WR - 11/19
Isaac Hewett

How to Surface Model

Preface:

This is a guide to get you started with surface modeling. The information will be broken up into a few different sections listed below. Each function will also go over common issues with why the function might be failing.

Organization:

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General design procedure:

This will be pretty basic and will cover the overarching goals of each step in creating a model for your surface designed part.

The first step will be your general design, for our purposes in formula this means the design of the bodywork part or wing. Look at your design parameters and come up with an idea of what you want to do.

Before modeling, break up your part into sections based on how you will model it using the following surface functions.

As you plan out your part it is essential to consider how it will be manufactured. For most parts we will need to create molds. For molds it is important to think about how they will come off of the plug. It is easy to have geometry that requires the plug to be split into two parts in order for the molds to be separable from the plugs. For example our current nosecone and side pods are this way.

Foundation knowledge:

Sketches:

A sketch is a drawing in either 2D or 3D. For certain operations it may need to be a closed sketch meaning that there is a defined inside and outside based on the sketch containing a closed loop.

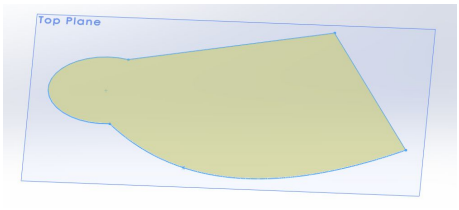
Splines:

Splines are curves that can change curvature at indicated points and are used for creating organic, flowing shapes. When using a spine you want to add a point only when the curvature direction changes. For organic curves you want *as few as points possible*.

Convert entities:

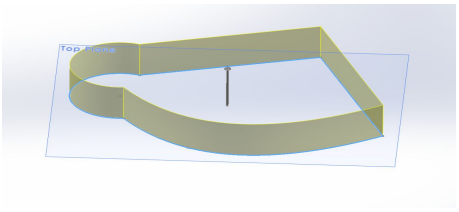
A tool to select a line/point and import it into your new sketch. This is used for creating continuous parts with multiple features. If the line/point does not coincide with the new sketch it will project the line/point normal to the sketch.

Basic Functions:



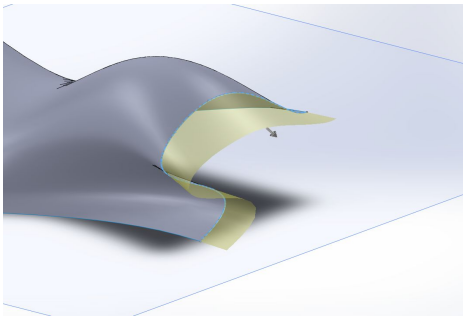
Planar Surface:

This function is pretty simple, any closed sketch can be made into a planar surface. This creates a surface bounded by the sketch.



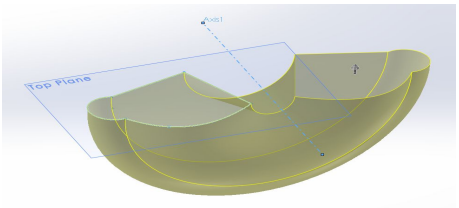
Surface Extrude:

This function will take any sketch and extrude the lines in a specified direction (default is perpendicular to sketch plane).



Surface Extend:

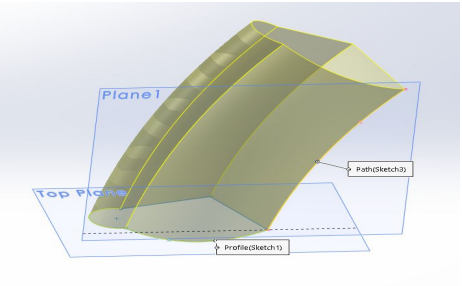
This is similar to surface extrude but allows you to extend the surface with the same curvature, this is super useful for extending edges for plug oversizing.



Surface Revolve:

This will take any sketch and revolve it around a given axis. A 180 degree revolution is pictured.

Complicated Functions:



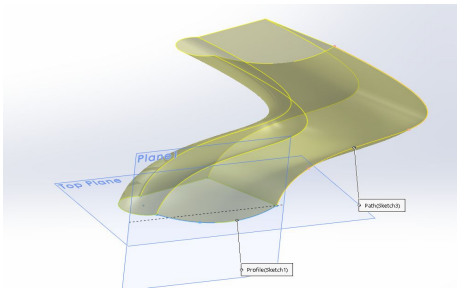
Surface Sweep:

This tool is a combination of extrude and revolve, allowing you to sweep a sketch (closed or open) along a path. This path can be in a line in a sketch created by you or a line created by some other

geometry. In the case of the picture to the right, the example sketch I created is being swept along a spline in another sketch.

