Easy Surfacing Techniques for Solid Models Using SolidWorks 2010

Step by step procedures

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Why use surfacing techniques on solids?

We can use surfacing techniques to repair or enhance solid models because a solid body in SolidWorks is actually made up of several faces. Faces are, in the end, surfaces.

It is sometimes hard to envision a solid body as made up of surfaces because in real life everything has a thickness and a mass, while surfaces have zero thickness. If I think of a block of wood and I cut an infinitely thin slice of it, no matter how thin, I will still see a solid block of wood. In SolidWorks, however, if I remove a face from a solid body, it will become a surface body and I’ll be able to see the inside of the body as a hollow shell that is made up of all the faces that enclose the volume. The images below show a solid body made of several faces on the left, and the same body, now a surface body, once some of the faces have been removed.

So, with that much said, it seems reasonable that we can modify solid models by modifying the faces (surfaces) in them. The following examples show a few ways to take advantage of surfacing techniques to repair or enhance solid models by eliminating geometry, repairing fillets and curves, blending faces, etc. I hope you find the examples useful.
Example 1: Eliminate Geometry

Use Delete Face to erase faces in the model. Pay attention to the options:

Delete will simply erase the face(s) and leave a hole in its place.

1. Open the file example1.sldprt. If asked about running feature recognition, answer “no”.
2. Click on Delete Face on the Surfaces Tab.
3. Select the option Delete in the property manager.
4. Choose the eight faces in the image and click OK to accept the feature.

5. The eight faces you chose will be erased and a hole will be left on its place. Notice also that the solid body is now a surface body.

Patch will erase the face(s) and extend surrounding faces to cover the hole.

1. In the Feature Manager Tree, right click on the Delete Face feature just created and choose Edit Feature from the flyout menu.
2. Using exactly the same eight faces as before, choose now the option Delete and Patch from the property manager. Mark the option to see preview and notice how edges of
Example 1: Eliminate Geometry

adjacent faces are being extended in order to patch the hole left by erasing the eight faces.

3. Click OK to accept the changes. The result: The model looks like those eight faces never existed.
**Example 1: Eliminate Geometry**

**Fill** will erase the face(s) and replace with one single surface.

1. In the Feature Manager Tree, right click the Delete Face feature and click on Edit Feature from the flyout menu.
2. Using the same eight faces as before, choose the option Delete and Fill from the property manager.
3. Click OK to accept changes. The result: the eight faces have been erased and replaced by one single surface that seems to fuse them all together.
Example 2: Repairing an ugly fillet

Every now and then, you’ll end up with ugly fillets and corners in your solid model. These bad faces can be simply unflattering for your model, or can also mean the difference between a shell operation that succeeds and one that fails. Use surfacing techniques to repair them by eliminating faces from the fillet and patching the hole with a new surface.

1. Open the file example2.sldprt.
2. Use Delete Face (Surfaces tab) to eliminate all the small faces in the fillet. Make sure to use the option Delete in the property manager. See image.

3. We’ll now trim part of a couple of other faces, to create a better shape for the hole and increasing the chances of achieving a better looking patch. Open a sketch on the planar face shown in the picture and sketch a line between the two vertices you see there.
Example 2: Repairing an ugly fillet

4. Use the sketch you just created to section the face. Go to Insert, Curve, Split Line. Under Type of Split, select Projection, under Selections, select the sketch and the same face it’s sketched on. See image.
Example 2: Repairing an ugly fillet

5. Follow the same procedure with the face on the left bottom that also has a sharp corner. See image.

6. As a result of the previous steps, we have sectioned the two bigger faces and can now safely eliminate only the small sharp corners that are getting in the way. Click on Delete Face and use the option Delete to erase them both at the same time. See image.
Example 2: Repairing an ugly fillet

7. We’ll now patch the hole. Click on Filled Surface (Surfaces tab). Select all edges around the hole. Make sure to get closer so you don’t miss any small edge by accident. There should be 10 edges total. If you enable the preview, you’ll see how a surface shows up and it’s somehow contoured in a fashion that tries to follow the shape of the other faces that meet at the vertex where the hole is.

