Student Documentation Package- Example

CDI 63:
ADVANCED SURFACING IN SOLIDWORKS

De Anza College

Instructor: Sam Tanna

Spring 2010

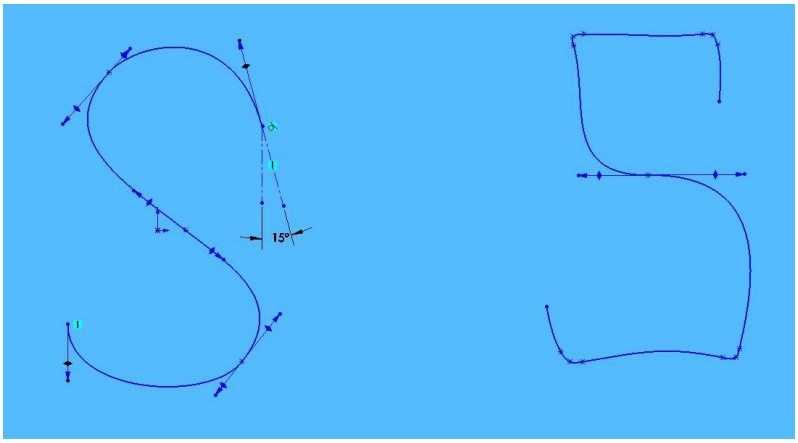
Student: Jill

CHAPTER 2: SURFACING PRIMER

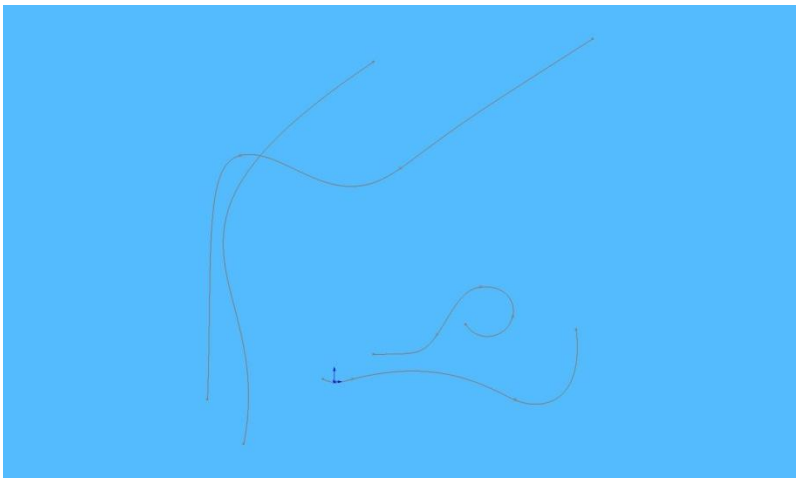
Surface modeling in SolidWorks enables the design of curvilinear, organic shapes for use in the manufacturing of plastic and cast metal parts. In this chapter I learned about the differences between modeling surfaces and solids, and the basic underlying mathematical concepts which control surfaces. Other items of interest were a description of screen display using tessellation and an explanation of surface curvature continuity (including contact, tangent and curvature continuous transitions).

CHAPTER 3: SKETCHING WITH SPLINES

I learned how to use the spline tool to create curvilinear sketch elements. Sketched splines can be constrained using relations to reference geometry and/or dimensions, and are adjusted using handles or control polygons.



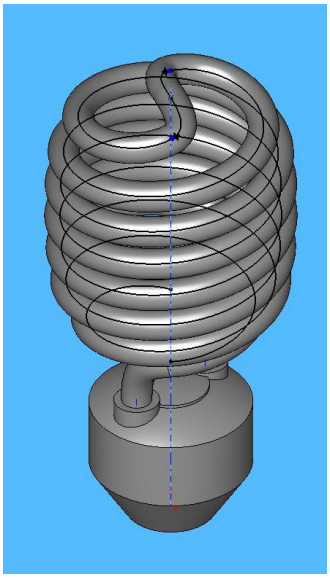
This first exercise involved creating splines with constraints and tight curves.