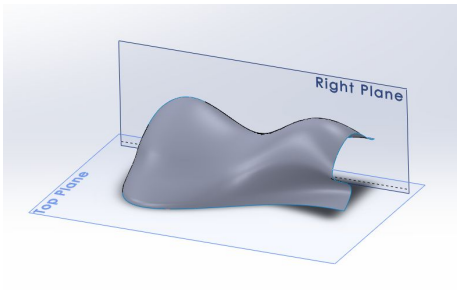
If this sweep is failing to create, a common issue is path is concave towards the sketch creating self-intersecting geometry. Try changing the path to be less concave.

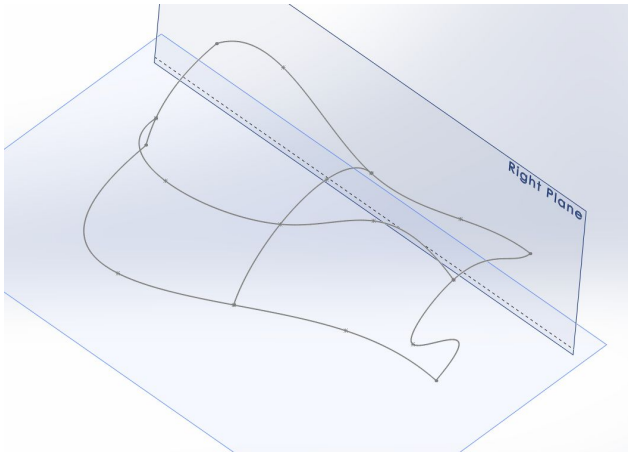
There is also an option to keep normal constant which allows for more complicated sweeps with the surface remaining parallel to the original sketch. As in the second picture.

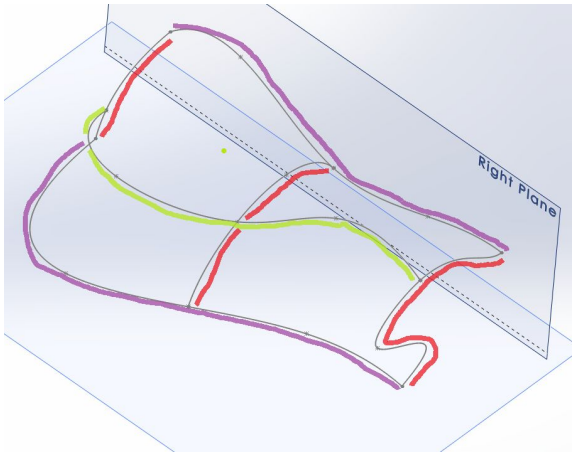


Surface Loft:

This is your bread and butter for organic surface shapes. The basis is two sketches, start and end, you can add intermediate sketches to help mold the loft. The next two pictures are what I call the skeleton model. I have also included a color coded version to help distinguish what lines serve what purpose in the surface loft.







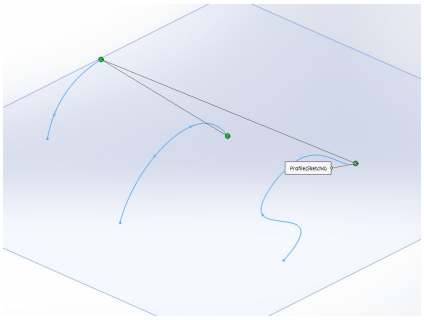
The red curves are the profiles, they show what a section of the surface would be in their plane. They are used as a start and end point, they also can be midpoints but

the order that they are selected indicated the loft order with an unintentional loft order causing the loft to fail to create.

The purple and yellow lines are guild lines / curves, they can be useful to get correct edge geometry, as well as manipulate shape of the surface loft in planes perpendicular to the profiles.

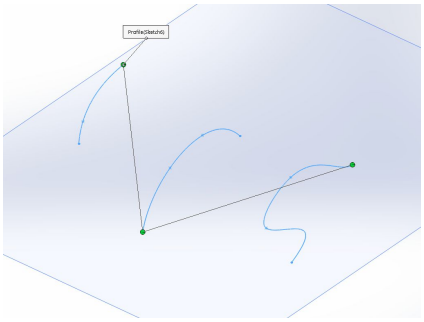
The yellow line is special on it's own as it is a 3D sketch. Although harder to define correctly, the 3D sketch is helpful to create geometry outside of the silhouette profile of the red and purple lines.

There are a few common errors that can occur when attempting to create your surface loft. When creating the loft, to identify the error it is important to pay attention to when the preview stops working. This usually happens when the sketch is added to profiles or guidelines that is causing the error.



