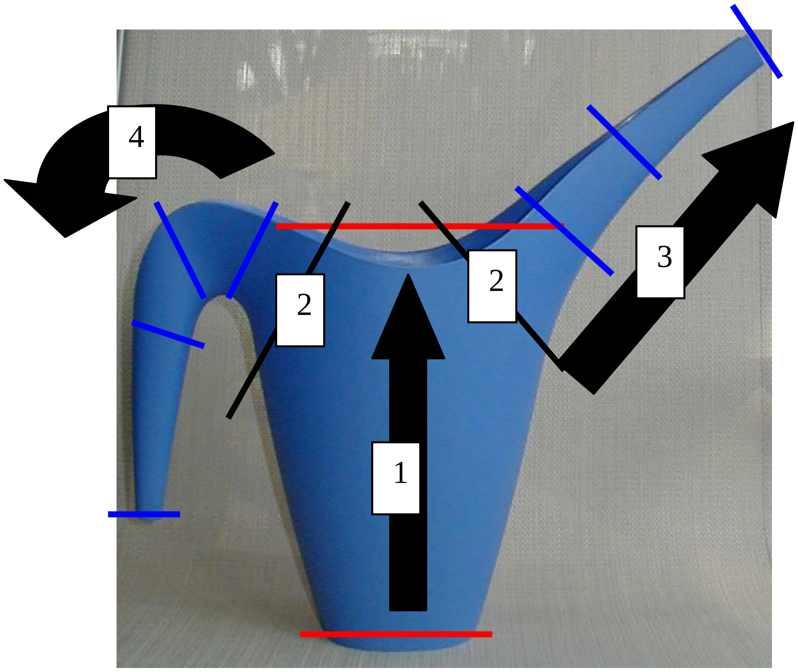
Regardless of what actual feature you select to do the work, the resulting surface is going to have a U-V mesh which works best as a four-sided mesh. It is always best to try to work *with* the software and the underlying geometry creation methods rather than *against* them. So, let's see if we can fit a four-sided mesh to this part.



The red lines represent possible loft profile locations, and thus also represent the orientation of the U-V mesh that we propose to make. Notice that this method neatly creates the four-sided mesh we are looking for, with the second direction being roughly perpendicular to the red lines. The bottom of the can will be trimmed flat.

One of the problems with this approach is that getting the bottom of the can to the shape in the pictures is going to be difficult. This method may not be the best if the shape of the bottom is critical. The shape of the bottom is shown in the image to the right.

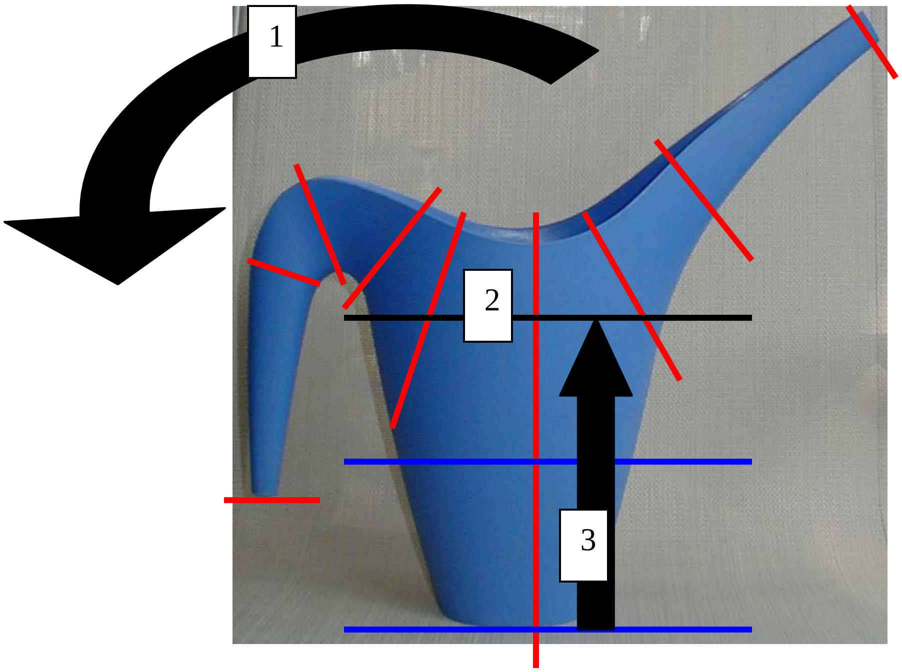
With the possibility that the first plan will not work, let’s devise a second plan that accounts for the shape of the bottom better. This plan will use three major surface features instead of just one.