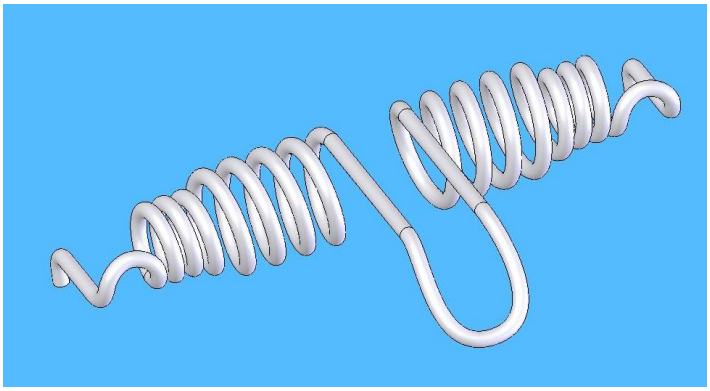
The second “mouse control” exercise required tracing existing shapes using splines.

CHAPTER 4: SKETCHING IN 3D

I learned how to create 3D sketches, which can span multiple planes and are often used as sweep paths. These special sketches can be drawn by plotting points with the mouse. They can also be derived from existing surfaces and/or solids, as in the Intersection Curve.



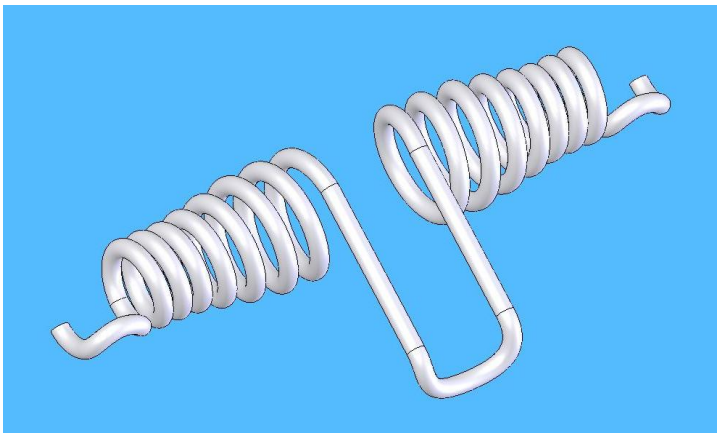
The first exercise utilized the Intersection Curve tool and Spline on Surface tool.



The second exercise utilized the Convert Entities tool.

CHAPTER 5: CREATING CURVES

I learned how create curves including a helix, spiral, projected curve, sketch on sketch (a new sketch made from the intersection of two projected sketches), a curve through reference points and a composite curve. One of the most useful tools I learned about is the Split Line tool, which splits up the faces of a solid so they can be colored differently or manipulated separately.



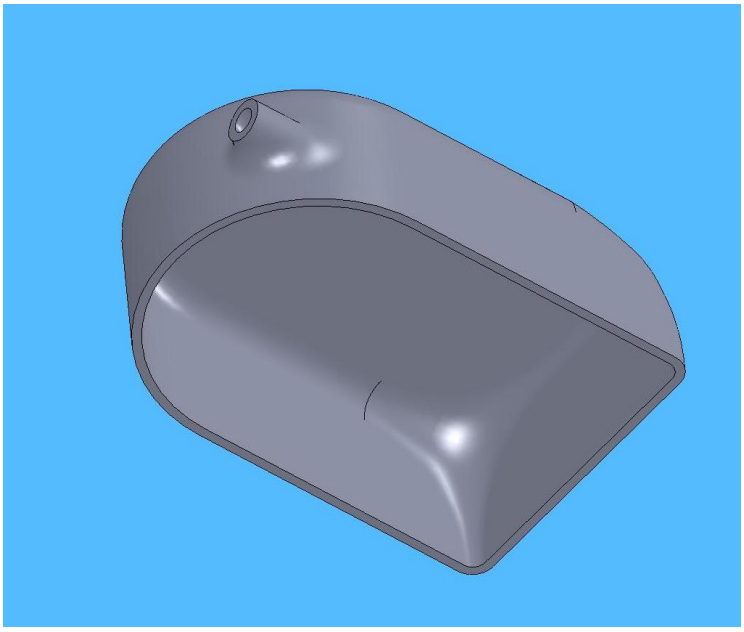
This exercise recreated the preceding spring and involved creating a helix curve.