If it is a profile it is common that the sketches are lofing out of order and creating self-intersecting geometry. Make sure they are ordered correctly in the profile list. Also when in doubt it is easy to clear the profile selection list and simply re-select the sketches in the order you want them to loft in.

If the issue is with guidelines, and it usually is, make sure that all guidelines are coincident with the profiles that they pass through. If they are not it will not work.



Finally, it is common that the loft profile sketches ends are switched, this creates obviously incorrect geometry that looks like a crazy "X" or will cause the surface loft to fail to create. To remedy this, when creating the loft, grab the big green dots on the profile and drag it to the edges of the sketch that it should loft to.

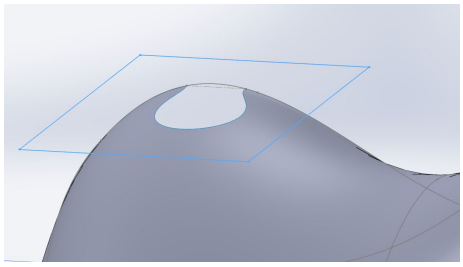
Boundary Surface:

Boundary surfaces are very similar to surface lofts as they use the same components, but boundary surfaces allow you to change tangency to profiles are not the start and end profiles. This allows you to create more accurate / complex shapes. If needed, this minor shortcoming of surface lofts can usually be remedied by the addition of more profiles.

Clean up functions:

Trimming:

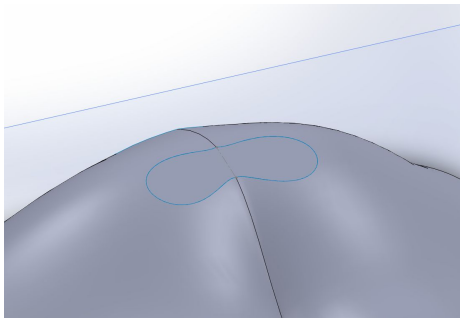
Trimming is using a surface to cut away at another. In the example pictured, I used a planar surface to slice the top off of my organic shape. The planar surface is hidden but the boundary is shown.



Filling:

This can be used to fix up geometry or create surfaces. It simply uses the bounding surfaces / edges to create a surface.

If the fill is failing to create, it is possible that edges selected to create the fill are not completely closed. They must be fully closed to fill the surface.



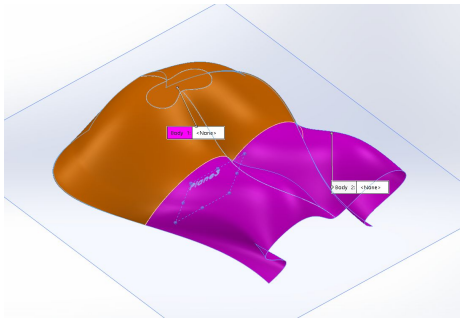
Kitting:

Kitting allows for the combining of surfaces that edges coincide with each other. In the picture for the surface fill I have mirrored my example shape and filled the whole on top. The blue line that outlines the fill shows a surface edge. If this exists in the middle of your part it means you need to knit the surfaces into one.

When kitting it is common to experience errors where the knit fails or it knits but there are still surface edges (blue lines) in the middle of the part. This is usually due to small issues with the geometry, for example the edges of the surfaces overlapping or small gaps. In order to remedy this, find the locations that are issues and either trim away the overlap or use the filling operation to fill the gap.

Splitting:

Splitting is the opposite of knitting, allowing you to cut your surface into parts. This is useful in a formula context for creating molds of two halves of a part. In the picture to the right I split the surfaces by a plane. There is also an option to save the surfaces to separate files if that is desirable.



Creating a 3D part:

Thickening:

This function is really self explanatory, it will take your infinitely thin surface and thicken it. You get to specify the direction.

This sounds great but errors are common with thickening. Commonly when thickening you want to thicken towards the concave side of the surface, for example the inside of the side pods or nose cone. The issue with this is that the thicken will commonly fail. This is due to radii existing on the surface that are smaller than your specified thickness. This would delete the small curves on the inside.

How to fix this is to inspect the surface and locate the location of these small curves. Once they are found you can add another profile that straightens them out due to the fact that they are usually unintentional. Also decreasing the amount to thicken by can allow it to work.

Surface cut & Keep / Delete:

This is mainly for making plugs and is a combination of a few features. The general concept is to extrude a base that is the size of the plug before machining. Then create a surface cut by selecting your surface and the base extrude. If they intersect correctly (the surface needs to be bigger or equal to the boss) this will separate the edges of the surface from the plug model, as well as cut the boss down to the surface. Then use the Keep / Delete tool to remove any excess surface.

The common error here is that your surface is smaller than the base and therefore cannot cut it. To fix this simply oversize the edges of your surface and then delete the extra in the keep / delete.