SolidWorks TUTORIAL 6

Complex Surface Modeling

Learning Objectives

After completing this tutorial, you will be able to:

• Construct and use surface features in solid modeling
• Incorporate surface features in part design when appropriate
• Construct Swept and Lofted surfaces
• Combine Boolean operations

Required Competencies

Before starting this tutorial, you should have been able to:

• Construct, constraint and dimension sketches
• Project geometry on sketch planes
• Extrude sketched profiles
• Revolve sketched profiles
• Loft sketched profiles
• Understand the concepts of work and placed features
• Understand how to manipulate the history tree in the browser

Figure 1: Toy Car Body
1. Open the file Car Body.SLDPRT. Show all of the sketches for this tutorial. Looks quite complex, but when we analyze the sketches one-by-one it is a much simpler problem.

2. Roll up the browser so that only the Clay Block Sketch is visible. Extrude the sketch as a solid 85mm. Hide the sketch.
3. Roll down the browser to reveal the Front Sketch and the Front Sweep Path. Create a Swept Surface with the sketches. Try alternating the Profile and Path selections and observe what difference the sketch selection makes. Hide the Sketches.

4. Select the Replace Face command from the Surfaces toolbar and click on the end face of the solid. Then click in the Replacement Faces box and select the swept surface. Click the OK checkmark and hide the swept surface.
5. Roll down the browser revealing the Back Sweep and Back Sweep Path Sketches. Sweep the Back sketch along the Path sketch. Replace the extruded solid face with the swept face and hide any visible sketches and the surface.

6. We could have used Insert>Cut>With Surface instead of Replace Face. Which method is better? I don’t know, but if you have trouble with a part try every method. (Problems might not show up until much further down the road – always be prepared to start over.)
7. Show the Left Side and Left Side Sweep sketches and create a Swept Surface. Mirror the Surface Body to the other side of the Front plane.

Figure 8

8. In addition to the Cut With Surface or the Replace Face techniques, we could also use the Insert>Feature>Split command. *(What additional options are available with the Split command?)*

Figure 9
9. Show the Grill Spline, Front Quarter, Rear Quarter and Tail Sketches.

10. Loft a surface across the four splines and then use the Replace Face command to replace the top face of the extruded solid. Hide any visible sketches or surfaces.
11. Show the Canopy and Canopy Sweep Path sketches and create a solid Sweep feature.

12. Create an 80mm fillet feature in the two front corners.
13. Create a 20mm fillet selecting the edges shown.

![Figure 14](image)

**Figure 14**

14. Add a 3mm fillet to the intersecting loop between the body and the canopy.

![Figure 15](image)

**Figure 15**
15. Turn the part over and Shell to a thickness of 1.5mm removing the bottom face.

![Figure 16](image1)

16. Make the Wheel Wells Sketch visible and Extrude-cut through all.

![Figure 17](image2)
17. Make the Mounting Posts sketch visible and Extrude Up To Next the two circles with a Taper of 3°. Mirror to the other side. Save the file.

18. Save As the file with the name Car Body Toolbody.SLDprt. In the new toolbody file delete the Mounting posts, Wheel Well and Shell features.
19. Extrude the Wheel Wells sketch mid-plane 250mm.

Figure 20

20. Start a new sketch on the Top plane and create the sketch shown.

Figure 21
21. Extrude the rectangle 75mm. Save and close the file.

22. Start a new file and insert the Car Body Toolbody and the Car Body parts. (Uncheck the Launch Move Dialog box.)
23. Save the file with the name Car Body Mold Core. From the pull-down menus select Insert>Feature>Combine and subtract the Car Body from the Toolbody. Save and close the file.

Figure 24

24. Start a new file and sketch a rectangle as shown and Extrude a distance of 100mm.

Figure 25

![Figure 26](image)

26. Subtract the toolbody from the extruded block.

![Figure 27](image)
27. Save the file with the name Car Body Mold Cavity.

In this tutorial we learned how to break a complex model into manageable steps. With time and effort some impressive work can be accomplished.