CHAPTER 6: USING THE PRIMARY SHAPE CREATION FEATURES

In this important chapter I learned the basics of how to create curvilinear forms using the powerful sweep, loft, boundary surface and fill surface features. These four foundational features offer the designer a myriad of possibilities for creating forms. The sweep utilizes a profile and path to make a single-shaped (but possibly variably sized and angled) surface or solid. The loft makes use of a series of profiles, which can be of all different shapes. The boundary surface (the most complex tool of the four) uses sketches and/or edges in two different directions. The fill surface patches up holes and can smooth out intersecting fillets.

CHAPTER 7: USING ADVANCED FILLETS

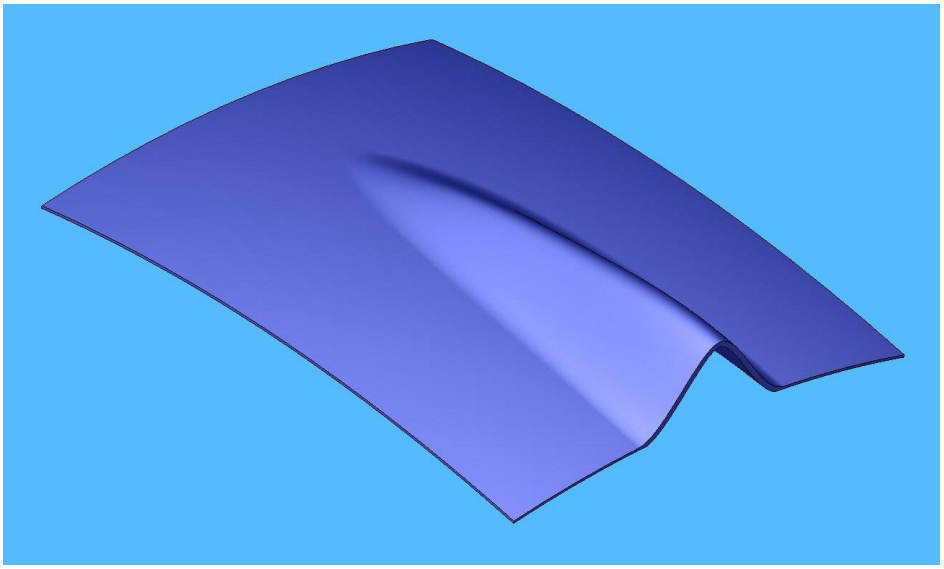
I learned about the different types of fillets, including constant radius, variable radius, face fillets, full round fillets, setback fillets, hold line fillets, curvature continuous fillets and constant width fillets. Each is useful in particular situations for rounding off edges and blending surfaces together smoothly. Fillets are applied to plastic part models for both manufacturing and safety concerns.



This exercise utilized a variety of fillet types to highlight their differences.

CHAPTER 8: SHELLING

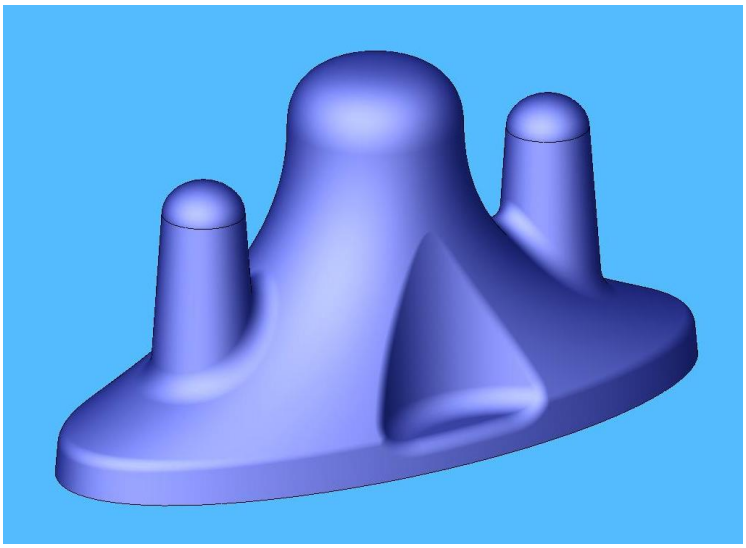
I learned how to apply the shell feature, and also how to manually shell a part using surfaces.



