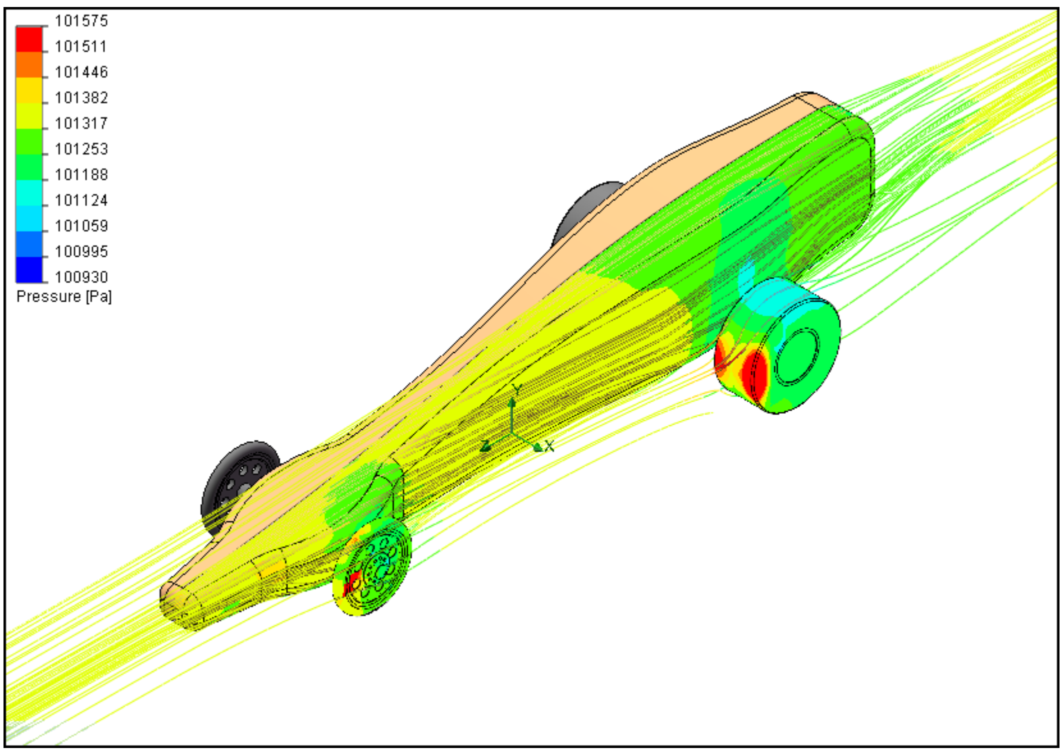
Engineering Design
and Technology Series

CO_2 Car Design Project with SolidWorks® Software



Dassault Systèmes - SolidWorks Corporation
300 Baker Avenue
Concord, Massachusetts 01742 USA
Phone: +1-800-693-9000

Outside the U.S.: +1-978-371-5011
Fax: +1-978-371-7303
Email: info@solidworks.com
Web: http://www.solidworks.com/education

Document Number: PME0619-ENG

Table of Contents

Lesson 1
Introduction

When you complete this lesson, you will be able to:

- Describe the relationship between Parts, Assemblies and Drawings;
- Identify the principal components of the SolidWorks user interface and toolbars;
- Identify the function of each mouse button when using SolidWorks.

Using This Book

The *CO_2 Car Design Project* helps you learn the principles of testing aerodynamic performance using SolidWorks and Flow Simulation as an integral part of a creative and iterative design process.

You will be learning by doing as you complete these phases of the project:

- Establish a baseline, or control car for aerodynamic testing. This car uses the unmodified blank as the body. The aerodynamic performance of the control car will serve as a reference for comparing and assessing the effects of changes to the car body's shape.
- Set up a wind tunnel problem using Flow Simulation.
- Make modifications to the body of the car and rerun the aerodynamic testing.
- Create your own car body design and test it in the virtual wind tunnel.
- Refine the design of your car body based on the results of the aerodynamic testing and retest it.
- Create fully detailed drawings of your car design.
- Create an exploded view assembly drawing complete with a bill of materials.
- Create a photorealistic rendering of your final car design using PhotoWorks.

What is SolidWorks Software?

SolidWorks is design automation software. In SolidWorks, you sketch ideas and experiment with different designs to create 3D models using the easy to learn Windows® graphical user interface.

SolidWorks is used by students, designers, engineers and other professionals to produce simple and complex parts, assemblies and drawings.

Prerequisites

Before you begin the *CO_2 Car Design Project* you should complete the following online tutorials that are integrated in the SolidWorks software:

- Lesson 1 - Parts
- Lesson 2 - Assemblies
- Lesson 3 - Drawings

You can access the online tutorials by clicking **Help**, **Online Tutorial**. The online tutorial resizes the SolidWorks window and runs beside it.

As an alternative, you can complete the following lessons from *An Introduction to Engineering Design With SolidWorks*:

- Lesson 1: Using the Interface
- Lesson 2: Basic Functionality
- Lesson 3: The 40-Minute Running Start
- Lesson 4: Assembly Basics
- Lesson 6: Drawing Basics

Conventions Used in This Book

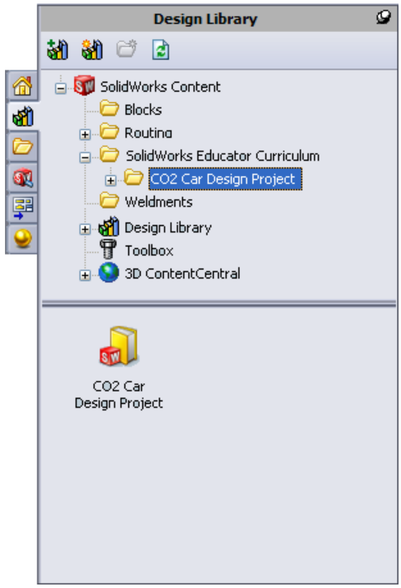
This manual uses the following typographical conventions:

Convention	Meaning
Bold Sans Serif	SolidWorks commands and options appear in this style. For example, **Insert, Boss** means choose the **Boss** option from the **Insert** menu.
`Typewriter`	Feature names and file names appear in this style. For example, `Sketch1`.
17 Do this step.	The steps in the lessons are numbered in sans serif bold.

Before You Begin

If you have not done so already, copy the companion files for the lessons onto your computer before you begin this project.

1 **Start SolidWorks.**
Using the **Start** menu, start the SolidWorks application.



2 SolidWorks Content.

Click **Design Library** to open the Design Library task pane.

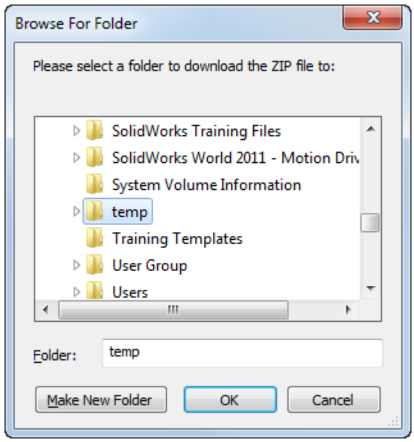
Click on SolidWorks Content to show the folders below it.

Click on SolidWorks Educator Curriculum.

Click CO2 Car Design Project.

Note: There may be more curriculum folders listed in addition to the CO2 Car Design Project.

The lower pane will display an icon representing a Zip file that contains the companion files for this project.



3 Download the Zip file.

Press **Ctrl** and click the icon.

You will be prompted for a folder in which to save the Zip file.

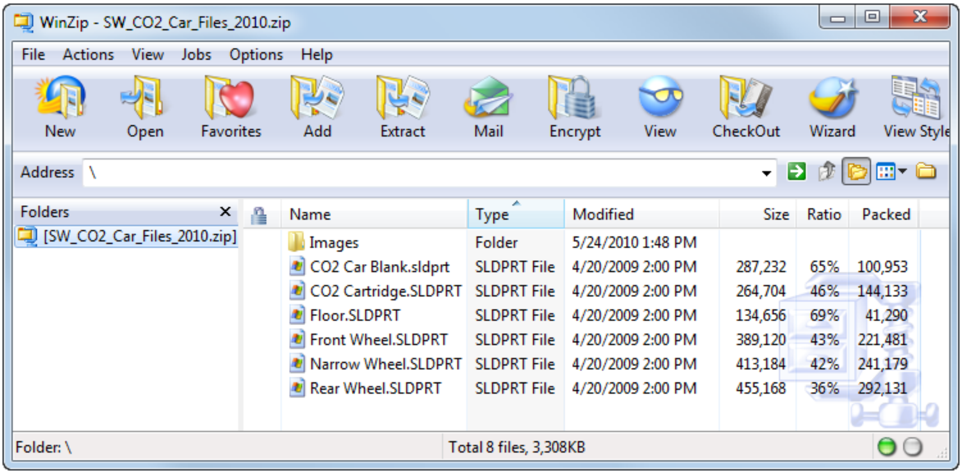
Ask your teacher where you should save the Zip file. Usually the C:\Temp folder is a good location.

Tip: Remember where you saved it.

4 Open the Zip file.

Browse to the folder where you saved the Zip file in step **3**.

Double-click the CO2 Car Design Project.zip file.



5 Click Extract.

Browse to the location where you want to save the files. The system will automatically create a folder named CO2 Car Design Project in whatever location you specify. For example, you might want to save it in My Documents. Check with your teacher about where to save the files.

You now have a folder named CO2 Car Design Project on your disk. The data in this folder will be used in the exercises.

Tip: Remember where you saved it.

Add the Folder to the Design Library Path

The Design Library is a convenient way to access the parts used in the exercises. It is more efficient than using **File, Open** and browsing for a file. All that is necessary is to add the CO2 Car Design Project folder to the search path of the Design Library.

1 Task pane.

Click **Design Library**  to open the Design Library task pane.

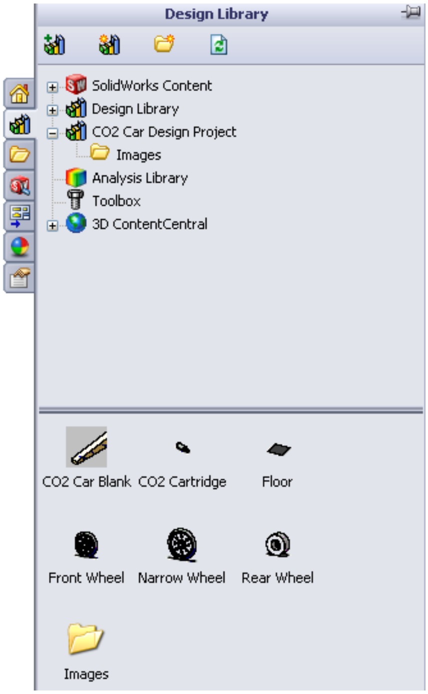
2 Add folder.

Click **Add File Location** .

Browse to where you extracted the companion files in step **4** on page 5.

Select the folder CO2 Car Design Project and click **OK**.

3 Results.
The contents of the CO2 Car Design Project folder will now be accessible through the Design Library task pane.



Lesson 2
Exploring and Assembling the Car

When you complete this lesson, you will be able to:

- Describe three factors important to the performance of a CO_2-powered dragster;
- Calculate the area of a solid's planar face;
- Calculate the volume of a solid, and, when given the density, calculate the mass;
- Create a new assembly;
- Insert components;
- Add mating relationships between components.

What is Important When Designing a Dragster?

Within the framework of the contest specifications, there are three factors to keep in mind when it comes to building a winning dragster. These are:

- **Friction**
  Energy used to overcome friction is energy that isn't being used to accelerate your dragster. Sources of friction include:
  - Wheels and axles: if the wheels do not spin freely, the dragster will be slow.
  - Misaligned axles: if the axle holes are not drilled perpendicular to the centerline of the dragster, the dragster will have a tendency to turn to the left or right. This will cost you speed.
  - Misaligned screw eyes: if the screw eyes are not positioned and aligned properly, the guideline can drag on them, the dragster body, or the wheels. This can slow the dragster dramatically.
  - Bumps or imperfections in the rolling surface of the wheel. The more perfectly round and smooth the wheels are, the better they will roll.
- **Mass**
  Sir Issac Newton's Second Law of Motion states that Force = mass x acceleration. There is a finite amount of thrust (or Force) produced by a CO_2 cartridge. It stands to reason that a car with less mass will accelerate quicker and travel down the track faster. Reducing the mass of your dragster is one way to build a faster car. Keep in mind that the contest specifications may stipulate a minimum mass for the vehicle.
- **Aerodynamics**
  The air exerts a resistance, or drag, as the dragster tries to move through it. To minimize drag, your car should have a smooth, streamlined shape.

Of these three factors – friction, mass, and aerodynamics – we will explore two: mass and aerodynamics, during this project.

About Balsa

Balsa trees grow naturally in the humid rain forests of Central and South America. Its natural range extends south from Guatemala, through Central America, to the north and west coast of South America as far as Bolivia. However, the small country of Ecuador on the western coast of South America, is the world's primary source of balsa for model building.

Balsa needs a warm climate with plenty of rainfall and good drainage. For that reason, the best stands of balsa usually appear on the high ground between tropical rivers. Ecuador has the ideal geography and climate for growing balsa trees.

Balsa wood imported into North America is plantation grown. Don't worry about destroying the rain forests by using balsa – it grows incredibly fast. In 6 to 10 years the tree is ready for harvesting, having reached a height of 18 to 28 meters (60 to 90 feet) and a diameter of about 115 centimeters (45 inches). If left to continue growing, the new wood on the outside layers becomes very hard and the tree begins to rot in the center. Unharvested, a balsa tree may grow to a diameter of 180 centimeters (6 feet) or more, but very little usable lumber can be obtained from a tree of this size.

Use balsa wood with a clear conscience. The rain forests aren't being destroyed when it's harvested.