1. Loft the main body between profiles at the red lines
2. Trim connections for the handle and spout at the black lines
3. Loft spout
4. Loft handle

The main risk with this technique is that the connections (edges) between the spout, handle and main body will not be as smooth as possible. Let's try one more idea and then evaluate to see which one to build.

In this third plan, we mix the best advantages of the first two plans by building the model as proposed in the first plan, then cutting off the main body of the can and rebuilding it using the second method. That might look like this:

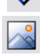


1. Loft spout, top and handle
2. Trim off lower main body
3. Reloft main body

The big advantage of this method is that it simplifies the loft to the connection between the surface bodies. Notice also that it is vitally important the order in which the features are created. It would probably be impossible to make it work if the lofts were reversed in order.

So let's get started.

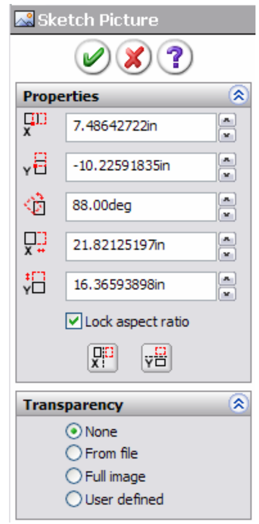
When working with digital pictures or digitized images, there is nothing like the **Sketch Picture** to get you going. This is found on the Sketch toolbar, or in the menus at Tools, Sketch Tools, Sketch Picture. You must have a sketch open to activate the icon.

1. Open a new part.
2. Open a sketch on the Right plane.
3. Draw a horizontal construction line *above* the Origin.
4. Dimension the line 9" from the Origin.
5. Click the Sketch Picture toolbar button, and browse to the file called wateringcan1.jpg in the materials for this chapter on the CD provided with this book.
6. Resize the image by dragging the handles on the sides and the corners so that the distance from the bottom of the

can to the low point of the saddle on top is about 9”. To finish with the positioning and sizing of the image, click the green check in the Confirmation Corner in the upper right of the graphics window. Set the rotation of the image to 358º.

> **Note:** Working from digital photos and hand sketches can be frightfully approximate. It is much easier if you are working from perfectly created digital images, but small imperfections of angles and perspective will introduce errors into the image. Use the image as a reference, even if that reference turns out to be for shape only, and then get real dimensions from physical measurements if the object is available.

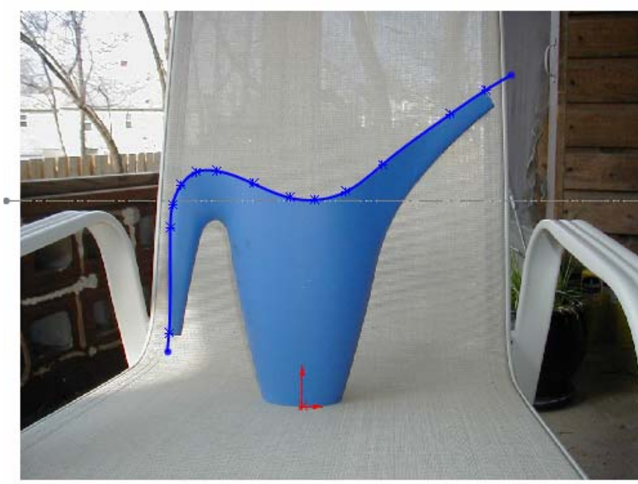


7. Exit the sketch and rename it Right View.
8. Go through the same process on the Top plane with the image waterincan2.jpg. Use the first picture as a size reference. You may need to rotate the view to something other than the standard 90 degree increments, for this image, 89º works best. Use the PropertyManager to rotate the image.

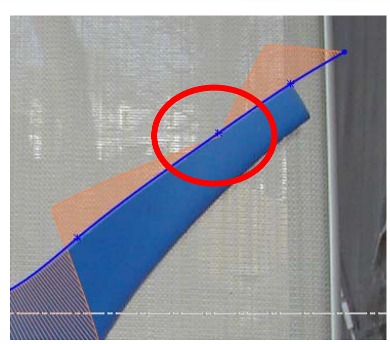
> **Note:** Notice that the image can be mirrored about X and Y, and a background color can be set to transparent. These images do not lend themselves to transparent backgrounds because the background is not a solid color.

9. Exit the second sketch and rename it Top View.

10. Open another sketch on the Right plane, and trace a spline over the upper edge of the part. Make the spline about one control point longer than the actual edge so that the tangency of the spline is correct at the ends. You do not need a lot of control points. Make control points closer together at the tight corner near the handle, approximately as shown in the image to the right.