This exercise focused on manual shelling, which can be tedious but necessary.

CHAPTER 9: USING SECONDARY SHAPE CREATION FEATURES

The secondary shape creation features modify existing solids and surfaces that were created by extruding, revolving, sweeping, lofting or using a boundary surface. They are useful for rounding off ends (as the dome and shape tools do), making last-minute changes to parts (the indent tool is especially good for this), creating molds (the radiate surface is useful in this regard), and making reference geometry (the ruled surface is often used to make reference surfaces for tangency settings).



This exercise utilized the ruled surface and dome feature.

CHAPTER 10: WORKING WITH HYBRID FEATURES

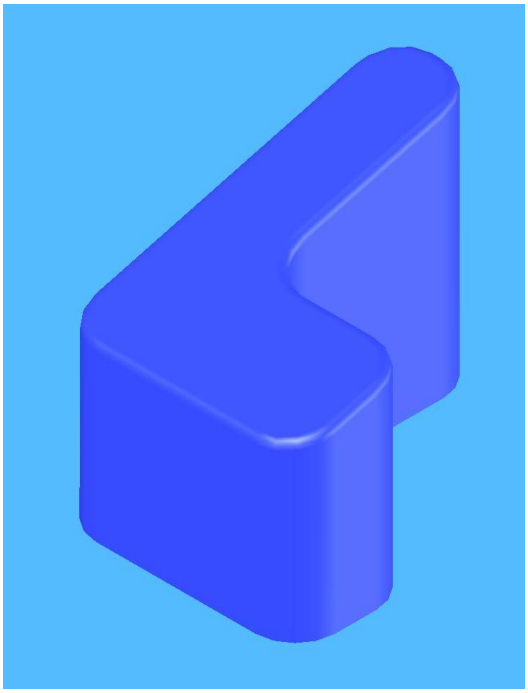
I learned about changing solid models into a series of surfaces, and how to change a series of surfaces into a solid. In a given situation surface tools may be the only way to create the organic form desired, so it is necessary to be able to switch from a solid to surfaces. I also learned different ways to change the size of a part. The thicken feature is useful for changing knit surfaces into solids. The wrap feature is perfect for adding decorative relief to cylindrical or curved faces.

Changing from solids to surfaces was practiced in this exercise.

This exercise was the most challenging in the book, and involved making numerous faces and fillets on a pie-shaped section. A repeated pattern was created by revolving the section.

CHAPTER 11: MANAGING SURFACES

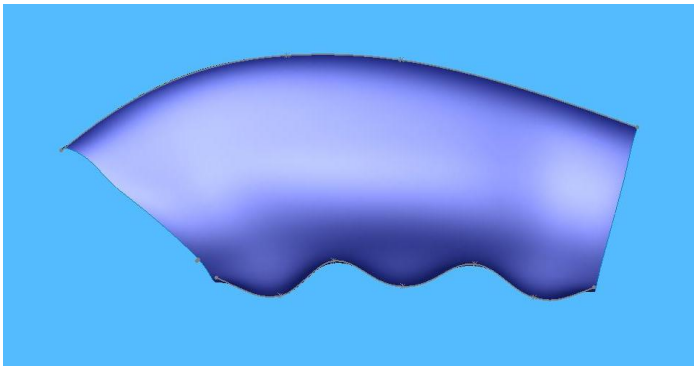
I learned about offsetting surfaces, copying them, and knitting surfaces together. Other helpful tools discussed were the trim surface, untrim surface, delete hole and extend.



This exercise made use of the delete hole function and edited a surface by reversing its angle.