Start SolidWorks and Open an Existing Part

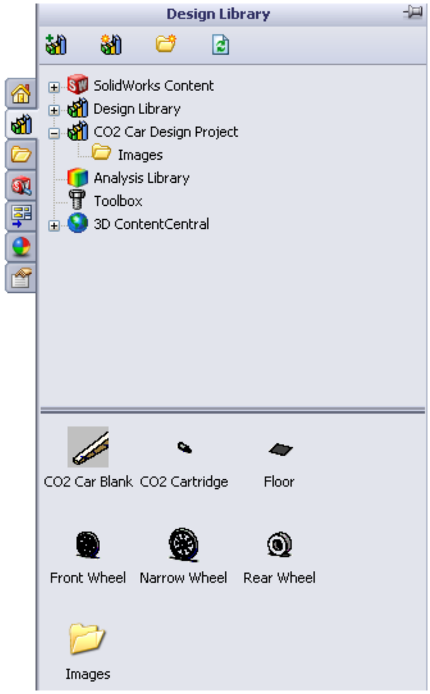
1 **Start the SolidWorks application.**
From the **Start** menu, click **All Programs, SolidWorks, SolidWorks**.

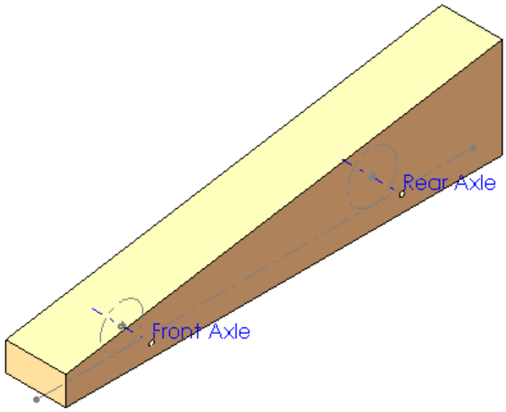
2 **Task pane.**
Click **Design Library** to open the Design Library task pane.

3 Open the `CO2 Car Blank`.
In the **Design Library**, click on the folder `CO2 Car Design Project`.

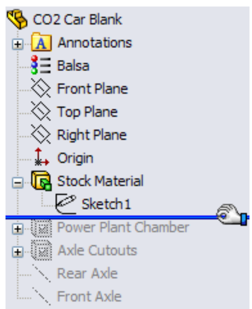
The contents of the folder appear in the lower portion of the **Design Library** window.

Drag and drop the part named `CO2 Car Blank` into the graphics area of the SolidWorks window.



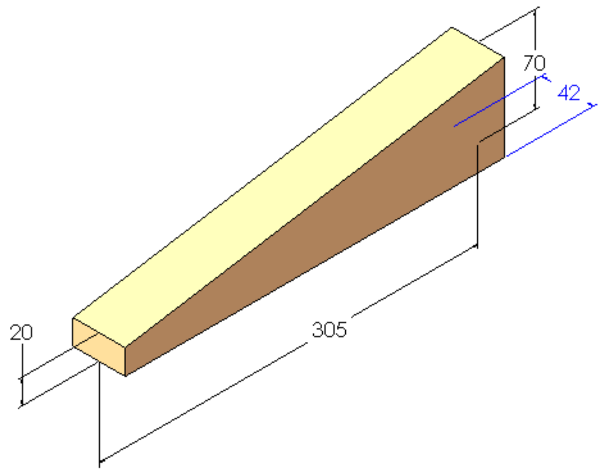


4 Rollback the FeatureManager design tree.
Drag the rollback bar upwards to a position right before (above) the `Power Plant Chamber` feature.



5 Dimensions of the blank.

Right-click the `Annotations` folder and select **Show Feature Dimensions** from the shortcut menu.



6 Turn off the dimension display.

Select **Show Feature Dimensions** again to turn off the dimension display.

Determining the Mass of the Blank

Mass is the amount of matter an object has. Calculating the mass is done by multiplying the density of the material times the volume of the object. So, to determine the mass of the car body blank, we need to know two things:

- The density of the material, in this case, balsa wood
- The volume of the blank

Density

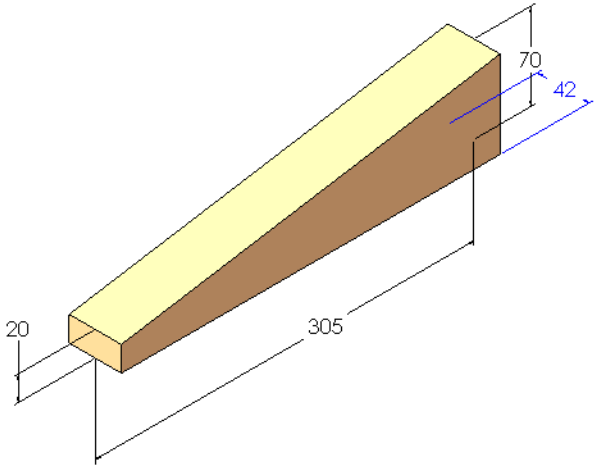
The density of balsa wood ranges from 100 kg/m^3 to 300 kg/m^3 depending on a number of factors including where the tree was grown, how old the tree was when it was cut, what part of the tree the wood was cut from, and how dry the wood is.

Typical, medium-density balsa wood is in the range of 140 kg/m^3 to 192 kg/m^3.

In this example, we use a density of 160 kg/m^3.

Calculating Volume

The car body blank is a trapezoidal prism. To calculate the volume, multiply the area of the trapezoid by the depth of the prism.

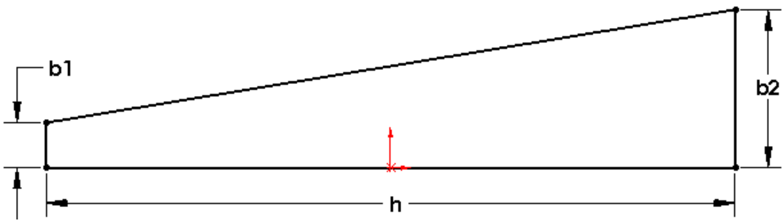


Area of a Trapezoid

A trapezoid is a quadrilateral (a 4-sided figure) with exactly one pair of parallel sides.

To determine the area of a trapezoid:

1. Add the lengths of the 2 parallel sides.
2. Divide the sum by 2 to get the average length of the parallel sides.
3. Multiply this by the distance between the parallel sides.



This can be written as: $area = h \times \left(\frac{b1 + b2}{2}\right)$

where $b1$ and $b2$ are the two parallel sides and h is the distance between them.

Substituting the dimensions of the car body blank, we get:

$$area = 305mm \times \left(\frac{20mm + 70mm}{2}\right) = 305mm \times 45mm = 13{,}725mm^2$$

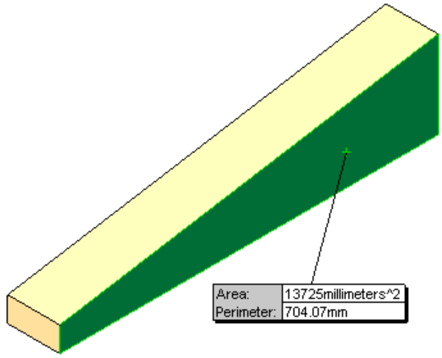
Checking Our Math

SolidWorks has tools for measuring values such as length and area as well as for calculating volume and mass.

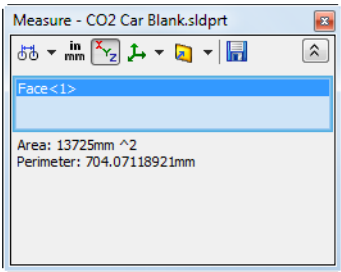
7 Measure.
Click **Tools, Measure**, or click **Measure** on the Tools toolbar.

Select the trapezoidal face of the car body blank.

The area is given as 13,725 mm^2, which matches our calculation.



8 Close the dialog box.
Right-click in the graphics area and select **Close Measure Dialog** from the shortcut menu. Or, click the “x” in the upper right corner of the dialog box.



Volume of a Trapezoidal Prism

To calculate the volume, multiply the area of the trapezoidal face, 13,725 mm^2, by the depth, 42 mm, which we saw in step **5**. Thus:

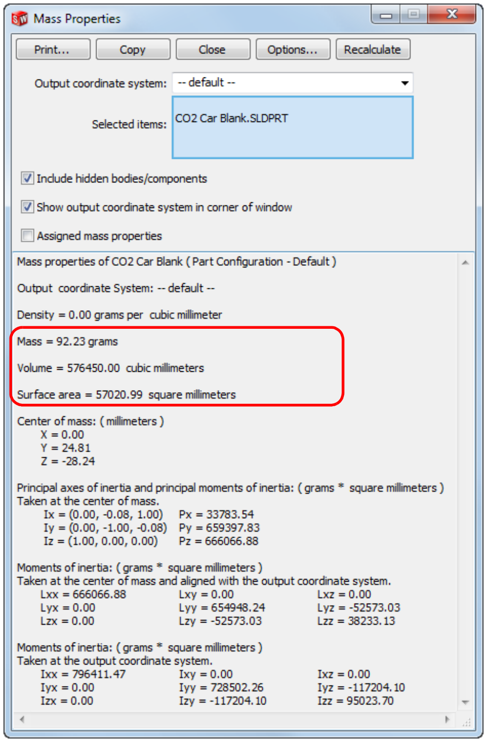
$$13{,}725mm^2 \times 42mm = 576{,}450mm^3$$

9 SolidWorks mass properties calculations.
Click **Tools, Mass Properties**, or click **Mass Properties** on the Tools toolbar.

A window appears giving all sorts of information about the part, including its volume.

Notice the volume equals 576,450 cubic millimeters. This matches our calculations.

The mass is 92.23 grams.



Why is the Density 0.00?

By default, the precision of the units in this part are set to two decimal places. That means a density of 0.00016 is displayed as 0.00. This does not effect the accuracy of the calculations, only the display of the results.

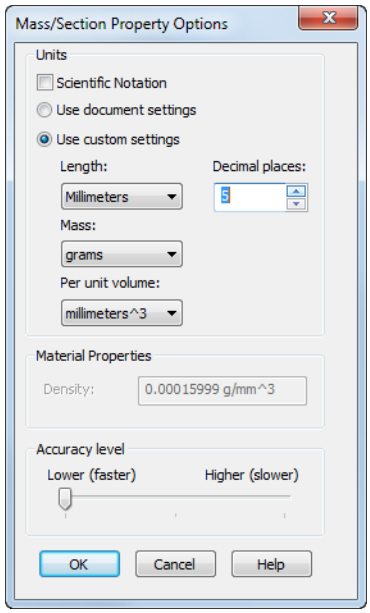
The number of decimal places in the report can be changed.

10 Options.
In the **Mass Properties** report window, click **Options**.

Click **Use custom settings**.

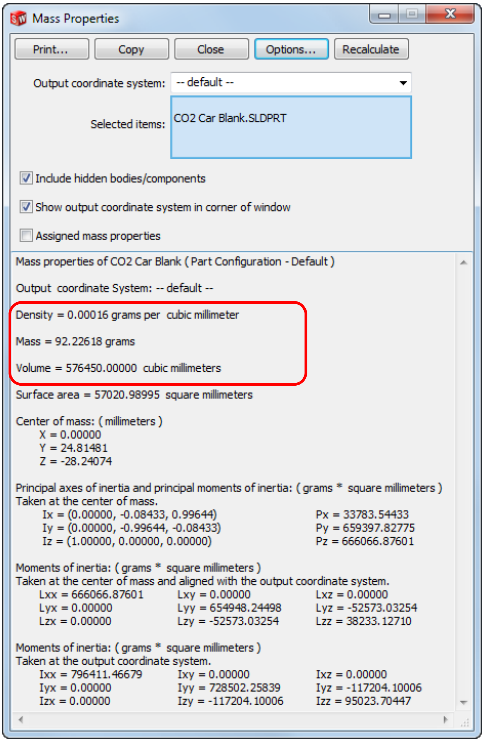
Under **Decimal places**, change the number to **5**.

Click **OK**.



11 Updated results.

The values in the report window are automatically updated to display with five decimal places.



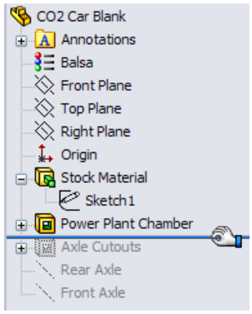
12 Close.

Click **Close** to close the report window.

What About the Hole for the CO_2 Cartridge?

The car body blank used when writing this book has a hole predrilled for the CO_2 cartridge. To accurately determine the volume and mass of the blank, this has to taken into account. That is, the volume of the hole has to be subtracted from the volume of the blank.

Rather than deal with more complicated mathematics, we will use SolidWorks to do the calculation.



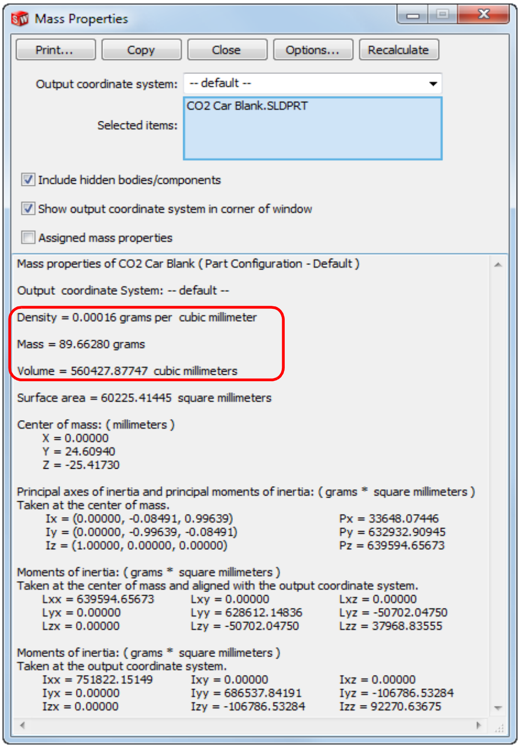
13 Roll forward.