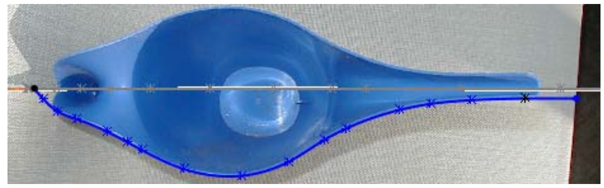


You should check your splines with a curvature comb to make sure that they do not have any unsightly inflections or gross curvature changes. For example, in this image you can see that on the spout there is a slight reverse in curvature at one point which was unintentional, and needs to be fixed.



Curvature comb can be found on the spline RMB menu, in the Spline toolbar or in the main menus at Tools, Spline Tools, Curvature Comb. A spline must be selected to activate the menu option and toolbar button.

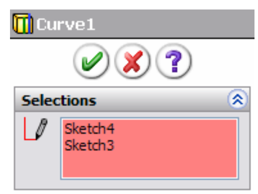
11. Do the same from the Top plane, but only trace *half* of the model. The spline should only touch the center plane at a single point. Where it touches the center plane it does not need to be perpendicular to it, since we will add a fillet after the model is mirrored.



The left-most point of this sketch shown in the image above should be coincident with the spline control point in the previous sketch located at the very tip of the handle. This will cause the projected curve to touch the Right plane at the very end of the handle.

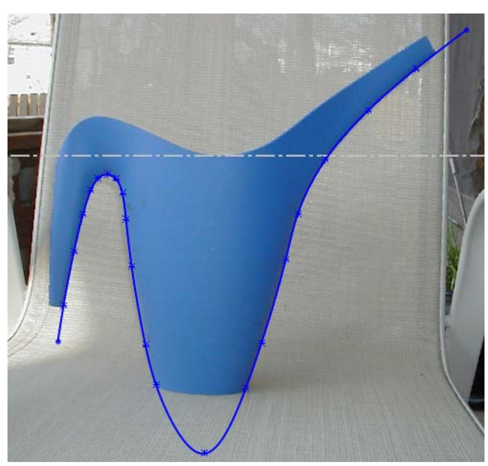
> **Note:** In the top image, the edge of the handle shown on the left seems to reverse itself and go in the opposite direction as it goes down in the Z direction, but this is simply an effect of perspective. Keep the spline extending in the same direction as the rest of the edge, approximately as shown in the image. You may also want to compensate for perspective by making the spout slightly narrower as it reaches the open end.

12. After exiting the top view sketch, click the Projected Curve button on the Curves toolbar, or get it rough the menus at Insert, Curve, Projected.
13. Select the Sketch on Sketch option and select the two sketches with the splines. Click OK.



> **Note:** There may be a display bug in SolidWorks which causes curves to not be visible when a sketch picture is in the background. To work around this, select the curve first, and then rotate the view to the desired orientation. You still cannot see the selected curve while rotating, but as soon as you release the mouse button, it will become visible.

14. Open another new sketch on the Right plane, and sketch the bottom curve, or what the can looks like at the center plane. Extend the spline as shown past the flat bottom so that the entire shape is made from one continuous spline. Again use a curvature comb to check the spline before closing the sketch.

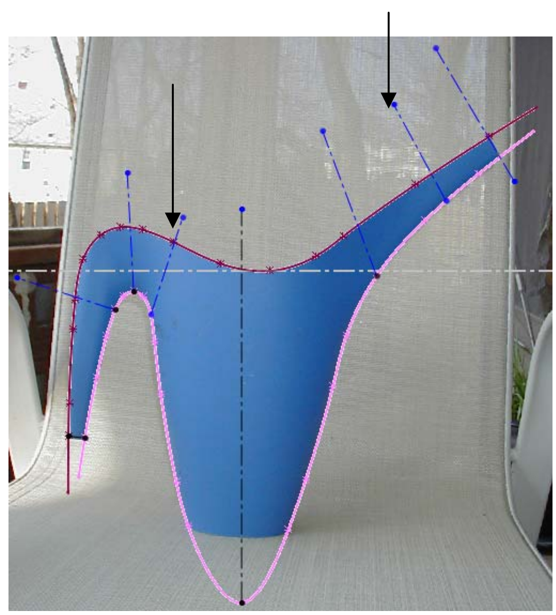


Remember to overbuild the spline on both ends. It is rarely an advantage to have a curve end *exactly* where you want a face to end, unless it is the beginning of a sweep, in which case the path must end at the profile plane.

15. Again make a new sketch on the Right plane. This one will simply be a construction sketch. To create a loft, you need several profiles, and for each profile you need a plane. This sketch will position and orient plane placement. The planes will be made using the "at angle" plane type, such that they will be made relative to the Right plane, at an angle of 90º through the selected construction line.

    Part of the planning that goes into this deals with understanding how the features work that you are going to use. For this particular shape, it is conceivable that one of several features could be used: Loft, Fill, Boundary or Sweep. The question comes down to which feature do you have the most confidence in, and which gives you the most control? In this case I have selected Loft because it is a very reliable feature, and it is usually easy to understand the feedback it gives.