8. Notice the callout that shows up pointing to each one of the edges as you choose them. By default, it always says Contact. This callout indicates the curvature control option at the edge. In other words, how is the curvature of the new surface going to be with respect to that of the pre-existing surfaces it meets at the edges? Contact simply creates a new surface, Tangent creates a surface that is tangent to the pre-existing one at the edge and Curvature creates a surface that shares the same curvature of the pre-existing one at the edge. You can adjust this option for each edge separately or apply the same to all. In this case, choose Tangent and mark the option Apply to all edges.

9. Scroll down to the bottom of the property manager and, under Options, choose Merge Result. If everything went according to instructions and you didn’t miss an edge while creating the patch, the option Try to form a solid should become available. Select that option. Click OK to accept.
Example 2: Repairing an ugly fillet

10. The result: You have once again a solid body and the fillet now looks like a smooth face that is tangent to all others adjacent to it. See image.

The following example shows a slightly different situation, although the procedure is very similar.

1. Open the file example2b.sldprt.
2. Click on Delete Face with the option Delete and select the face that appears in the image. Click OK to erase it.
Example 2: Repairing an ugly fillet

3. Click on Offset Surface (Surfaces tab) and select the face shown in the following image. Use an offset distance of zero. By using offset face we’ll create a copy of the face right on top of the original one. We do this because we need to trim this face and it’s often easier to trim a copy of it and then simply get rid of the original and replace it with the one we modified.
Example 2: Repairing an ugly fillet

4. Hide the model to facilitate working with the copy of the face (Right click on it on the graphics area or on the Feature manager tree and select Hide from the flyout menu).
5. Under Tools, Sketch Tools, select Face Curves. Face Curves is used to extract iso-parametric (UV) curves in surfaces. These curves are saved as 3D sketches. In the property manager, select the copy of the surface. You’ll see a mesh of curves appear. We need curves at a specific location at the point where this face meets the others in the model, so in the position vertex field make sure to select the vertex shown in the following image. The curve we want is the one shown in pink. Click OK to accept.

6. You will see a new 3D sketch has been created in the Feature Manager tree. We’ll use the 3D sketch to trim the copy of the face. We will use Trim Surface this time. Trim Surface allows you to trim and discard parts of a surface using sketches or other surfaces as your trimming tools. The surfaces are cut where they meet the other surfaces or the normal projection of the sketches and you choose what parts to keep and which ones to discard. Click on Trim Surface (Surfaces tab) and select the 3D sketch as the trimming tool. Select the part of the surface to keep as shown in the image.
Example 2: Repairing an ugly fillet

7. Make the model visible again and use Delete Face with the option Delete to erase the original face.
Example 2: Repairing an ugly fillet

8. Use Offset Surface again to create a copy of the cylindrical face as shown in the image. Use an offset distance of zero just like before.

9. Click on the copy of the face we just made and then go to Tools, Sketch Entities, Spline on Surface to sketch a spline that will be bound to this surface we just created. Sketch a two point spline between the points shown in the image.
Example 2: Repairing an ugly fillet

10. Add a tangent relation between the spline and the edge of the fillet.

11. Use this sketch to trim the surface, just like we did with the previous one. You can choose to temporarily hide the other two surface bodies (the model and the modified copy of the surface we worked with previously) if you wish.
Example 2: Repairing an ugly fillet

12. Show all surface bodies and use Delete Face with the option Delete to erase the original face and leave the modified copy in its place, just like we did with the other one.

13. By this time we have a nice looking hole waiting to be patched, but if we attempt to use Filled Surface right now we’ll run into trouble choosing the edges around the hole on the left side. This problem is solved by knitting the faces we have together into one surface body first. Click on Knit Surface (Surfaces tab) and choose all the surface bodies we have. There should be three surface bodies. Click OK to knit them together.