Drag the rollback bar downward to the position right below the feature named `Power Plant Chamber`.

14 Repeat the mass property calculations.

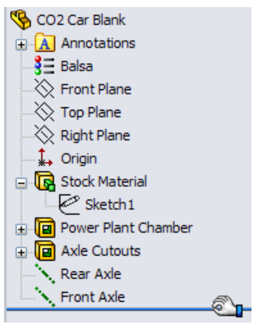
Click **Tools, Mass Properties**, or click **Mass Properties** on the Tools toolbar.

The report window now indicates a mass of 89.66280 grams and a volume of 560,427.87747 cubic millimeters.



15 Roll forward.

Drag the rollback bar downward to the bottom of the FeatureManager design tree.



More to Explore

What is the density of *your* balsa wood blank? To find out, follow this procedure:

1. Using a scale that is accurate to ±0.1 grams, measure the mass of your car body blank. Write the value here:________________________________
2. Divide the mass by the volume of 560,427.88 cubic millimeters that we obtained in step **14**. Remember: $Density = \frac{Mass}{Volume}$
3. Write the value here:___________________________

Converting Units

A quantity is made of two parts: the *magnitude* and the *units*. For example, if you measure the length of something to be 42 mm long, then the magnitude is *42* and the unit is *mm*.

The same quantity can be expressed in different ways. For example, the same length can be stated as 42 mm or 1.65 inches. The magnitude and units change but the quantity, that is the length, is the same. Quantities that represent the same thing are said to be *equivalent*. This is typically indicated by the = sign.

The key to converting units is to understand that you are not changing the quantity – only changing the way it is expressed. If you had 10 pennies and you gave them to someone in exchange for a dime, you would still have the same amount of money. Just because you went from having 10 things called pennies to having one thing called a dime doesn't mean you have less money. You just have fewer coins.

Let's look at the math: $10\ pennies = 1\ dime$

Note: We use the = sign because the quantities are equivalent.

Divide both sides of the equation by 1 dime: $\frac{10\ pennies}{1\ dime} = \frac{1\ dime}{1\ dime} = 1$

By this process we have progressed from an equivalent, $10\ pennies = 1\ dime$, to a conversion factor, $\frac{10\ pennies}{1\ dime} = 1$

Tip: The best way to write a conversion factor is to have the number 1 on one side of the equals sign.

This demonstrates that the conversion factor between pennies and dimes is just a specialized way of expressing the number 1.

This leads to two important realizations:

- The key to converting units is to multiply by the right form of the number 1.
- When you multiply something by 1 you do not change its value.

Another Example

Suppose we want to convert 60 miles/hour to feet/second. First list the equivalents that you know (or can look up):

- 1 mile = 5,280 feet;
- 1 hour = 60 minutes;
- 1 minute = 60 seconds.

Tip: Remember that any of these can be written in reverse order. For example, 5,280 feet = 1 mile.

Now write them as conversions factors, fractions that equal 1. These are the specialized forms of the number 1 that we can use.

- $\frac{5{,}280\ feet}{1\ mile} = 1$
- $\frac{1\ hour}{60\ minutes} = 1$
- $\frac{1\ minute}{60\ seconds} = 1$

Now write the equation where we multiply by various specialized forms of the number 1.

$$\frac{60\ miles}{1\ hour} \times \frac{1\ hour}{60\ minutes} \times \frac{1\ minute}{60\ seconds} \times \frac{5{,}280\ feet}{1\ mile}$$

Tip: Write the conversion factors so the units cancel out. Since we started with miles per hour, hours is in the denominator. That means the next conversion factor should have hours in the numerator. This in turn determines the arrangement of the next conversion factor, and so on.

Cancel out the units and two of the 60 values:

$$\frac{\cancel{60}\ \cancel{miles}}{\cancel{1\ hour}} \times \frac{\cancel{1\ hour}}{\cancel{60}\ \cancel{minutes}} \times \frac{1\ \cancel{minute}}{60\ seconds} \times \frac{5{,}280\ feet}{1\ \cancel{mile}}$$

This leaves us with feet per second, which is what we want:

$$\frac{5{,}280\ feet}{60\ seconds} = 88\ feet\ per\ second$$

So How Does This Apply to Density?

Convert the density you calculated for *your* balsa wood blank from g/mm^3 to kg/m^3. You will need to use some or all of these conversion factors:

- 1 kilogram = 1,000 grams;
- 1 meter = 1,000 millimeters;
- 1 cubic millimeter = 1 millimeter x 1 millimeter x 1 millimeter;
- 1 cubic meter = 1 meter x 1 meter x 1 meter;
- 1 cubic meter = 1,000 millimeters x 1,000 millimeters x 1,000 millimeters;

For illustration purposes, we will use the density 0.00016 grams/mm^3 that we obtained in step **11** on page 15.

Write the equation where we multiply by various specialized forms of the number 1.

$$\frac{0.00016\ grams}{mm^3} \times \frac{1\ kg}{1{,}000\ grams} \times \frac{1{,}000^3 mm^3}{m^3}$$

Cancel out the units: $\frac{0.00016\ \cancel{grams}}{\cancel{mm^3}} \times \frac{1\ kg}{1{,}000\ \cancel{grams}} \times \frac{1{,}000^3\ \cancel{mm^3}}{m^3}$

This leaves us with: $\frac{0.00016\ kg \times 1{,}000^3}{1{,}000\ m^3}$

Dividing out 1,000 from the numerator and denominator give us:

$\frac{0.00016\ kg \times 1{,}000^2}{m^3}$ which is $\frac{160\ kg}{m^3}$

You Do It

Take the density you calculated for your balsa wood blank (see page 17) and using the procedure above, convert it from grams/mm^3 to kilograms/m^3.

Write your answer here:_________________________________

Summary

- When you multiply something by 1 you do not change its value.
- Conversion factors are equivalents, written as fractions that equal 1.
- The key to converting units is to multiply by the right form of the number 1.
- Sometimes you have to multiply by several different forms of the number 1.

Creating an Assembly

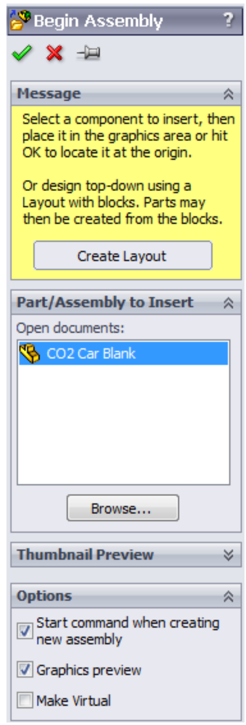
1 Create an assembly.
Click **Make Assembly from Part/Assembly** on the Standard toolbar.

2 Insert component.
The **Insert Component** PropertyManager automatically appears.

The `CO2 Car Blank` part file is listed in the **Open documents** list.

Be sure the **Graphics preview** option is selected.

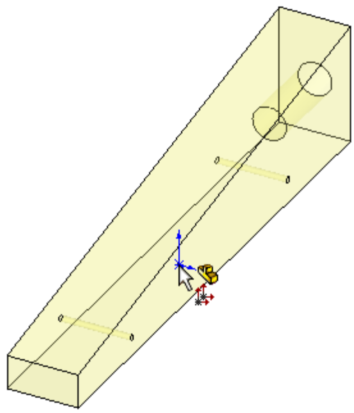
Select the `CO2 Car Blank.sldprt` part file.



3 Show origin.
Click **View, Origins** to turn on the display of the origins.

4 Locate component.
Move the cursor onto the origin and place the component at the origin by placing the cursor over the origin symbol. The double arrow symbol appears when the cursor is snapping to the origin.

The part will appear in the assembly FeatureManager design tree as `Fixed(f)`.

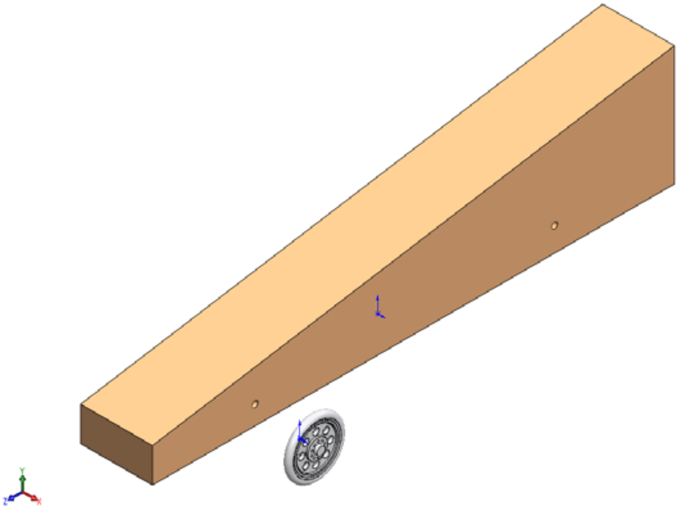


Note: The initial component added to the assembly is **Fixed** by default. Once you have inserted a fixed component into position in an assembly it cannot be moved unless you float it.

5 Isometric view.
Click **Isometric** on the Standard Views toolbar.

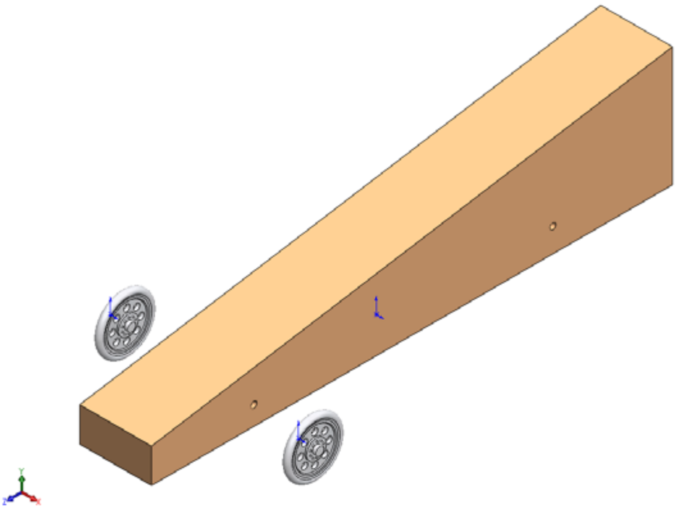
6 Save the assembly.
Save the assembly under the name `CO2 Car Baseline` in the `CO2 Car Project` folder.

7 Add the front wheels.
Drag and drop the `Front Wheel` from the Design Library window into the assembly window.



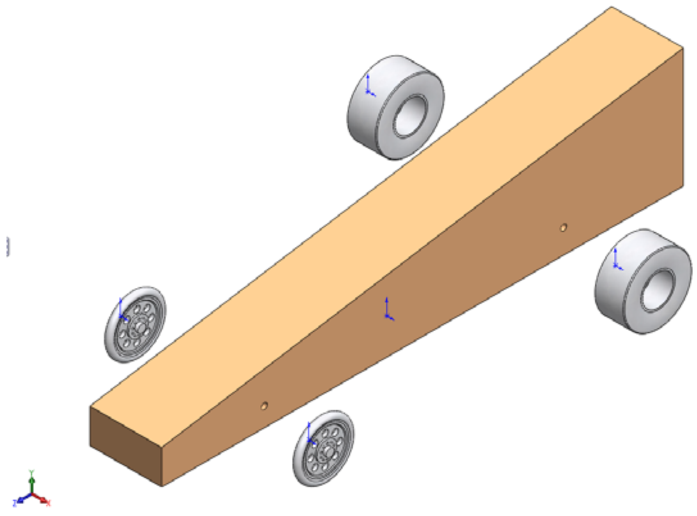
Do not click **OK** yet. Continue with the next step.

8 Add another front wheel.
Click in the graphics area to add a second wheel.



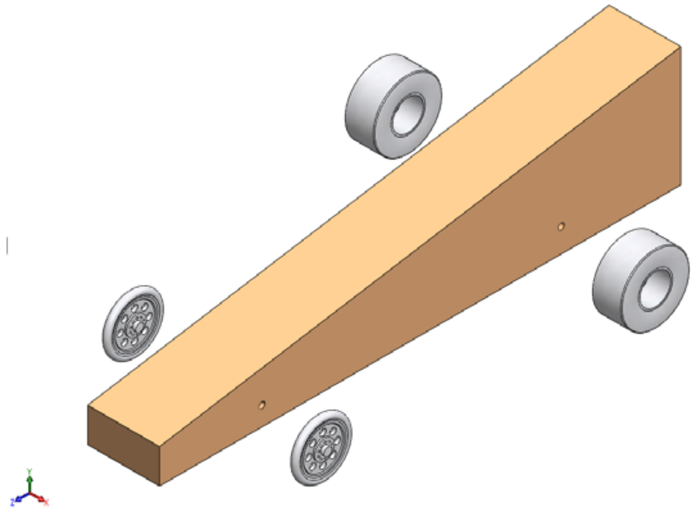
9 Add the rear wheels.

Drag and drop two copies of the `Rear Wheel` from the Design Library window into the assembly window.



10 Turn off the origins.

Click **View**, **Origins** to toggle off the display of the origins.