    When a Loft feature uses multiple guide curves, the *first* guide curve drives the direction of the U-V mesh. In this case, the bottom curve is pretty radical, with a sharp kink in it at the bottom, so it would not make a good choice to drive the mesh. For this reason, the projected curve will be used for this purpose.

    With that decided, we can now place the loft profile sketch planes. Certainly we need one each at the end of the spout and the end of the handle, and another at the middle deepest point. This is very much like selecting where to place control points on a spline, since a lofted surface is just a two-dimensional version of a one-dimensional spline.

    The image to the right shows how the sample part has been created. Note the two lines with arrows pointing to them. I found that profiles were necessary at these points to avoid certain geometrical problems. How would you know that kind of thing without being told? You wouldn't. You would figure it out the way I did, by troubleshooting the original loft.

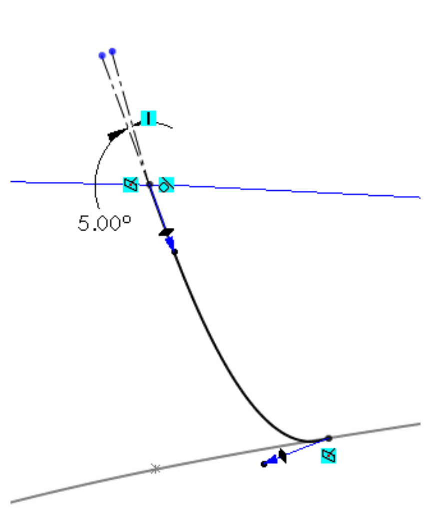
    Rarely do you get complete perfection on the first attempt. You wind up tugging and pulling and tweaking and experimenting.

Exit the sketch and rename it Plane Placement Sketch.

16. Create a new plane at each construction line, 90º from the Right plane through the selected line.

17. It helps at this point to turn off the sketch pictures, partially due to the display bug with the projected curve. You can turn off the pictures in one of two ways. One way is to simply hide the sketch or all sketches. The other is to click the + next to the sketch, RMB on the sketch picture under the sketch and select Suppress.

18. Open a new sketch on the plane at the spout end of the watercan. Sketch two construction lines, one vertical, the other connected to the first line's lower end, and then sketch a two-point spline, one end at the intersection of the construction lines, the other anywhere where it does not create an automatic relation.

    Make a 5º angle dimension between the construction lines, and make the spline tangent to the line that is not vertical. Create Pierce relations between each end of the spline and the guide curves, the end with the construction lines should be pierced by the projected curve.

    Lastly, select the spline to display the handles, then select the handle near the end pierced by the bottom sketch spline, and assign the handle a Horizontal sketch relation. The result should look like the image above. Sketch relation icons have been shown for clarity.

    Exit the sketch when complete.

19. This same sketch will be reused on each of the sketch planes. To avoid recreating the same geometry over and over again, click on the sketch in the FeatureManager, press Ctrl-C, then click on a plane in the FeatureManager and click Ctrl-V.

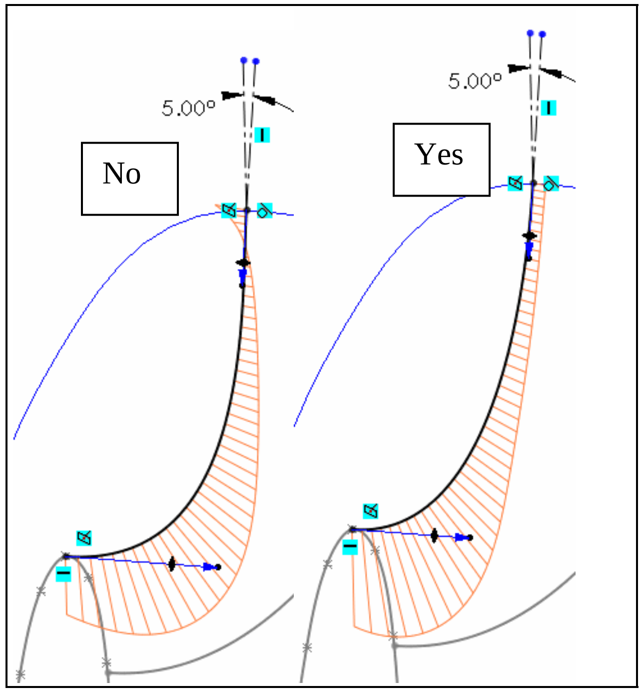
    The copied sketch copies everything except external sketch relations, which means that the pierce relations will have to be recreated in each sketch.