CHAPTER 12: USING DIRECT EDITING TOOLS

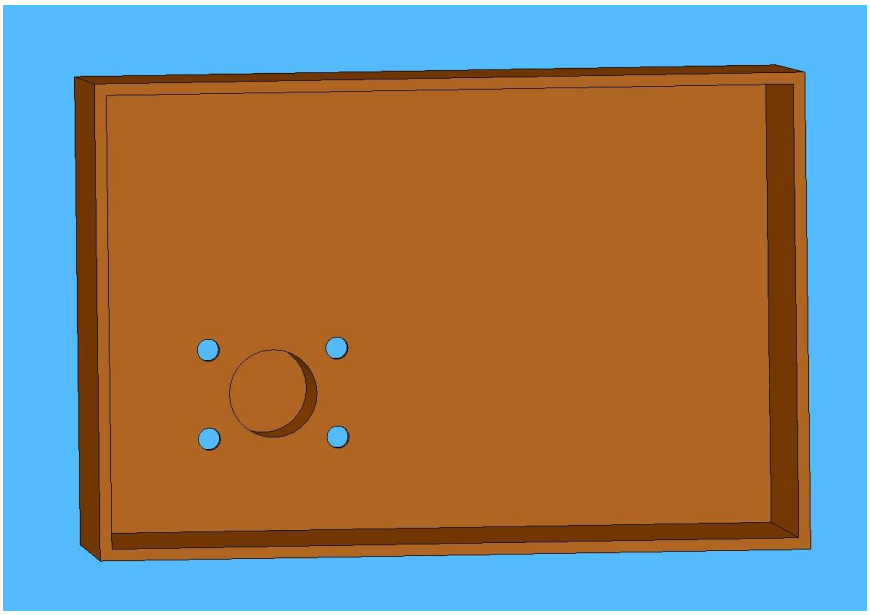
The direct editing tools work well on surfaces which have been imported from other programs into SolidWorks. In this chapter I learned about how to move a face, delete a face, and use the freeform tool. The flex tool can be handy for creating an image for a PowerPoint presentation illustrating how a plastic or metal part may bend. The deform tool allows a surface to be reshaped based on another sketch, point or part.

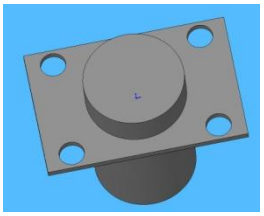


This exercise deformed a surface using sketches.

CHAPTER 13: MANAGING BODIES

I learned how to insert an existing part into another part in order to create matching holes and surfaces. I also learned that imported parts cannot really be deleted, just hidden or accessed through the rollback bar.

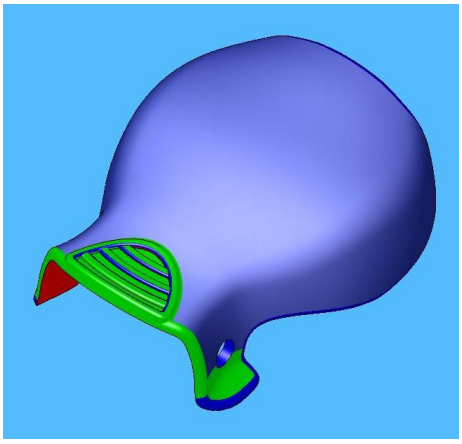




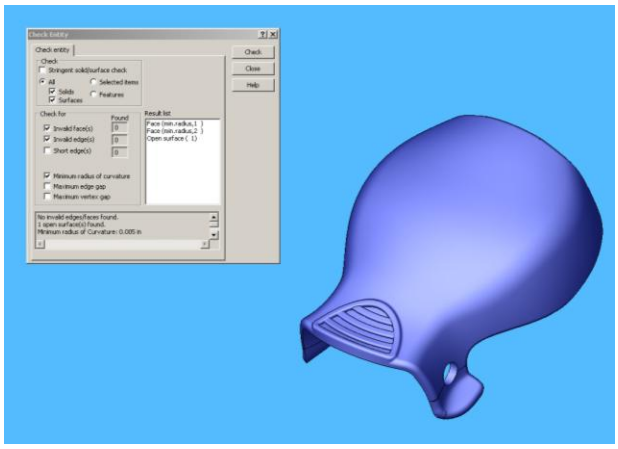
In this exercise the grey part was brought in and used to create an indent in the brown part.