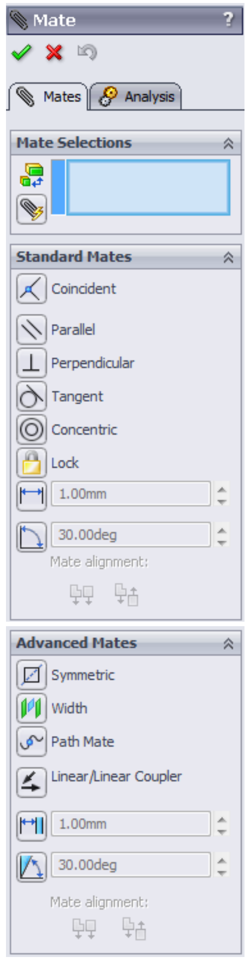
Insert Mate

An assembly is a document in which two or more parts and other assemblies (sub-assemblies) are mated together. Parts and sub-assemblies are called *components* in an assembly. Mates are used to create relationships between components. Faces are the most commonly used geometry in mates. In this case the existing sub-assemblies are mated to build an assembly based on the car part you created.

There are three types of mates, the **Standard Mates**, the **Advanced Mates** and the **Mechanical Mates**.

Standard Mates

- Coincident
- Parallel
- Perpendicular
- Tangent
- Concentric
- Distance
- Angle

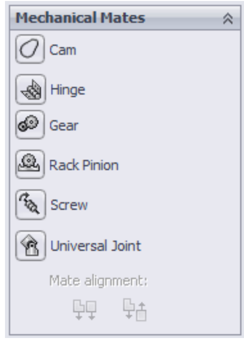


Advanced Mates

- Symmetric
- Width
- Path Mate
- Linear/Linear Coupler
- Distance/Angle Limit

Mechanical Mates

- Cam
- Hinge
- Gear
- Rack Pinion
- Screw
- Universal Joint



You can select many different types of geometry to create a mate:

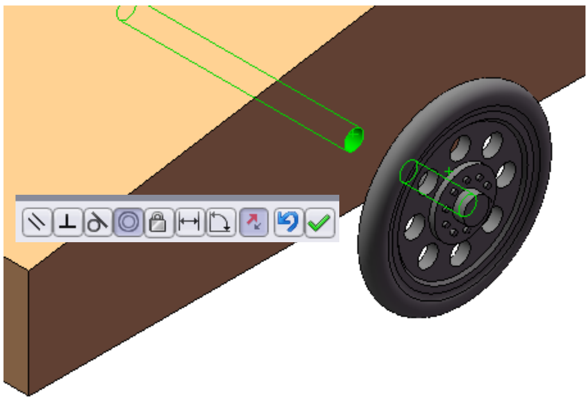
- Faces
- Planes
- Edges
- Vertices
- Sketch lines and points
- Axes and Origins

11 Mate one of the front wheels to the body of the car.

Click **Insert**, **Mate…**, or click the **Mate** tool on the Assembly toolbar.

12 Selections and preview.

Select the cylindrical faces of the axle hole in the body and the hole in the wheel as indicated.



Tip: Zoom and/or rotate the view to make it easier to select the faces you want to mate.

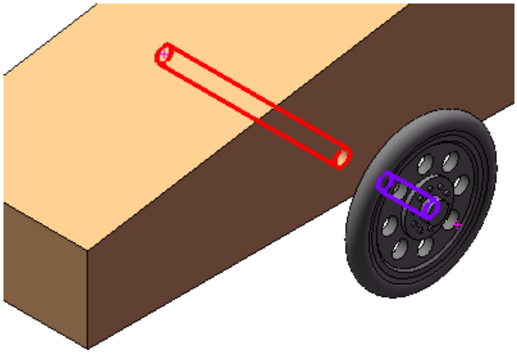
The Mate pop-up toolbar appears to make selections easier by displaying the available mate types right in the graphics area. The mate types that are available vary by geometry selection and are the same as those that appear in the PropertyManager. Either the on-screen toolbar or PropertyManager can be used.

Concentric is selected as the default and the results of the mate are previewed.

Click **Add/Finish Mate** to accept the **Concentric** mate.

Tip: When the mate is previewed, the mouse pointer changes to look like this: . Clicking the right mouse button enters **OK** and applies the mate.

Planning Ahead



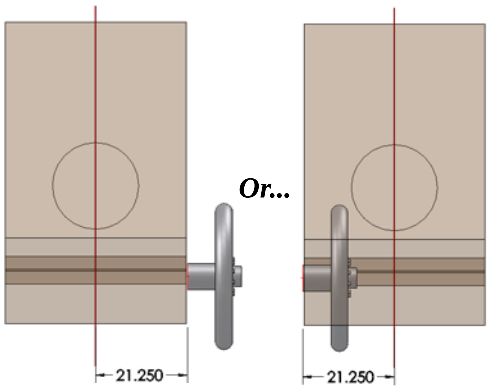
Now that we have the wheel mated so it is concentric with the axle hole, we have to control its position along the axis of the hole. In other words, how far out from the centerline of the car body is the wheel?

At first thought it might make sense to add a **Coincident** mate between the side of the car body and the inner face of the hub of the wheel.

However, this is not the best strategy. A **Coincident** mate requires two planes, either faces or reference planes. If we modify the shape of the car body so that the side is no longer planar, then the **Coincident** mate will fail.

A better approach is to plan ahead and use a type of mate that will define the location of the wheel regardless of what happens to the shape of the car body. The best mate to use in this situation is a **Distance** mate.

Distance Mates



Distance mates are a bit more complicated than some other types of mates. The reason for this is when you specify a distance between two objects, there are two solutions.

The illustrations at the right show a front view of the car body and wheel.

In the leftmost picture, the wheel is positioned 21.25mm to the *right* of the `Right` reference plane.

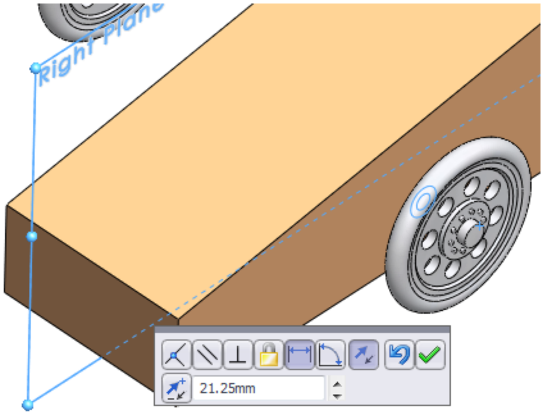
In the rightmost picture, the wheel is positioned 21.25mm to the *left* of the `Right` reference plane.

Both solutions are technically correct. However, only one is the solution we want.

When you add a **Distance** mate, pay attention to the preview in the graphics window. If the previewed solution is not the one you want, click **Flip Dimension** on the Mate pop-up toolbar.

13 Add a Distance mate.

The **Mate** PropertyManager stays active so you can continue adding mates without having to restart the command.



In the FeatureManager design tree, expand the features of the `CO2 Car Blank` and select the `Right` reference plane.

Next select the planar face on the hub of the wheel.

The system assumes you want a **Coincident** mate, which unfortunately is not what we want.

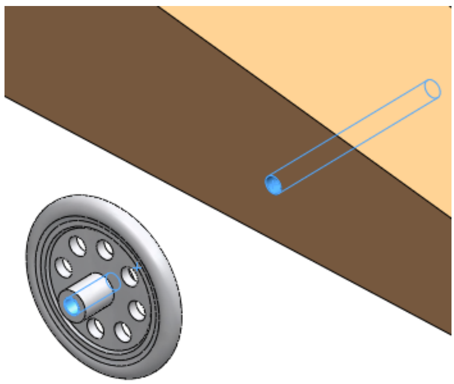
Click **Distance** on the Mate pop-up toolbar and enter a distance value of **21.25 mm**.

Look at the preview to make sure the solution is correct.

Click **Add/Finish Mate** to accept the mate.

14 Mate the other front wheel.

Select the cylindrical faces of the axle hoes in the body and wheel like you did in step **12**.



The system assumes you want a **Concentric** mate, which in this case is correct.

However, the wheel is oriented the wrong way. This is because **Concentric** mates also have two solutions. By default, the system gives you the solution closest to the way the parts are already oriented. There are two ways to address this problem:

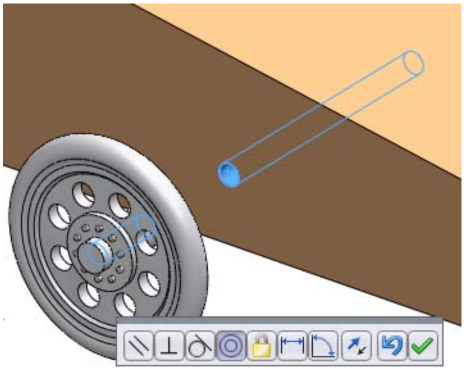
1. Click **Flip Mate Alignment** on the Mate pop-up toolbar.
2. Or, rotate the wheel around so it is more or less in the correct orientation *before* you add the mate.

15 Flip mate alignment.

Since we have already started adding the **Concentric** mate, click **Flip Mate Alignment** on the Mate pop-up toolbar.

The wheel rotates so it is aligned correctly.

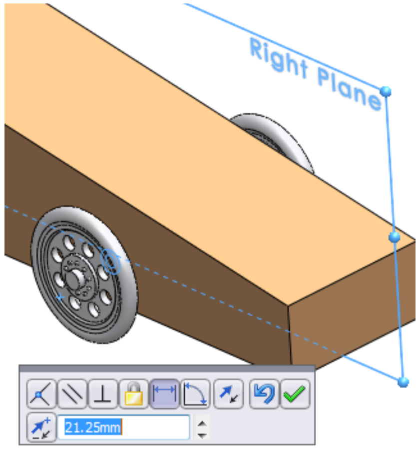
Click **Add/Finish Mate** to accept the mate.



16 Add a Distance mate.

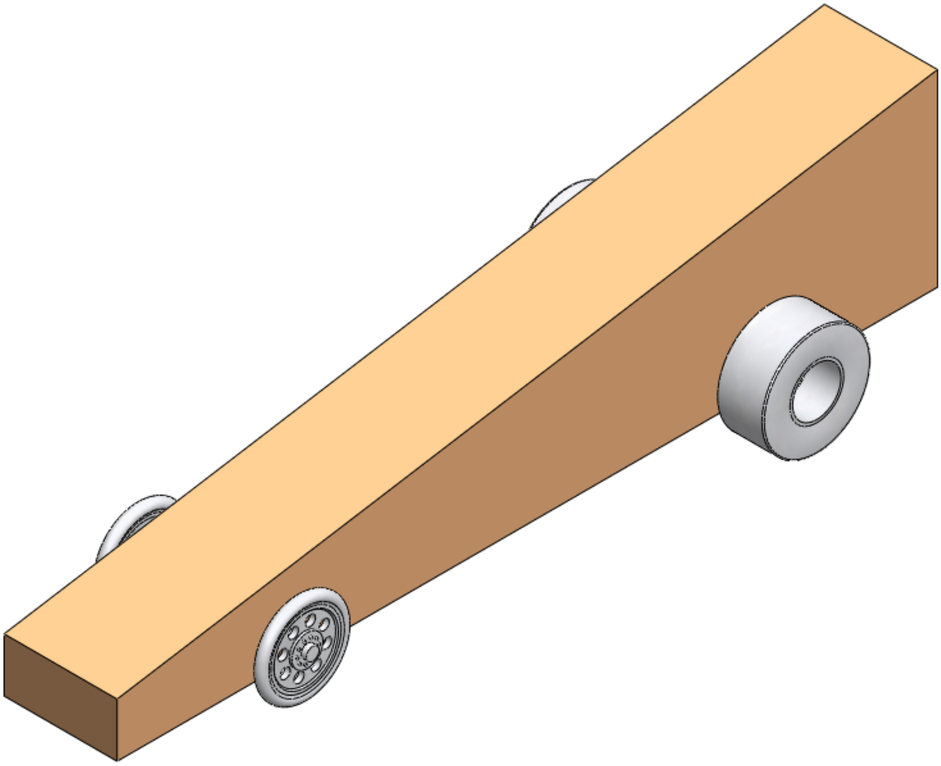
Following the same procedure as in step **13**, mate the wheel to the `Right` reference plane of the car body using a **Distance** mate.

Because this wheel is located on the opposite side of the reference plane you should pay particular attention to the graphic preview. You may have to click **Flip Dimension** on the Mate pop-up toolbar to get the correct result.



17 Mate the rear wheels.

Repeat step **12** through step **16** mating the rear wheels to the body of the car using **Concentric** and **Distance** mates.



18 Mass calculation.

Now that we have added the wheels to the assembly, what is the total mass?

Click **Tools, Mass Properties**, or click **Mass Properties**  on the Tools toolbar.

The total mass is 107.29 grams.

Is this correct?

Not really. This assembly doesn't contain any axles. We could increase the accuracy by modeling the steel axles and adding them to the assembly, but that is beyond the scope of this lesson. The point is, the accuracy of the calculations is no better than the accuracy of the models we create.

19 Save the file.

Turn off the **RealView Graphics** tool , if the RealView Graphics mode is on.

Save the assembly file.

Lesson 3
Analyzing the Car Using SolidWorks Flow Simulation

When you complete this lesson, you will be able to:

- Describe SolidWorks Flow Simulation;
- Describe a fluid flow analysis;
- Load the SolidWorks Flow Simulation add-in;
- Create a SolidWorks Flow Simulation project;
- Run an analysis on the car assembly;
- View the results.

The Aerodynamics Analysis of the Car

During this lesson, you will use SolidWorks Flow Simulation to analyze the aerodynamics of the car. Think of SolidWorks Flow Simulation as a virtual wind tunnel.

What is SolidWorks Flow Simulation?

SolidWorks Flow Simulation is the only fluid flow analysis tool for designers that is fully embedded inside SolidWorks. With this software you can analyze the solid model directly. You can also easily set up units, fluid type and fluid substances and more by using the wizard. The change operation to an analysis result is possible within the same GUI as the FeatureManager design tree in SolidWorks.