    The 5º angle is to apply a draft to the opening of the watercan. The sketch is never pointing exactly in the line of draw, but it should still ensure that there are no undercuts.

    > **Note:** Sometimes when reconnecting these sketches to the projected curve and the spline, the sketch will overdefine for no obvious reason. If this

happens, select and delete the handle at the spline end of the spline, then once it is reattached to its pierce point, recreate the Horizontal relation to the handle.

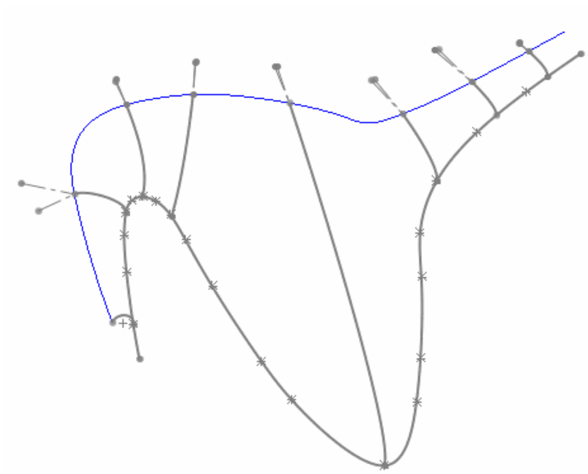
20. When editing the individual loft profiles to add the pierce relations, you should also apply the curvature comb and adjust the lower handle to make sure the bulge is sufficient without being too much. The curvature comb should remain on the outside of the spline. If it moves to the inside, that means that the curvature has *inflected* and a portion of the spline is *concave* when it should be *convex* with the rest of the spline.



    This shape can be achieved by pulling the Weighting handle on both ends of the spline. The goal is to get a curvature comb that is as smooth as possible, and remains completely on one side of the spline.

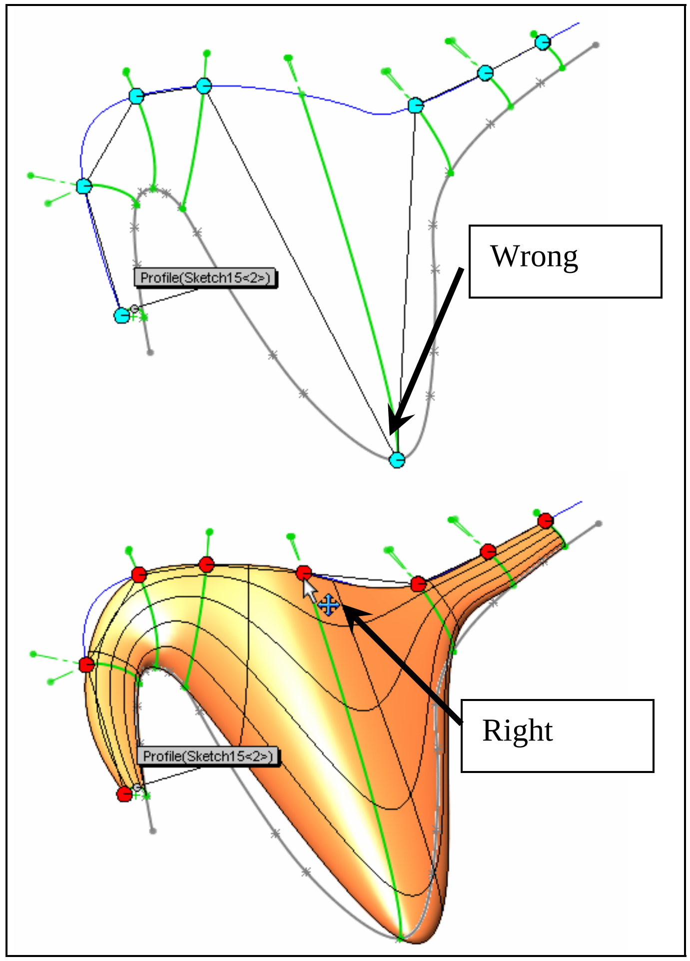
    This is important to maintain the draft for the opening. If the curve flips convexity, this may cause an undercut situation.

21. For the last two profiles, the handle has turned the corner, and draft is not important for those two profiles. The very last profile at the end of the handle should be a semi circle, approximately as shown.



22. The next step is where the fun begins. Start the Surface Loft feature.

Select each of the loft profiles *in order* at approximately the same location. Don't select them at the endpoints, but selecting toward one end near an endpoint is probably the best idea.



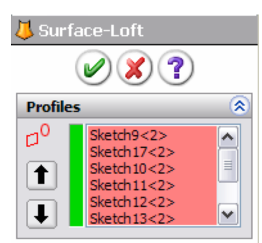
Several things can go wrong when selecting profiles. One thing that can happen is that you might select one of the profiles in the wrong location. If that happens, the loft preview might fail, or it might display a twisted surface. This is easy to fix. Just select the blue dot called a **connector** and move it to the other end of the spline. The connectors should all attach to the same end of the profiles.