CHAPTER 14: EVALUATING GEOMETRY

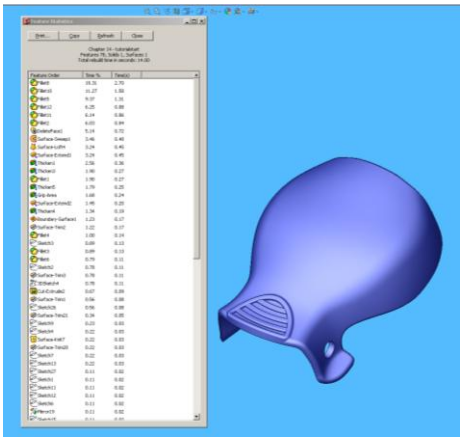
I learned about draft, and how to check that it has been correctly applied to a part that will be manufactured out of plastic or cast metal.



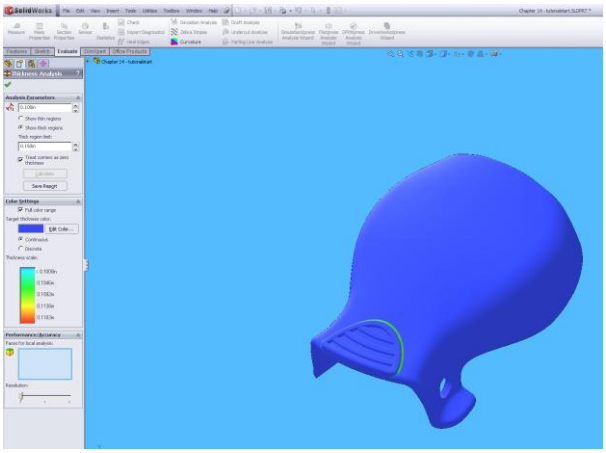
Draft analysis



Evaluation for minimum radius



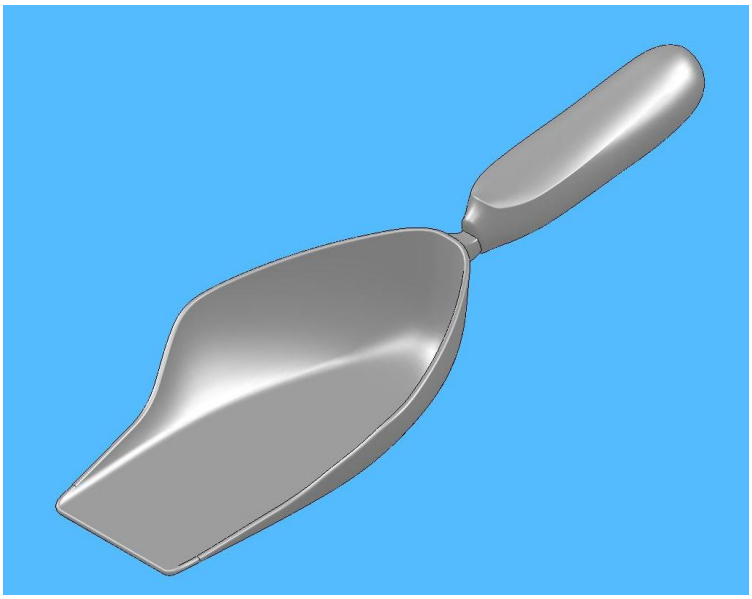
Evaluating rebuild time for features



Thickness analysis

CHAPTER 16: MODELING A TROWEL

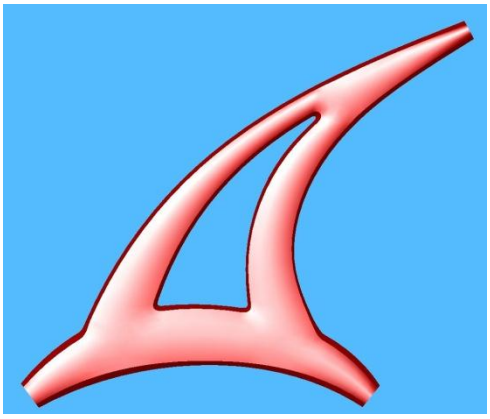
This exercise incorporated a number of surfacing techniques (including using imported sketches, lofting, using the fill feature, extruding a surface, knitting, filleting and trimming) to model a consumer product.