There are several steps to the analysis:

1. Create a design in SolidWorks.
   SolidWorks Flow Simulation can analyze parts, assemblies, subassemblies and multibodies.
2. Create a project file in SolidWorks Flow Simulation.
   SolidWorks Flow Simulation projects will contain all the settings and results of a problem and each project that is associated with a SolidWorks configuration.
3. Run the analysis. This is sometimes called solving.
4. Viewing the Flow Simulation results which include:
   Results Plots:
   - Vectors, Contours, Isolines
   - Cut Plots, Surface, Flow Trajectories, Isosurfaces
   Processed Results:
   - XY Plots (Microsoft Excel)
   - Goals (Microsoft Excel)
   - Surface Parameters
   - Point Parameters
   - Reports (Microsoft Word)
   - Reference Fluid Temperatures

Fluid Flow Analysis

Fluid Flow Analysis is used to dynamically study the action of *liquids* such as water and oil, or *gases* such as hydrogen and oxygen. The simulation of a weather report, tsunami information or auto traffic are phenomena of fluid flow analysis.

The benefits of fluid flow analysis are *Energy Conservation* and *Heat Transfer*.

Energy Conservation: The overall stress load of an engine can be lessened by analyzing its structure and weight, while a fluid flow analysis can gather combustion efficiency data to improve the power output.

Heat Transfer refers to the physics of the exchange of energy in the form of temperature. For example, in a nuclear reactor, the radioactive degradation does not directly produce electrical energy. It is the heat energy which is transmitted into water to produce steam which drives the turbines to produce electricity.

Fluid flow analysis is used in many fields of the manufacturing industry:

- **Aerodynamic design and machine**
  Fans and power generating windmills
- **Cooling and heating**
  Predicting the potency of a temperature transfer
- **Fluid centered machines**
  Pumps, compressors, and valves
- **Electrical devices**
  Personal computers and exothermic measurements of precise electrical devices
- **Transport machinery**
  Cars, ships and airplanes (engines are another)

Why Do Design Analysis?

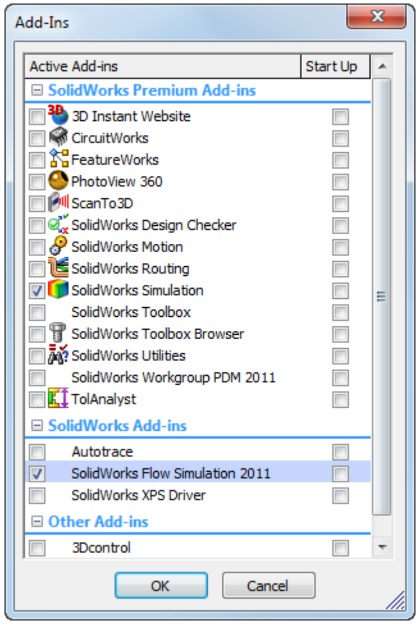
After building your design in SolidWorks, you may need to answer questions like:

- Will the part run quickly?
- How will it handle air resistance?
- Can I use less material without affecting performance?

In the absence of analysis tools, expensive prototype-test design cycles take place to ensure that the product’s performance meets customer expectations. Design analysis makes it possible to perform design cycles quickly and inexpensively on computer models instead. Even when manufacturing costs are not important considerations, design analysis provides significant product quality benefits, enabling engineers to detect design problems far sooner than the time it takes to build a prototype. Design analysis also facilitates the study of many design options and aids in developing optimized designs. Quick and inexpensive analysis often reveals non-intuitive solutions and benefits engineers by allowing them to better understand the product’s behavior.

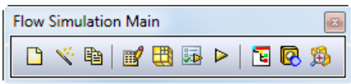
Check Before Using SolidWorks Flow Simulation

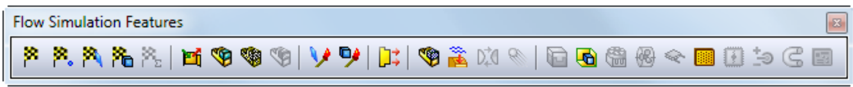
- Make sure SolidWorks Flow Simulation software is installed;
- Click **Tools, Add-ins…** and click **SolidWorks Flow Simulation 2011** to load SolidWorks Flow Simulation.

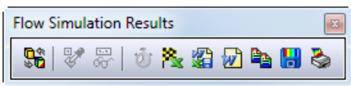


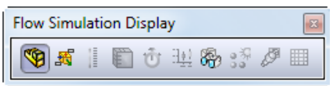
SolidWorks Flow Simulation Toolbars

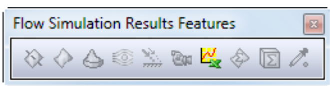
The SolidWorks Flow Simulation toolbars contain shortcuts for many commands. You can also access commands from the SolidWorks Flow Simulation pull-down menu.





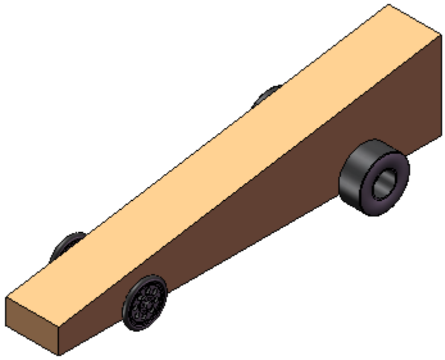






Let's Analyze the Car

We will perform an aerodynamic analysis of the CO_2 car.



1 **Open the car assembly file.**
If the assembly is not already open from the previous lesson, click **Open**.

From the **Open** window, browse to the `CO2 Car Project` directory.

Select `CO2 Car Baseline.sldasm`, and click **Open**.

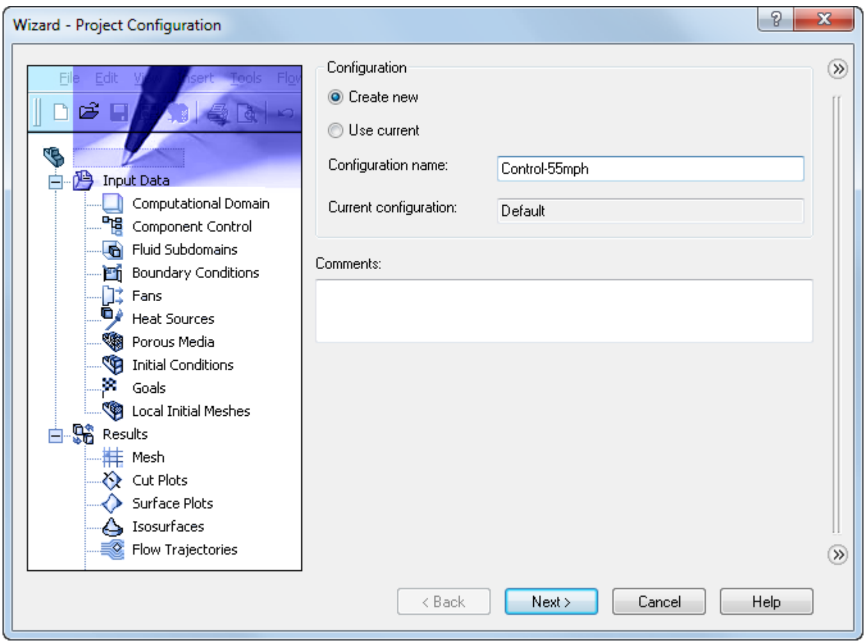
2 **Change the view orientation.**
If the part is not already in the isometric view, click on the **Isometric** tool on the Standard Views toolbar.

Create a Flow Simulation Project

1 **Click Flow Simulation, Project, Wizard...**
Or, click **Wizard** on the Flow Simulation Main toolbar.

2 **Configuration name for the project.**
Select **Create new** to create a new configuration and name it `Control-55mph`.

Click **Next**.

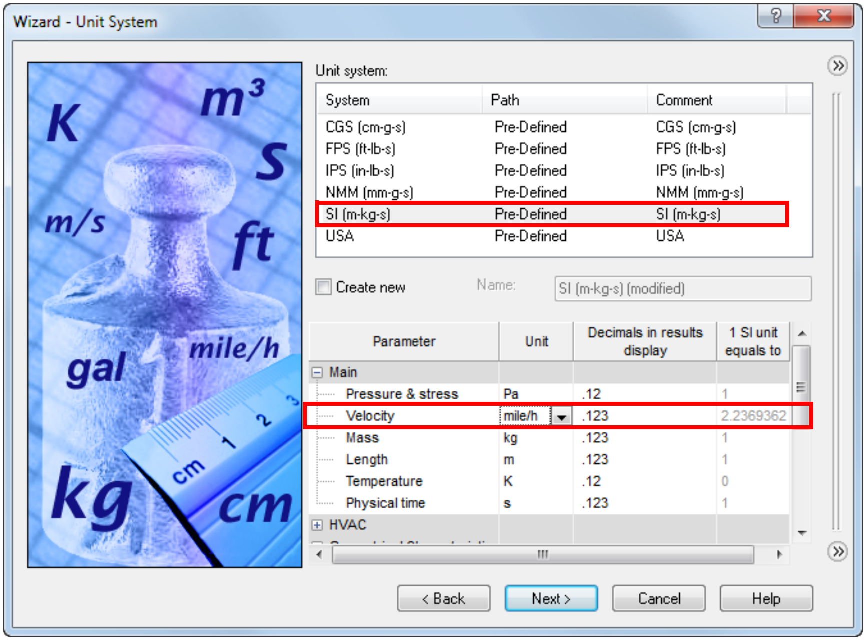


Note: All required analysis data for this project is saved in this configuration, which is associated with SolidWorks.

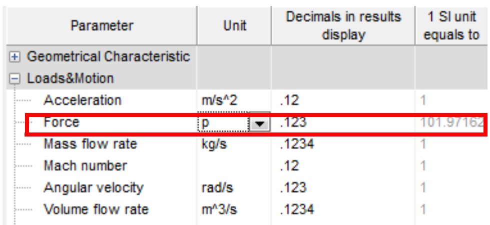
3 Unit System.

Choose **SI(m-kg-s)** in the **Unit system** area.

In the **Parameter** window, under **Main**, set the **Velocity** to **mile/h** (miles per hour).



Scroll down and under **Loads&Motion**, set **Force** to **Gram force** which is displayed as the symbol **p**.



Click **Next**.

Note: You can change the unit system anytime by clicking **Flow Simulation, Units**. You can also create your own custom unit system by clicking **Create new**.

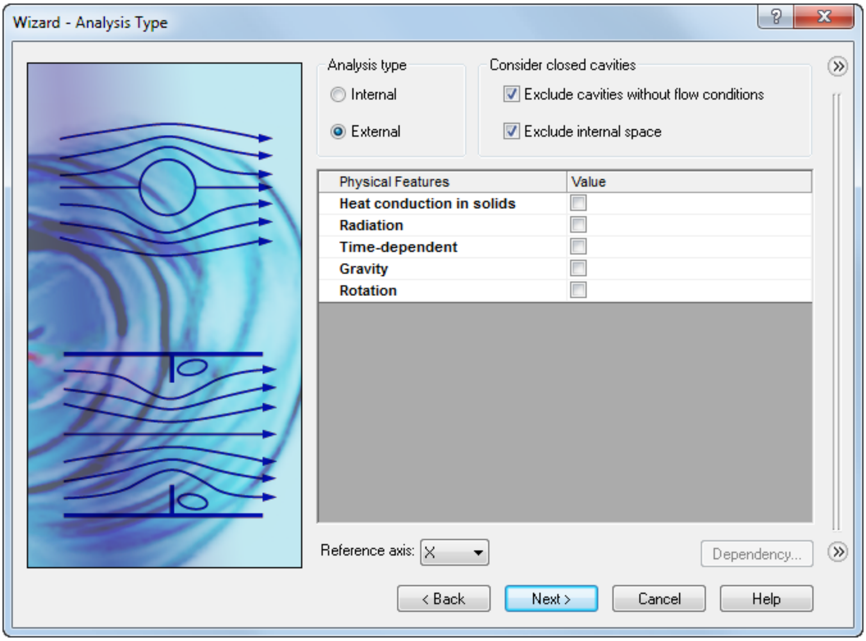
Gram-force

Gram-force is a unit of force, approximately equal to the weight of a 1 gram mass on Earth. However, the local gravitational acceleration *g* varies with latitude, altitude and location on the planet. So to be precise, one gram-force is the force that a 1-gram mass exerts at a place where the acceleration due to gravity is 9.80665 meters per second per second.

4 Analysis Type and Physical Features.

Select **External** as the **Analysis type**.

Select the **Exclude cavities without flow conditions** and the **Exclude internal space** check boxes.



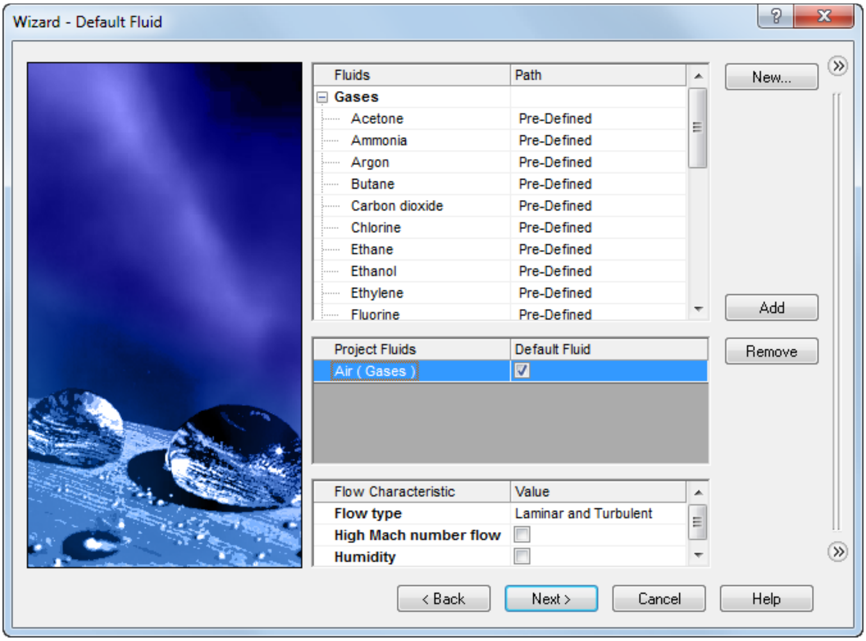
Note: An internal analysis examines enclosed flow pathways while an external analysis examines open flow paths. You would use an internal analysis for something like an exhaust manifold for an automobile engine.