Notice that here in this preview you can also see the U-V mesh. It is a little contorted, but it is nonetheless a four-sided mesh.

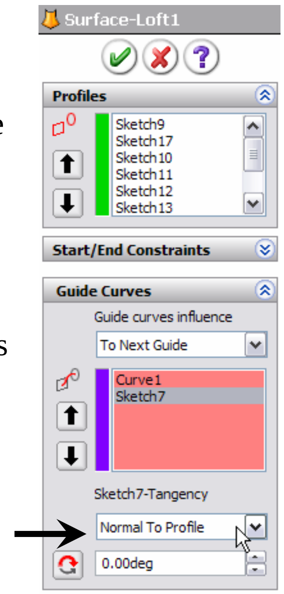
Notice also that the loft doesn't fit the projected curve and bottom spline very well. We remedy that in the next step.

A second thing that can go wrong is that you miss one of the profiles, or you select the profiles in the wrong order. This is also easy to fix. Normally you can just select the missed profile, and SolidWorks will sometimes try to automatically reorder the profiles. They must be listed in the correct order in the Profile selection box – not the order in which they were created, but the order in which they appear in the loft, starting at one end and going to the other. If you have a closed loop loft, the order is still important, but the starting point is not.

If SolidWorks does not reorder the profiles automatically, you will need to select the one you need to manually reorder in the Profile selection box, and use the Up and Down arrows to the left of the box in the PropertyManager, as shown in the image to the right.



23. Once the profiles are all selected, and the preview is showing, expand the Guide Curves panel of the Surface Loft PropertyManager and select the Projected Curve *first*, then select the bottom spline. Notice that the shape changes or the loft preview may even fail if you use the arrows to reorder the guide curves. This is because the first guide curve in the list is driving the shape of the U-V mesh.

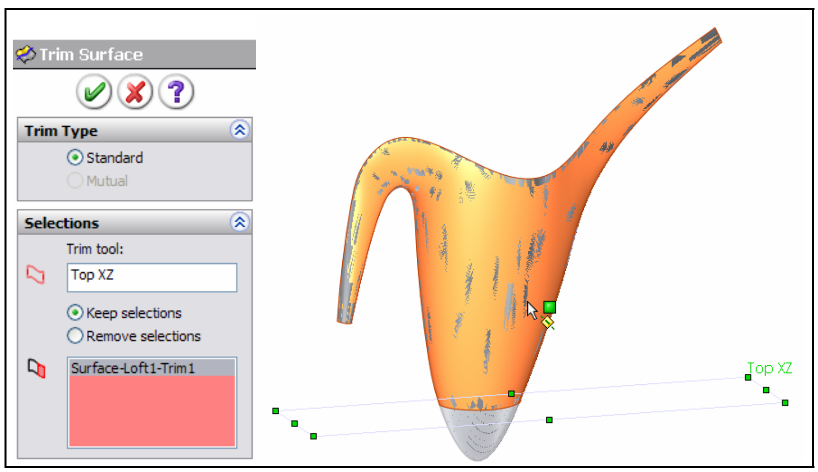


With the bottom spline selected in the Guide Curves selection box, set the Tangency Type to Normal to Profile, as shown in the image to the right. This is because the part is symmetrical, and we are modeling just half of it, the part has to be tangent across the center plane.

In this situation we do not need to set any end tangency conditions. If everything looks good, click OK.

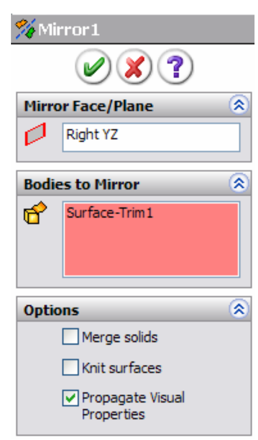
24. Next we trim the bottom off of the watercan. Click on the Trim icon on the Surfaces toolbar, or from the menus select Insert, Surfaces, Trim.

In the Trim Tool selection box, pick the Top plane.