14. Now we can use Filled Surface to patch the hole. Click on Filled Surface and select all edges around the hole. There should be five edges. Choose Curvature Control Tangent and apply to all edges. See image.
Example 2: Repairing an ugly fillet

15. Scroll down to the bottom of the property manager and choose the option Merge Result. If everything went according to plan, the option Try to form solid should become available. Select it and click OK to accept the patch. The result is a new, smoother face where the ugly fillet used to be.
Example 3: The lumpy corner

1. Open file example3.sldprt. If you are prompted to start Feature Recognition say “no”. In this example, you’ll find a vase that has been previously sectioned using Cut with Surface to leave only a quarter of it. If you rotate and examine the model, you’ll notice a discontinuity on the corner at the upper edge of the vase. That’s what I called the lumpy corner. The idea is to get rid of the lump. This image shows a closer look to the corner on the upper edge. To get a closer view of the corner in SolidWorks, change to the custom view View1.

2. Use Offset Surface with a distance of zero to create a copy of the narrow inner face of the corner. See image.
Example 3: The lumpy corner

3. If you want to, hide the solid body to facilitate working with the copy of the face we just created. Click on the surface and go to Tools, Sketch Entities, Spline on Surface to sketch a two point spline on this surface between the two vertices shown in the image.

4. If you hid the solid body, show it again and apply a coincident relation between the spline and the edges of the vase. See image.

5. Use the spline to trim the copy of the face. Click on Trim Surface, select the spline as trimming tool and keep only the lower portion of the surface. See image.
Example 3: The lumpy corner

6. Use Delete Face with the option Delete to erase the original face and the face of the corner. This turns the solid body into a surface body. See image.

7. Now we have a hole in the corner that needs patching. We’ll use Filled Surface for this purpose. Click on Filled Surface (Surfaces tab) and select all the edges that surround the hole. There should be four edges. This time we can’t choose Tangent as a curvature.
Example 3: The lumpy corner

control option for all the edges, because we would end up with undesirable results. We can use Tangent for two of the edges and contact for the other two. See image.

8. Scroll to the bottom of the property manager and select the option Merge Result. If everything went fine and you didn’t miss any edge, the option to Try to form solid should be made available. Choose that option. Click OK to accept.

9. The result should be a solid body without the lump on the corner.
Example 4: Achieving a better blend

1. Open the file example4.sldprt. You’ll find a model of a shower head (use your imagination if you must). There’s a solid body, the head, and a surface body, the handle. We want to blend the head to the handle and make the transition nice and smooth. For this purpose, instead of adding a fillet, we will use surfacing techniques. We will begin by turning the solid body into a surface body by eliminating one of its faces using Delete Face with the Delete option. It must be a face that is easy to replace. The small planar face shown in the image will do just fine.

![Delete Face](image)

2. Now we need to make room for the blend. Notice the two sketches “Trimming sketch top” and “Trimming sketch bottom”. We will use these sketches to trim the surfaces of the head and handle respectively. Click on Trim Surface and select the “Trimming sketch top” as the trimming tool and the upper part of the head as the area to keep. See image.

![Trim Surface](image)
Example 4: Achieving a better blend

3. Notice that, after trimming, two edges were created on the head. That’s because the area of the head that we trimmed was made up of two faces and not just one. See image.
Example 4: Achieving a better blend

4. Since there are two edges on the head, we should create two edges on the handle as well. Notice that the Trimming sketch bottom is simply a line, but it’s been cut in two parts using Split Entities (Tools, Sketch Tools). This will produce two edges when trimming the surface of the handle using this sketch. If we trimmed using the line without splitting it, we would be left with only one oval edge on the handle. Use Trim Surface again; select the Trimming sketch bottom as the trimming tool and the lower part of the handle as the area of the surface to keep. See image.