Click **Next**.

5 Default Fluid.

Under **Gases**, select **Air**, and then click **Add**.

Tip: You can also double-click **Air**, or drag and drop it from one list to the other.



Note: Flow Simulation has a database library of several liquids and gases which is called the Engineering Database. With this database you can create your own materials.

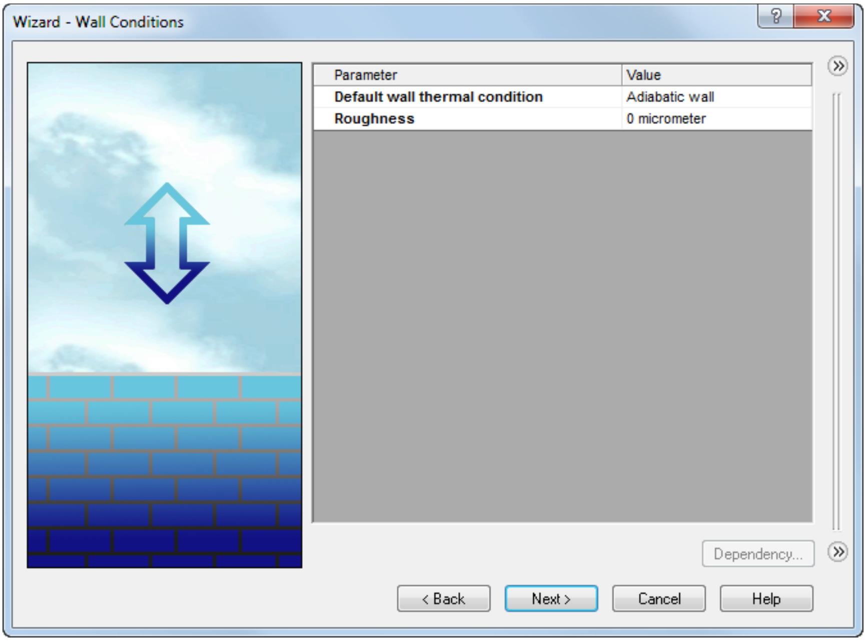
Flow Simulation can analyze either incompressible liquids or compressible gases but not both during the same run. You can also specify other advanced physical features which the program should take into account.

Click **Next**.

6 Wall Conditions.

Use the default values of **Adiabatic wall** and a **Roughness** value of **0 micrometer**.

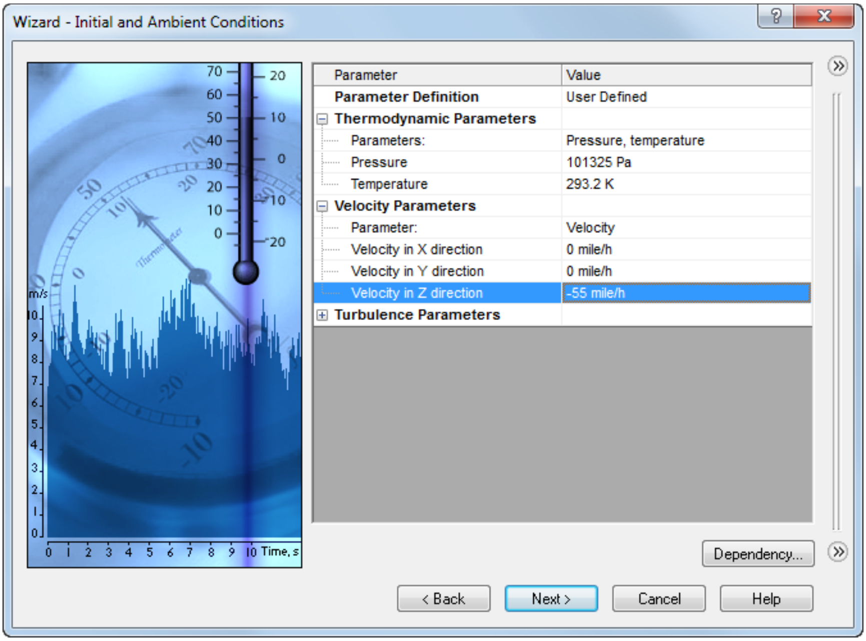
Click **Next**.



7 Initial and Ambient Conditions.

Under **Velocity Parameters**, double-click the value of **Velocity in Z direction** and type **-55 mile/h**.

Note: The minus sign is important! It indicates that the air is flowing *towards* the car.

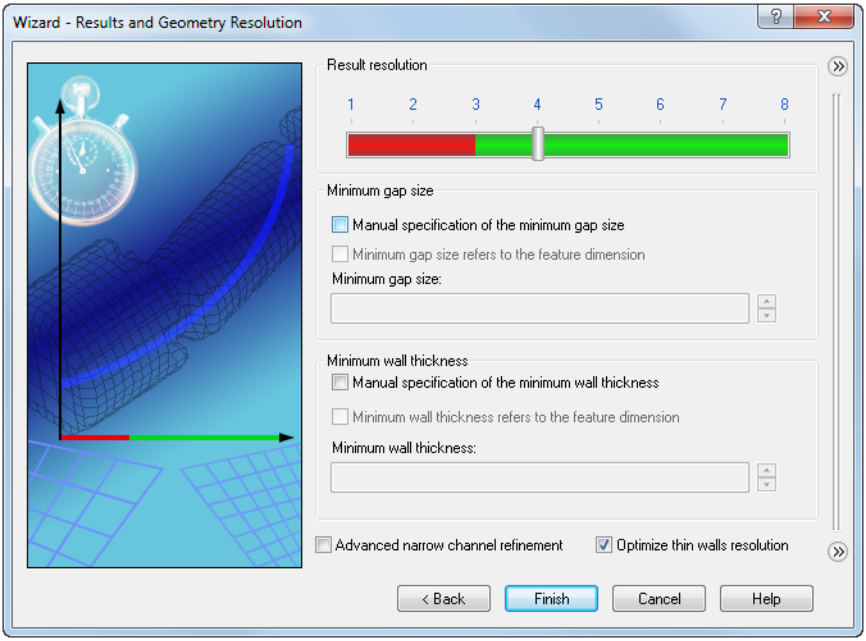


In the real world, the car would be moving through stationary air. In a wind tunnel, the car is stationary and the air is moving. You can think of this Flow Simulation example as a virtual wind tunnel – the car is stationary and the air is moving.

Click **Next**.

8 Results and Geometry Resolution.

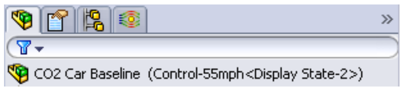
Set the **Result resolution** to **4** which will yield acceptably accurate results in a reasonable amount of time.



Click **Finish**.

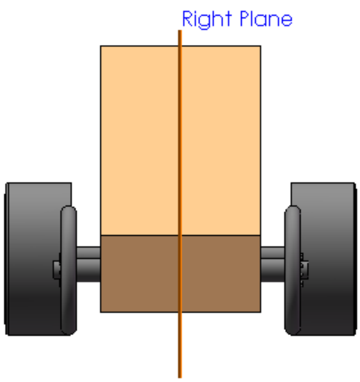
9 Flow Simulation analysis tree.

A tab for the Flow Simulation analysis tree is added to the FeatureManager area.



Using Symmetry

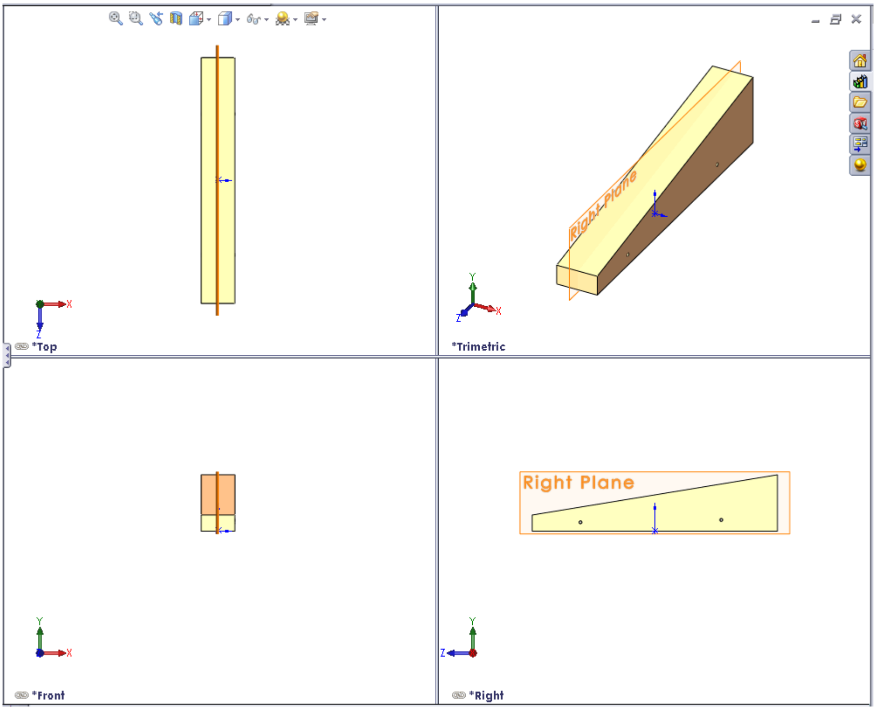
We can reduce the amount of time required for the analysis by taking advantage of the symmetry of the car. If you look at the car from the front view you can see that the right-hand side of the car is the mirror image of the left-hand side and that the `Right` reference plane splits the car down the middle. By specifying symmetry as one of the conditions of the computational domain, we only have to do half of the calculation.



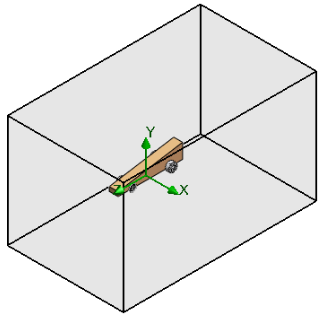
However, when using symmetry, there are some important things to keep in mind:

1. This only works if the object being analyzed is indeed symmetrical.

2. In this model we use half symmetry, therefore any forces you calculate will have to be doubled. For example, the magnitude of the force that represents drag on the car is actually only half the total force because only half of the car is analyzed.
3. Neatness counts! Be sure the car body is built so it is centered with respect to the origin. And, be sure that when you added the car body to the assembly, you positioned it on the origin of the assembly as shown in step **4** on page 20. These two things are related. It doesn't do you any good to position the body at the origin of the assembly if the body wasn't built on the part's origin in the first place.



Computational Domain



Flow Simulation calculations are performed inside a volume called the computational domain. The boundaries of this volume are parallel to the global coordinate system planes. For external flows, the size of the computational domain is automatically calculated based on the size of the model.

In the illustration at the right, the black box represents the computational domain.

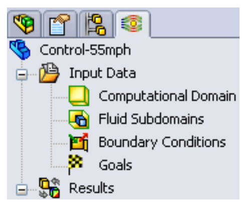
Modifying the Computational Domain

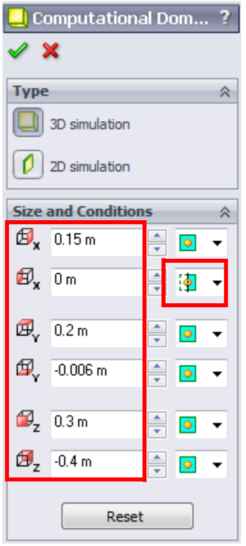
We will make changes to two aspects of the computational domain:

- **Size**
  We are going to reduce the size of the computational domain in order to reduce solving time, at the expense of accuracy. A smaller domain means there are fewer fluid cells to calculate. Using the default sizes for the domain could result in solving times in excess of 1.5 hours even on a moderately fast computer. Such solving times are not practical in a school environment.
- **Boundary Conditions**
  Changing the condition for the X minimum boundary is what defines the symmetry for this example.

1 Show the analysis tree.
Click on the Flow Simulation analysis tree tab .

Expand the `Input Data` listing.





2 Computational domain size.
Right-click `Computational Domain` and select **Edit Definition**.

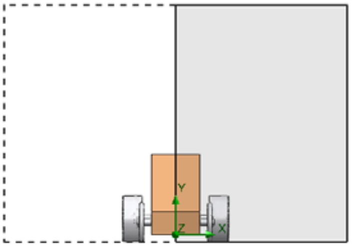
Enter the following values:

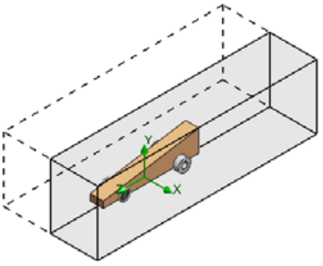
- **X max = 0.15 m**
- **X min = 0.0**
- **Y max = 0.2 m**
- **Y min = - 0.006 m**
- **Z max = 0.03 m**
- **Z min = - 0.4 m**

Also, choose **Symmetry** from the drop-down for X min.

Click **OK**.

3 Results.
The resulting computational domain is shown in the illustrations below.





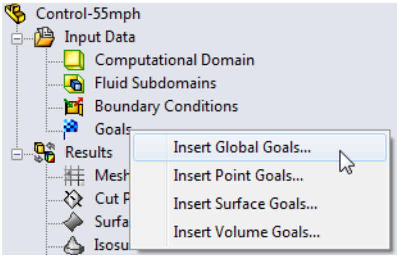
Note: Reducing the size of our computational domain and utilizing a symmetry boundary condition will allow for more efficient calculations. We must always be sure, however, that our computational domain is large enough to allow our flow field to fully develop.