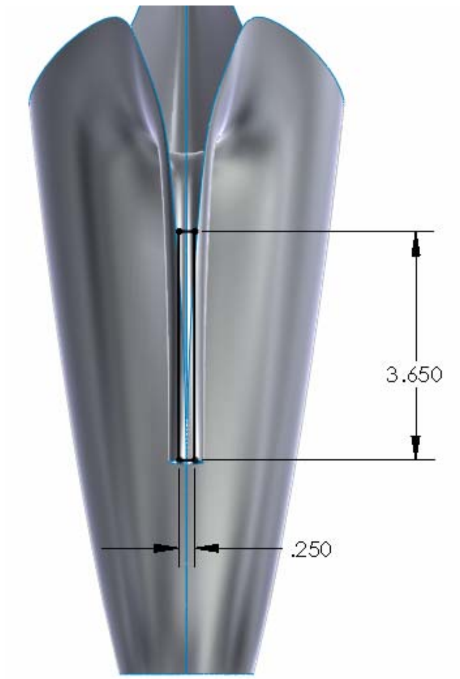
With the lower selection box activated, notice that portions of the model change colors when you float the cursor over them. Make sure that the option is set to Keep Selections, and click the upper part of the watercan. Click OK to accept the result.

25. Since this is a symmetrical part and we have modeled only one half of it, we now need to mirror the part. Use the Right plane and mirror the surface body. Do not use the Knit Surfaces option. The surface will not knit because the handle has a single point where it touches the Right plane, and this will cause the knit to fail.



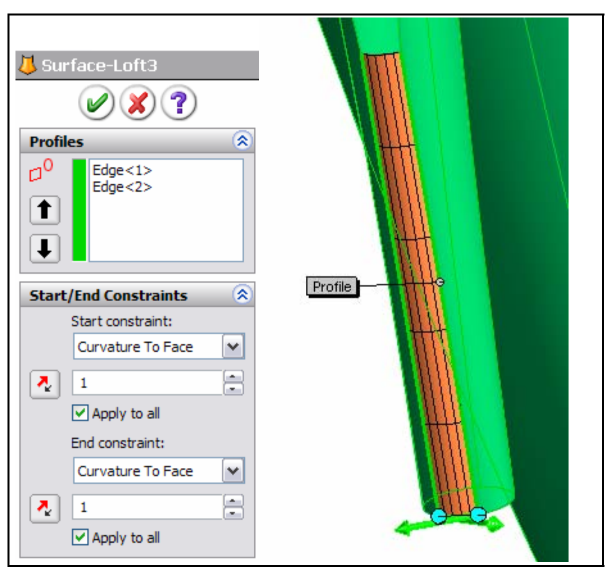
26. Change to a Front view of the model, which should show you the handle. Open a new sketch on the Front plane, and sketch a rectangle .250” wide and 3.650” tall. Both lower corners of the rectangle should have relations to the outer edge of the handle, as shown in the image to the right.

    With this sketch we are going to trim away part of the handle so that it can be recreated in a way that better reflects the shape that we are trying to achieve.

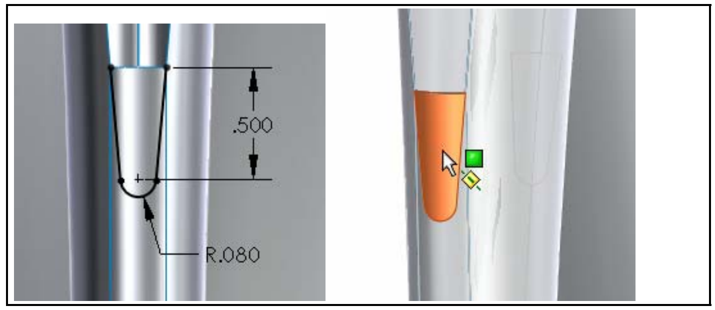


27. Next, use the Trim command again, but this time change the option to Remove Selection, and pick the two triangles of surface on the handle.

28. Create a surface loft between the straight edges created by the trim in the last step. Expand the Start/End Constraints panel and set both edges to Curvature to Face, accepting all of the defaults.

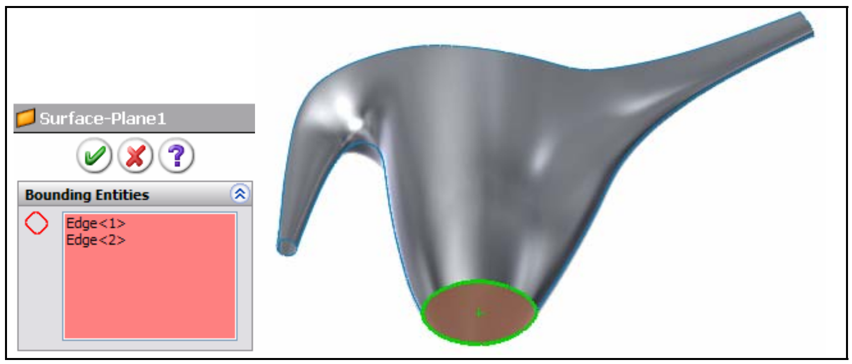


29. Again shift to a Front view, and sketch on the Front plane two lines and an arc as shown in the image to the right.

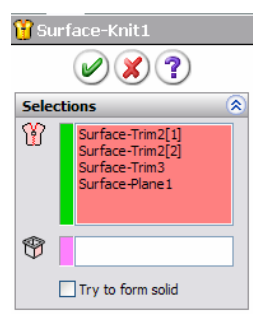


30. Use the Trim feature to remove the small notch in the handle loft we just added.

31. Now we need to add a flat for the bottom, so tilt the model over so you can see the bottom, and select the Planar Surface from the Surfaces toolbar, and select both edges around the hole.