5. Using Lofted Surface, create a surface between the two shorter edges in the head and handle. In the property manager, expand the section about start/end constraints. Notice that, similarly to Filled Surface, there are options here to choose and that will determine the curvature of this lofted surface with respect to that of the faces it’s connecting through the loft. If your Start profile is the one for the edge, choose Curvature to face as a start constraint. If your End profile is the one of the handle, choose Tangency to face as the end constraint. In other words: Curvature to face for the edge of the head, Tangency to face for the edge of the handle. Notice there’s a couple of arrows that appear next to the edges as soon as you change the constraints; their default length is 1. Changing the length of these arrows also changes how much these constraints influence the whole lofted surface. Too much or too little and you may end up with undesirable results as the surface begins to wrinkle. You can play with these values until you’re satisfied. For this example, however, leave both arrows a length of 1.
Example 4: Achieving a better blend

6. Attempting to do another loft between the other two edges results in a surface that overlaps the first one and prevents us from knitting the model back into a solid, so we’ll cover that with a Filled Surface, instead. Click on Filled Surface and select the all four edges. Change the curvature control options to Tangent for the edge of the handle and Curvature for the other three edges. See image. It looks weird in the preview, but it works.
7. Using Planar Surface, create again the face we deleted previously. Click on Planar Surface and select the two edges shown. Make sure to select the inner edge also and not only the outer edge, otherwise a circular face will be created under the dome and will prevent us from knitting the surfaces back into a solid. Click OK.

8. Use Planar Surface again to create a face at the end of the handle.

9. Use Knit Surfaces to knit all the surface bodies back into a solid. Click on Knit Surface and select all the surfaces. Instead of selecting from the graphics area, it’s easier if you
Example 4: Achieving a better blend

open the folder Surface Bodies from the Feature Manager tree and select all the surface bodies in it. If everything went fine and there are no overlapping surfaces or gaps, the option to Try to form solid should be available. Choose that option and click OK. The result is a solid body with a nicer smoother blend between head and handle.
Example 5: The cover

1. Open the file example5.sldasm. Here you have a three part assembly of a pair of pliers. We’ll use surfacing techniques to add a plastic sleeve or cover to the handle. We’ll be working in the context of the assembly throughout this example, but you can also chose to simply open the part if it’s easier for you that way. In the Feature Manager tree, right click on Plier Half (Bottom Half) and select Edit Part from the flyout menu.

2. Once in Edit Part mode, click on Offset Surface and select all the faces that form the handle, as shown in the image. There should be 11 faces. Use an offset distance of zero, since we only want to make a copy of the faces on top of the originals.

3. Hide the solid body by right clicking on it and selecting Hide from the flyout menu. This will make it easier to work with the surface.

4. Open a sketch on the top plane and offset one of the edges of the surface a distance of 2 mm, as shown in the image. Extend its ends so the sketch completely intersects the surface. We’ll use this sketch to trim the surface.
5. Click on Trim Surface and select the sketch we just created as the trimming tool. Select the part of the surface shown in the image as the area to keep.

6. All that's left to do is thicken the surface to a solid. Usually, whenever you use Thicken on multiple surfaces, you will have to knit them together first. We don’t have to knit the surfaces in this case because they already behave as one single surface body. In fact, they’ve been behaving this way since the very moment we offset them all together. Click on Thicken and select the surface we just trimmed. Thicken gives you three options for direction in which to add thickness. Choose the first option, to add thickness to the outside. Use a thickness of 1.5 mm. Do not merge the result.
Example 5: The cover

7. Now we have two solid bodies in this part document: the handle and the sleeve or cover we just created for it. Show the handle again and change the appearance of the cover, if you please. Here I’ve changed its color to blue. You can use different materials for each body and/or save each one of them as a separate part using Save Bodies, for instance.
Example 6: Freeform

1. Open the file example6.sldprt. You will find a copy of the same model of showerhead we worked with in the example 4. This time, we’ll use Freeform to add a grip to the handle, but first, we must section the face of the handle and create a smaller, more localized face for the Freeform operation. Go to Insert, Curve, Split Line. Under type of split, select projection. Under sketch to project, select the “Sketch for split line” from the graphics area. Under faces to split, select the face of the handle. See Image.

