Chevy Malibu

Vehicle Design with Solidworks

Start SolidWorks
Create a New SolidWorks Document
Click: Part and then OK

Now you are ready to make a Part.
Right Toolbar: Click on Grid
Document Properties: Select Units and then highlight inches

Search the web and type the following.
http://www.chevrolet.com
Highlight and copy the drawing from the net
On the Part drawing Paste the document
Feature Manager Design Tree: Highlight Right
Sketch Toolbar: Click Sketch
Standard Views: Click Normal To

Sketch Tools: Click the Rectangle
Sketch Relations: Add Relations, Click on the bottom of the Rectangle and make sure that the bottom line is the only one that is highlighted and then click midpoint.
Using the Sketch tools: Draw the following centerlines and circle.
Add Dimensions as shown.

Using the Sketch tools: Draw the following 3 Point Arc
Make sure that you use the centerline as reference and try to sketch the form of the car.
Using the Sketch tools: Click on Sketch Trim and trim the unwanted lines, make sure that you’ll have a continuous line all around the side profile of the car, as shown in blue.

Using the Sketch tools: Click on Sketch Trim and trim the unwanted lines, make sure that you’ll have a continuous line all around the side profile of the car, as shown in blue. For the tires convert them to Construction Geometry, once done they will turn into centerlines. Click on Fillet and highlight the roof and front window of the car and apply the desired measurements. Do the same thing for the rear.
Now erase all the Geometric Construction Lines or the centerlines, you should be left with something like the one on the picture below. Leave the tire profile because you will be needing this as reference for future reference. Then click on the Rebuild button, the one that looks like a stop light.

Feature Manager Design Tree: Highlight Right
Sketch Toolbar: Click Sketch
Standard Views: Click Normal To
Sketch Tools: Click Convert Entities and select one of the lines on the car, then select Closed Contour.
Features: Click Extruded Boss/Base
Direction 1: Apply the necessary units
Direction 2: Apply the necessary units
Click check.

Feature Manager Design Tree: Highlight Top
Sketch Toolbar: Click Sketch
Standard Views: Click Normal To
Sketch Tools: Draw the following using lines and 3 Point curve and apply dimensions.
Add Relations: Click the curve and the bottom centerline
Click on Tangent and hit enter.
Features: Click Extruded Cut:
  Direction 1: Apply the necessary units
  Draft on/off: Apply the necessary units and check draft outward.

Features: Click Mirror:
  Mirror Face/Plane: Click Right
  Features to Mirror: highlight the Cut Extrude 1
  Click Check
Features:
- Click Filet:
- Fillet Type: Constant Radius
- Items to Filet: highlight the green lines
- Radius: Specify the desired units

This shows the Part after the Filet Command.
Features:

- Click Filet:
- Fillet Type: Constant Radius
- Items to Filet: highlight the green and the pink lines
- Radius: Specify the desired units.
- Click check.

This shows the Part after the Fillet Command.
Features:
- Click Fillet:
- Fillet Type: Constant Radius
- Items to Fillet: highlight the green lines
- Radius: Specify the desired units.

This shows the Part after the Fillet Command.
Plane:
Selections: Click Right Plane, and Select Distance.
Distance: Specify the desired units.

Feature Manager Design Tree: Highlight Plane1
Sketch Toolbar: Click Sketch
Standard Views: Click Normal To
Sketch Tools: Highlight the wheel wells and click Convert Entities, if prompted select Single contour.
Offset Entities: Click the wheel well and specify the desired parameters.
Features: Click Extruded Boss/Base
Direction 1: Up To Surface, and highlight the surface shown on green
Apply the necessary units, Click Check.

Features: Click Mirror:
Mirror Face/Plane: Click Right
Features to Mirror: highlight the Extrude 4 shown on green Click Check
Features:
- Click Fillet:
  - Fillet Type: Constant Radius
  - Items to Fillet: highlight the green and lines
  - Radius: Specify the desired units.

This shows the Part after the Fillet Command.
Feature Manager Design Tree: Highlight Right Plane
Sketch Toolbar: Click Sketch
Standard Views: Click Normal To
Sketch Tools: Highlight the wheel wells and click Convert Entities, if prompted select Single contour. Then draw a line connecting the end points of the arc. Add properties if necessary.

Features: Click Extruded Boss/Base
Direction 1: Apply the necessary units
Click check.
Feature Manager Design Tree: Highlight Right Plane
Sketch Toolbar: Click Sketch
Standard Views: Click Normal To
Sketch Tools: Draw a circle to the center of the tires, add relation if necessary, make them concentric to the tires, then dimension them as shown

Features: Click Extruded Boss/Base
Direction 1: Apply the necessary units
Click check.
Feature Manager Design Tree: Highlight Plane1
Sketch Toolbar: Click Sketch
Standard Views: Click Normal To
Sketch Tools: Draw a circle through the center of the tires, add relation if necessary, make them concentric to the tires, then dimension them as shown

Features: Click Extruded Boss/Base
Direction 1: Apply the necessary units
Click check.
Feature Manager Design Tree: Highlight Plane1
Sketch Toolbar: Click Sketch
Standard Views: Click Normal To
Sketch Tools: Draw circles through the center of the tires, add relation if necessary, make them concentric to the tires, then dimension them as shown.

Features: Click Extruded Boss/Base
Direction 1: Apply the necessary units
Click check.
Features: Click Mirror:
  Mirror Face/Plane: Click Right
  Features to Mirror: highlight the Cut Extrude 3, Extrude9, Extrude8, Extrude6
  Click Check

Features: Click Fillet:
  Fillet Type: Constant Radius
  Items to Fillet: highlight the green lines
  Radius: Specify the desired units.
  Click check.
Finish Product