32. At this point, all of the faces you need to create for the part exist. You may notice that edges of the surfaces are a different color from how they normally display. This is due to a setting in Tools, Options, Colors called Surfaces, Open Edges. This is an indication to you that the surface features are still all independent bodies. This can be verified by looking at the Surface Bodies folder at the top of the FeatureManager, which should indicate four surface bodies.



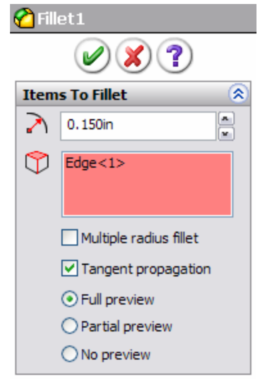
The next step is to join all of the surface bodies together into a single body, thicken it to make it into a solid, and then add some final features.

From the Surfaces toolbar, select the Knit Surfaces button. Select all of the surface bodies. This can be done from the graphics window or the Surface Bodies folder.

Click OK when you are done. If you get an error, check to make sure that all of the bodies are selected, and that there are no gaps or overlaps between the bodies. Once the bodies are knit together, notice that the colored edges have changed.

33. Before making this into a solid, add a fillet around the bottom of the watercan. Fillets can only be added when the faces on either side of the fillet edge are members of the same body, which is to say that fillets can only be applied to a single body at a time.

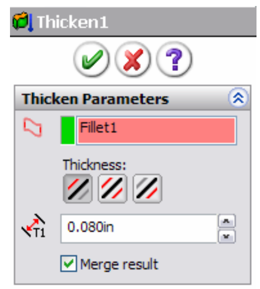
    Make the fillet around the bottom edge .100” radius.

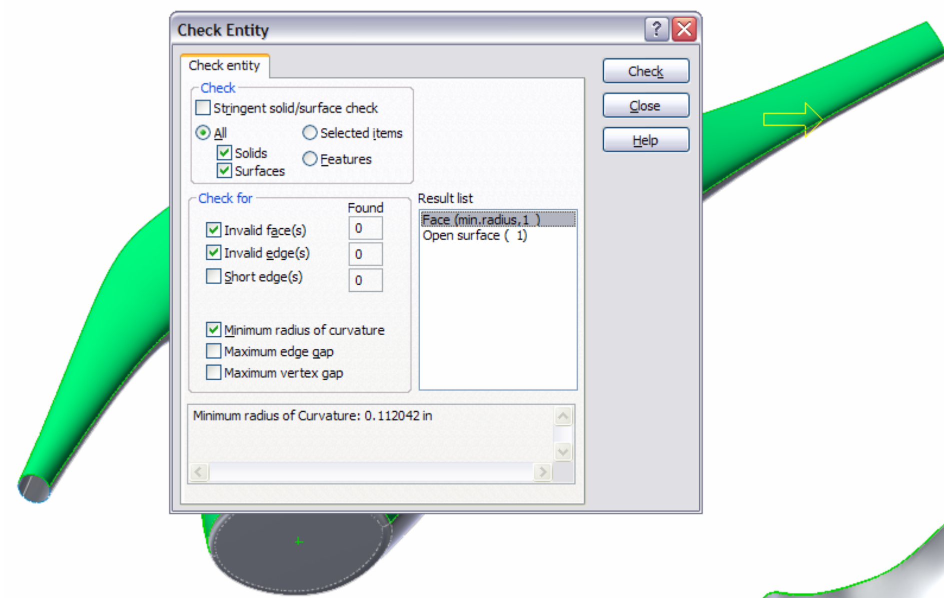


34. To make the model into a solid with a constant thickness, access the Thicken command from the menus at Insert, Boss/Base, Thicken.

    Give the model a thickness of .080” to the inside.

    Again, if you encounter errors, they are probably due to very small curvature at some point on the model. The best way to find errors of this sort is to use the Check utility, available through the menus at Tools, Check. Select the option to find minimum radius of curvature, and when the result displays, click on the result in the list box and SolidWorks will point to the area with minimum radius. If it is less than .080”, that is probably the cause of the error.





35. Create another fillet feature, this time going around all the edges of the outside of the rim with a radius of .040”.

36. Go water the flowers!