2. Click on Freeform. Under surface to deform, select the smaller face in the handle, the one that we just created in the previous step. Freeform allows us to take advantage of symmetry. Choose Direction One Symmetry to make the deformation symmetrical along the length of the face. Notice the callout next to the edge of the face. As is the case with other surfacing tools, Freeform gives us a choice to control how the curvature of the face to deform will be with respect to that of the adjacent face or faces at the edge. Choose Curvature for a smoother result. See image.
3. In the property manager, under Control Curves, choose Through points and click on Add Curves. In the graphics area add a curve exactly at the location of the plane of symmetry. We only need one curve. We’ll use this curve to deform the face. Notice the change in the cursor from an arrow to a pencil and the symbol of a small spline. This is because the control curve is actually like a spline. Notice the green line that appears on the surface as we hover over it. Click to add a curve when the green line is coincident with the plane of symmetry, just as in the image. Then right click with your mouse or click on Add Curves again to stop adding any more curves.
Example 6: Freeform

4. Under Control Points, click on Add Points. We’ll be adding a few points to our curve. This is similar to having a spline with several inflection points. You can add as many or as little as you think you may need for better control of your shape. You may find that less is more. When you are done, right click with your mouse or click on Add Points again to stop adding any more points.
Example 6: Freeform

5. Click on any of the points and you’ll see a Triad appear. By dragging the Triad, you can reshape the face by pushing and pulling. Although you can drag the Triad by its center, I find it’s always better to use the arrows, so you know exactly in which direction you are dragging and you are less likely to end with undesirable results. Use the red arrow to drag a few of the points up and some others down along the plane of symmetry. Alternate the direction in which you drag the points to create hills and valleys. You can guide yourself using the ruler you see in the graphics area, or you can directly type the distance in the appropriate field in the property manager on the left.

6. When you’re done pushing and pulling and you’re happy with the result, click OK to accept the feature. The result should be a grip like the one in the image and this time there’s nothing to knit or thicken because the body never stopped being a solid while we were using Freeform to reshape the face of the handle.
Example 6: Freeform

Although you can’t actually use a sketch as a guide curve like you would with a loft, for instance, you can display a sketch while using Freeform and use it to guide yourself while dragging points.
Example 7: Thickened cut

1. Open the file example7.sldasm. You’ll find an assembly of a pair of scissors. We will use surfacing techniques to create a cut for some lettering on one of the handles. We’ll be working in the context of the assembly, merely for convenience, but feel free to open the part if you prefer. In the Feature Manager tree, right click on scissor1 and select Edit Part from the flyout menu.

2. Once in Edit Part mode, go to Insert, Curve, Split Line. Under Type of Split choose projection. Under sketch to project choose the sketch “Sketch for split line” from the Feature Manager tree. This sketch includes a rectangular area and some lettering. Under faces to split select the two faces of the handle shown in the image. Click OK.

3. Use Offset Surface with an offset distance of zero to create a copy of the faces surrounding the letters, as shown in the image.
Example 7: Thickened cut

4. Use Thickened Cut to make a cut on the scissors handle using the copy of the face we created in the previous step. Thicken to the inside, using a thickness of 1 mm. See image. Thickened cut works in a similar way to Thicken. It thickens a surface, only instead of creating a solid, it thickens to cut through another solid.

5. Notice that the little face inside the letter “a” didn’t get thickened and, as a result, that part of the handle wasn’t cut. Simply repeat the same procedure as in last step for that little face alone.

Both Thicken and Thickened Cut can be used for parts that require overmold, for instance.
For more information:

- Surfacing class with your VAR
- SolidWorks Surfacing and Complex Shape Modeling Bible by Matt Lombard
- A few websites:
  - [www.dezignstuff.com/blog/](http://www.dezignstuff.com/blog/)
  - [www.solidworkstutorials.com](http://www.solidworkstutorials.com)
  - [www.mikejwilson.com](http://www.mikejwilson.com)
  - [www.swxdesign.com/](http://www.swxdesign.com/)
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