Setting Goals

You can specify the following four engineering goals:

- **Global Goal**
  A physical parameter calculated within the entire computational domain.
- **Surface Goal**
  A physical parameter calculated on a user-specified face of the model.
- **Volume Goal**
  A physical parameter calculated within a user-specified space inside the computational domain, either in the fluid or solid.
- **Equation Goal**
  A goal defined by an equation with the specified goals or parameters of the specified project's input data features as variables.

4 **Insert global goals.**
Right click `Goals` in the Flow Simulation analysis tree and select the **Insert Global Goals** from the shortcut menu.



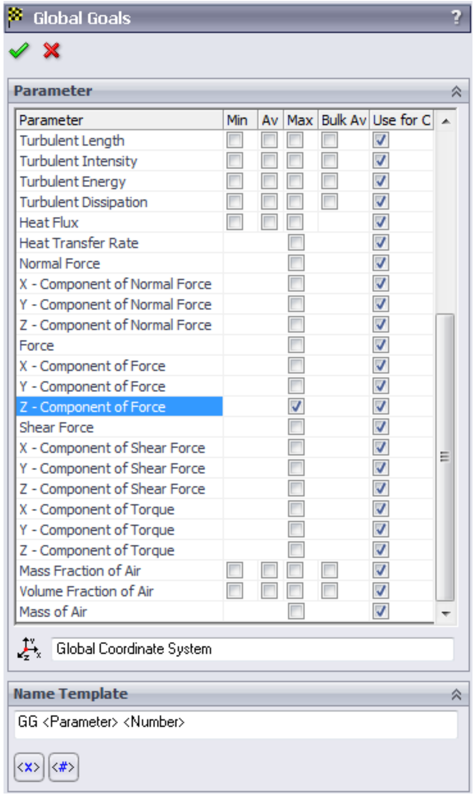
5 Setting the goal for drag.
Drag the boundary of the PropertyManager window to the right to make it wider. This makes it easier to read the parameter names.

In the **Parameter** list, find the parameter named **Z - Component of Force**.

Tip: You will have to scroll down the towards the bottom.

Click the check box.

Click **OK**.

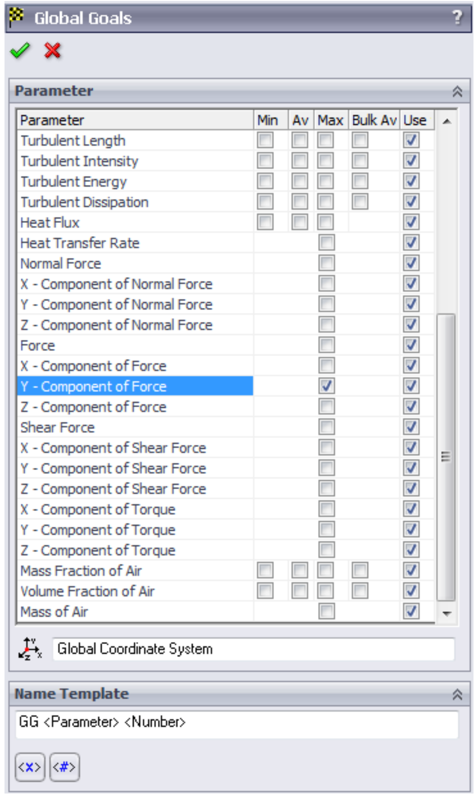


6 Insert a second global goal.
Right click `Goals` again and select **Insert Global Goals**.

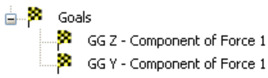
7 Setting the goal for lift.
In the **Parameter** list, find the parameter named **Y - Component of Force**.

Click the check box.

Click **OK**.



8 Rename the goals.
Two goal icons appear in the Flow Simulation analysis tree.

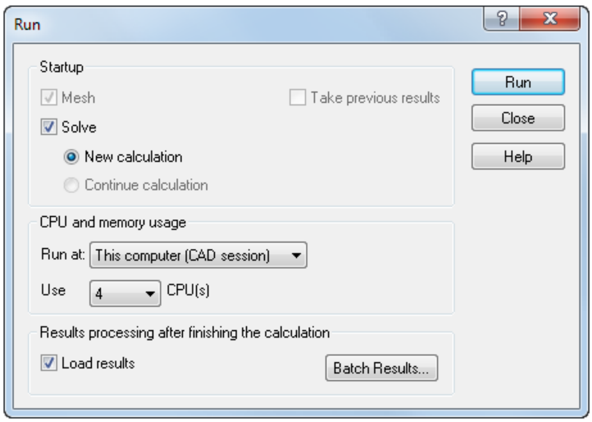


Rename the Z component to `Drag` and the Y component to `Lift`.

Running the Analysis

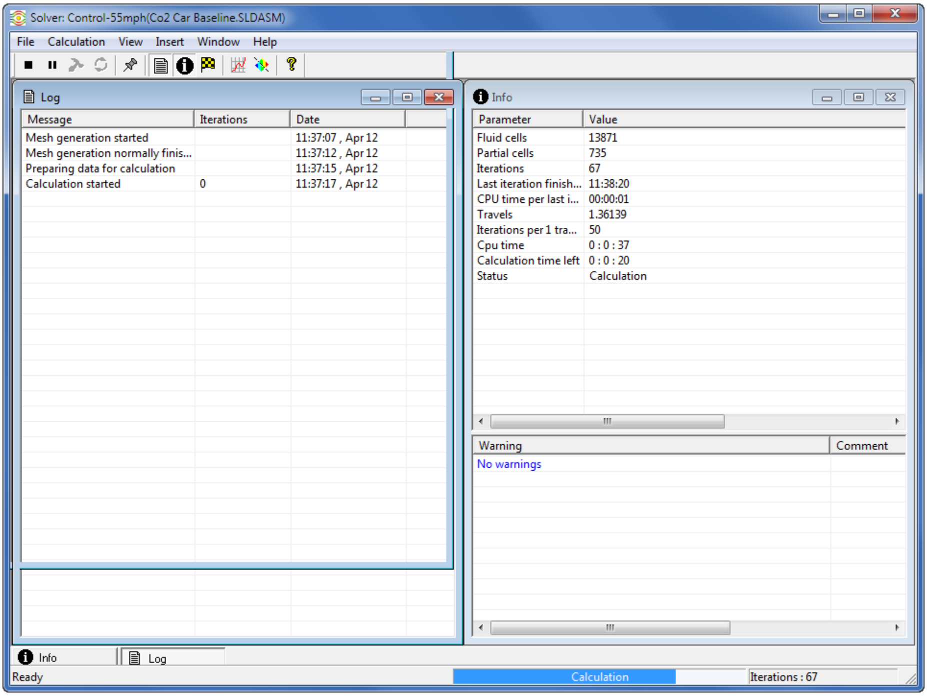
This starts the calculation for the current project.

9 Run the analysis.
Right-click `Control-55mph` and click **Run...**, or click **Flow Simulation , Solve, Run**, or click **Run Solver** ▷ on the Flow Simulation toolbar.



10 Solver information.

The **Solution Monitor** window appears after a minute or so. On the left of the window is a log of each step taken in the solution process. On the right is an information window with mesh information and any warnings concerning the analysis.



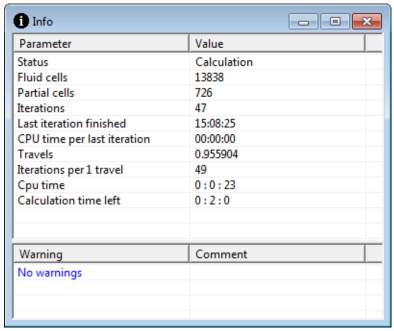
Monitoring the Solution

The **Solver** window has its own toolbar that you can use to display the current results during the calculation.

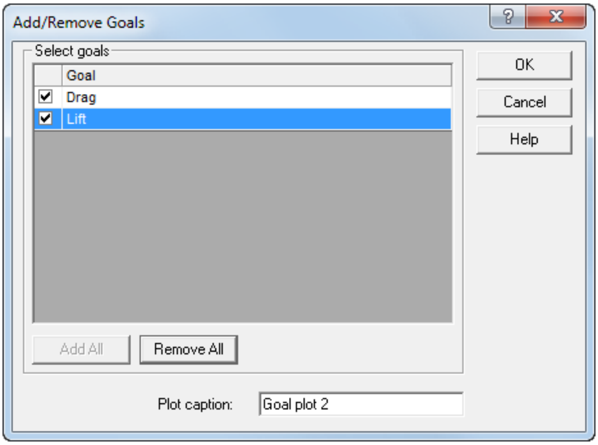
- **Stop** 
  Stops the calculation. You will be asked to save the current results. If you save the current results, you will be able to continue the calculation from the saved state rather than starting over.

- **Suspend** 
  Suspends the calculation. When the calculations are suspended, you cannot modify either the model or Flow Simulation project,. However, CPU resources used by Flow Simulation are released.

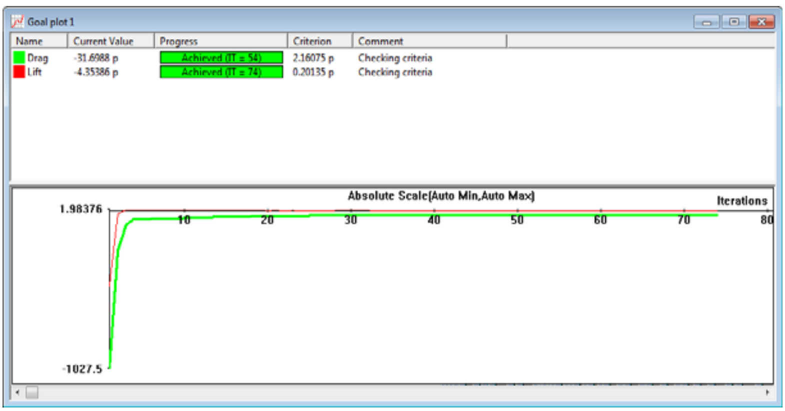
- **Information**
  Displays mesh statistics, information about the current calculation step and number of iterations as well as warning messages if an inaccurate solution is possible.



- **Goal Plot**
  When you click **Goal Plot**, the **Add/Remove Goals** dialog appears. Select the goals whose plots you want to view and click **OK**.

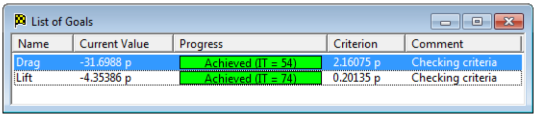


For each goal selected in the **Add/Remove Goals** dialog box, the **Goal Plot** shows the goal convergence diagram.



- **Goal Table**

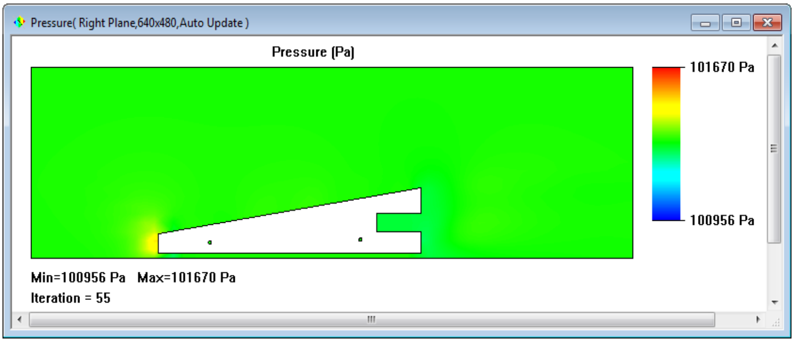
  Shows the list of all specified goals. The **Goal Table** contains the same information as the upper portion of the **Goal Plot** window.



Name	Current Value	Progress	Criterion	Comment
Drag	-31.6988 p	Achieved (IT = 54)	2.16075 p	Checking criteria
Lift	-4.35386 p	Achieved (IT = 74)	0.20135 p	Checking criteria

- **Insert Preview**

  Allows you to view the current results on the specified plane.



11 Pause the calculation.

After about 55 iterations, click **Suspend** on the **Solver** window's toolbar.

This pauses the calculation so you can explore some of the different types of previews.

12 Preview the results.

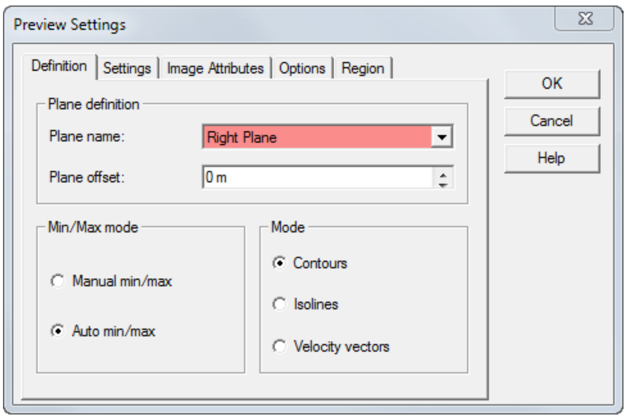
Click **Insert Preview** on the **Solver** window's toolbar.

The **Preview Settings** dialog box appears.

For the **Plane definition**, select the **Right Plane**.

For **Mode**, select **Contours**.

Do not click **OK** yet. Continue with the next step.