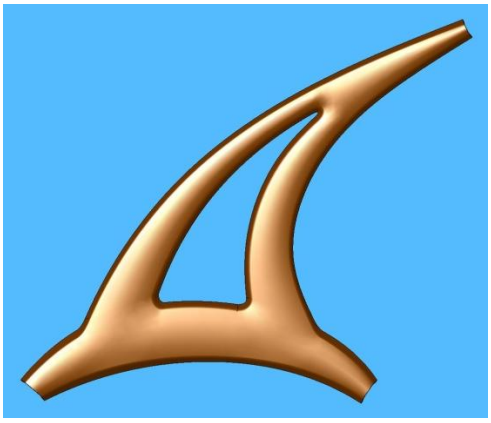
The finished trowel model

CHAPTER 17: MODELING BLENDS

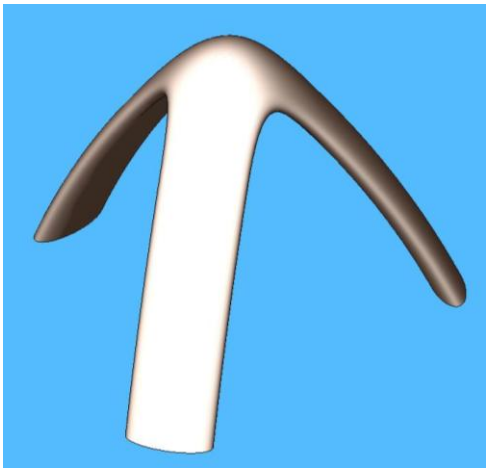
I learned how to make rounded corners using lofted surfaces, and how to make a smooth fill surface.



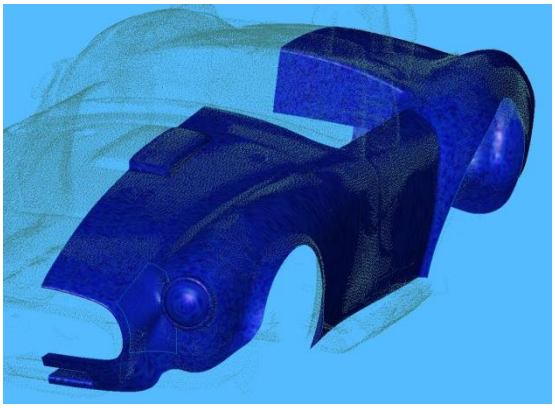
Practicing rounded corners



Replacing a face



Creating a smooth fill surface

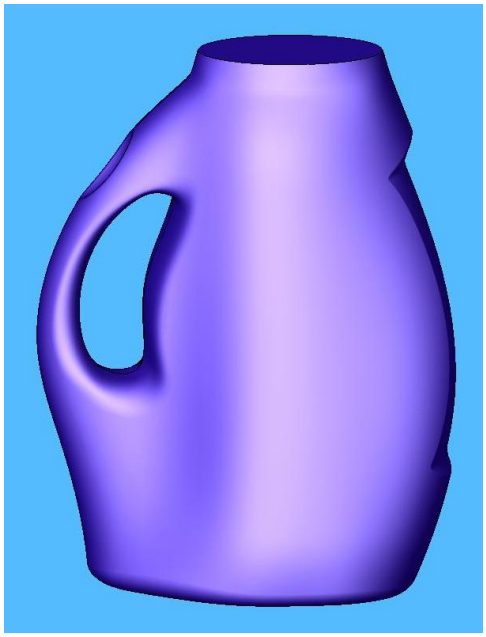


Using fill surface to smooth a complex area

CHAPTER 18: MODELING A PLASTIC BOTTLE

This project involved creating a bilaterally symmetric plastic bottle from artist's sketches using a series of surfaces. The edited and trimmed surfaces were knitted together, then thickened to make a solid. A thread was made on the bottle's neck using a helical curve. Finally, the volume of inside of the bottle was calculated using the Combine/Subtract tool.

The completed bottle



The form used to calculate the volume inside the bottle