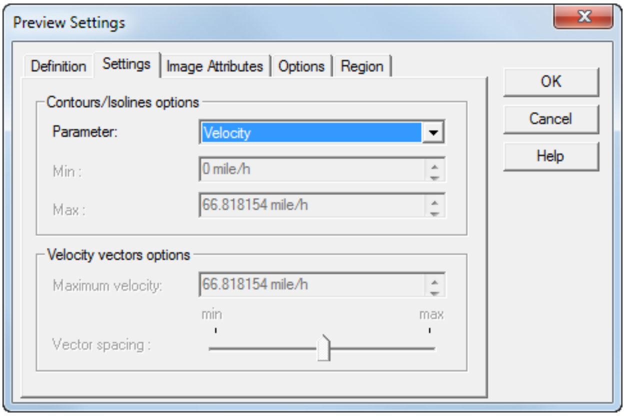


13 Settings.

Click the **Settings** tab.

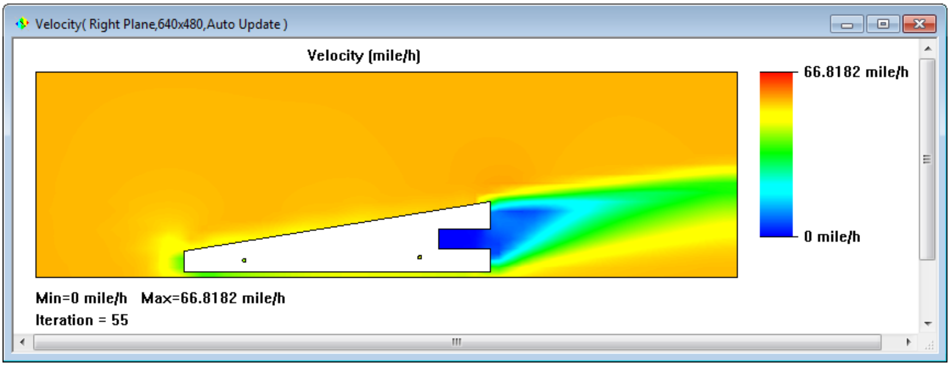
For the **Parameter**, select **Velocity.**

Click **OK**.



14 Preview.

The plot preview is displayed in its own window.



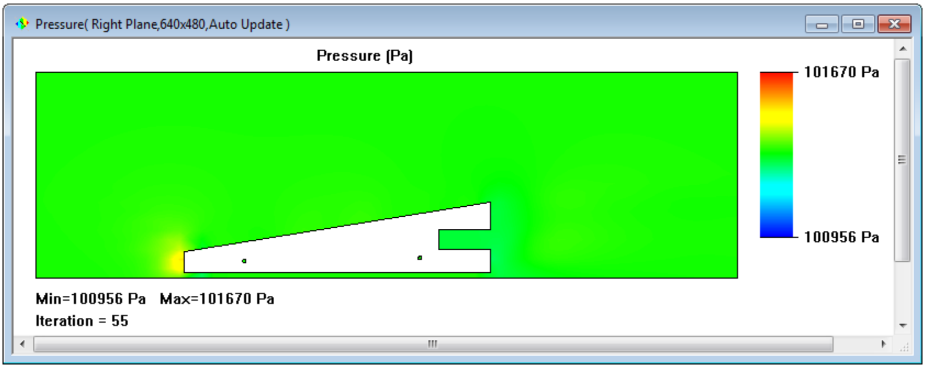
15 Preview the pressure.
Click **Insert Preview** on the **Solver** window's toolbar.

In the **Preview Settings** dialog box select the **Right Plane** for the **Plane definition**.

For **Mode**, select **Contours**.

Click the **Settings** tab.

For the **Parameter**, select **Pressure and** click **OK**.



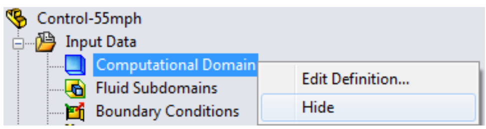
16 Resume calculation.
Click **Suspend** on the **Solver** window's toolbar to resume the calculation.

17 Completion.

The status bar at the bottom of the window will indicate when the solver is finished.

18 Close the Solver window.
In the **Solver** window, click **File, Close**, or click the red X in the upper right corner of the window.

19 Hide the computational domain.



In the Flow Simulation analysis tree, right-click the `Computational Domain` icon and select **Hide** from the short cut menu. This hides the box that represents the size and position of the computational domain.

20 Save the file.
After investing time in running the analysis it is prudent to save your work.

Viewing the Results

Once the calculation finishes, you can view the saved calculation results through numerous Flow Simulation options in a customized manner directly within the graphics area. The results options are:

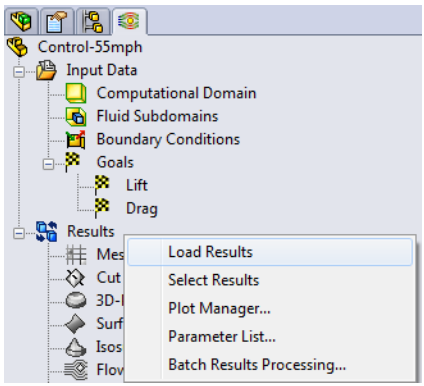
- Cut Plots (section view of parameter distribution)
- Surface Plots (parameter distribution on a selected surface)
- Isosurfaces
- Flow Trajectories (streamlines and particle trajectories)
- Goal Plot (behavior of the specified goals during the calculation)
- XY Plots (parameter change along a curve, sketch)
- Surface Parameters (getting parameters at specified surfaces)
- Point Parameters (getting parameters at specified points)
- Report (project report output into Microsoft Word)
- Animation of results

We will view the surface plots and the flow trajectories next.

Accessing the Results

1 Load the results.

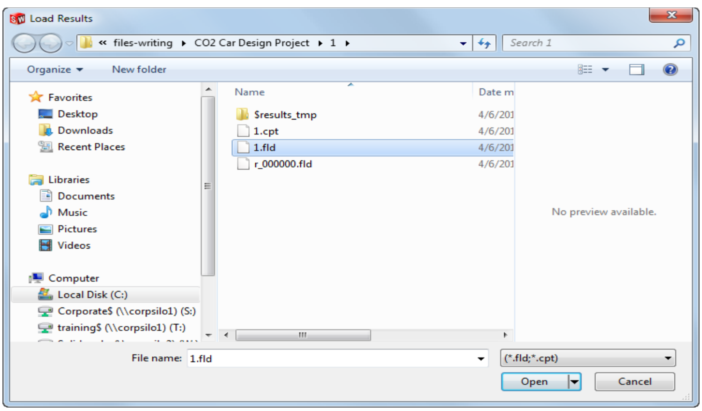
Right-click `Results` in the Flow Simulation analysis tree and select **Load Results**.



If **Unload Results** appears in the list, the results have already been loaded.

The **Load Results** window opens.

Select `1.fld` and click **Open** to load the results file.

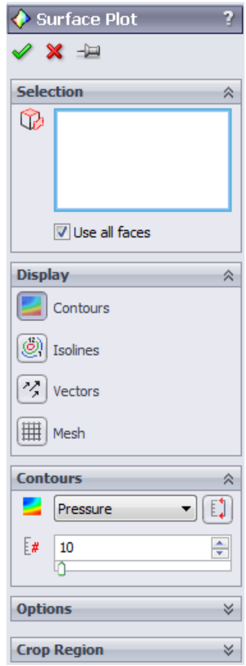


2 Insert a surface plot.
Right-click `Surface Plots` in the Flow Simulation analysis tree and select **Insert**.

Select **Use all faces**.

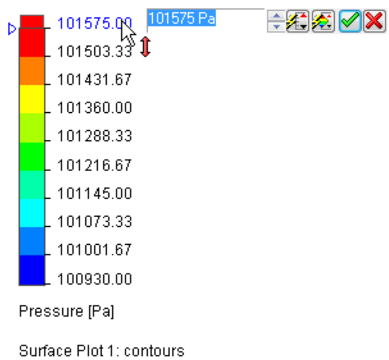
Under **Display**, click **Contours**.

Click **OK**.



3 Change the legend.
Click on the top value of the legend and enter **101575 Pa**.

Repeat this process for the bottom value and enter **100930 Pa**.

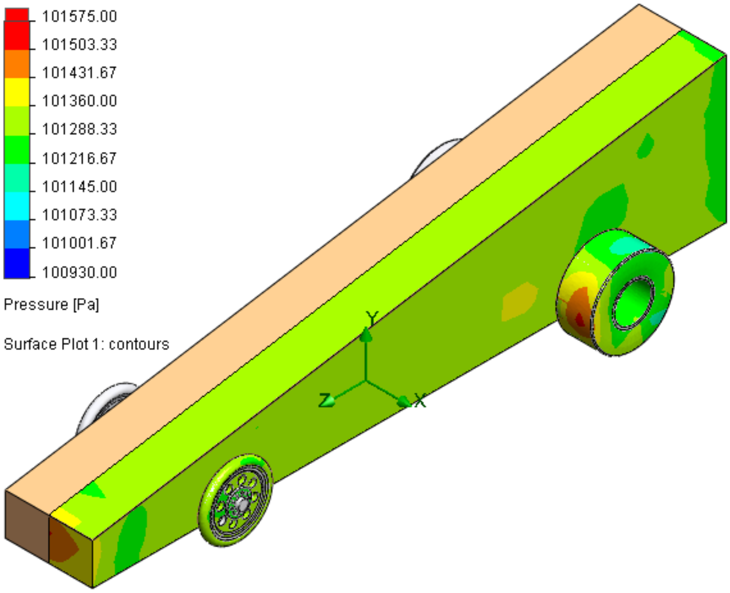


Note: The reason we do not use the default values is because if we make a design change to the car and rerun the analysis, the minimum and maximum pressure values will be different. That means red would represent one pressure on one plot and a different pressure on a different plot. Using the same minimum and maximum settings for each analysis allows for meaningful comparisons between different iterations of the design.

4 Surface plot results.

The **Surface Plot** displays the pressure distribution on the selected model faces or SolidWorks surfaces.

Only half of the car displays the colors because we used symmetry in the calculations. We assume the right-hand side of the car would be identical.



Interpreting the Results

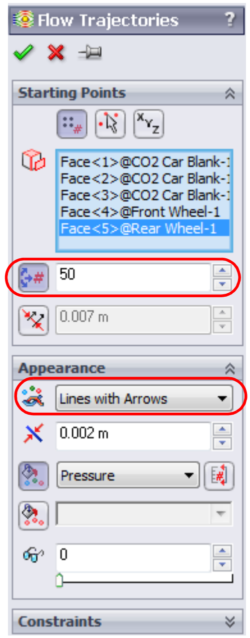
Red indicates areas of high pressure. Blue indicates areas of low pressure. By looking at the surface plot we can see that the pressure is highest on the front face of the car body and on the front portion of the rear wheels.

5 Hide the surface plot.

Right-click `Surface Plot 1` and select **Hide**. This hides the surface plot so we can more easily see the flow trajectories. You can always turn the surface plot back on later using **Show**.

Flow Trajectories

Flow trajectories are another way to qualitatively view the results of the analysis. They are analogous to the streamers of smoke in a wind tunnel.

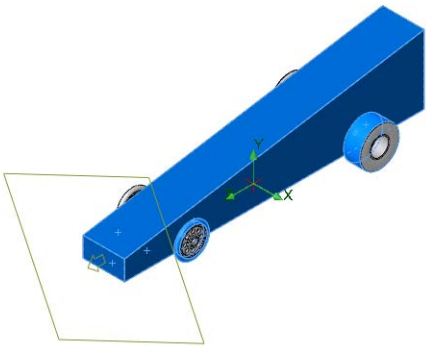


6 Inserting a flow trajectory.

Right-click `Flow Trajectories` in the Flow Simulation analysis tree, and select **Insert**.

For **Reference**, select the faces of the car body as shown:

- Front, top, and left side faces
- Faces that form the tread portion of the front and rear wheels



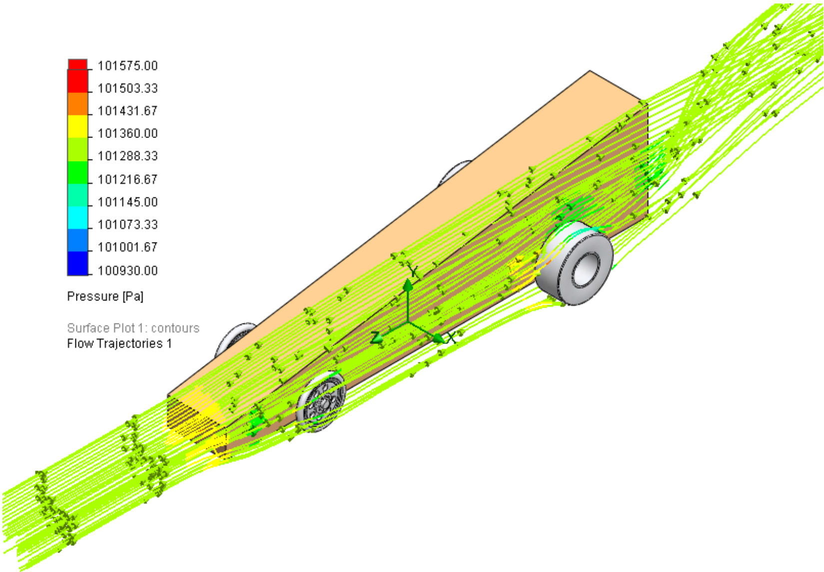
Set the **Number of Points** to **50**.

For **Draw trajectories as**, select **Line with Arrow**.

Leave the other settings at their default values.

Click **OK**.

7 Resulting flow trajectory.
This type of display helps visualize how the air flows around the car.



Experiment With Other Flow Trajectories

There are two ways to experiment with flow trajectories:

- Edit the definition of the existing plot
- Insert a new plot

If you create multiple flow trajectories, you can display them one at a time or you can display several at the same time.

We will create some other flow trajectories.

8 Hide the flow trajectory.
Right-click `Flow Trajectories 1` and select **Hide**.

9 Insert a new flow trajectory.

Right-click `Flow Trajectories`, and select **Insert**.

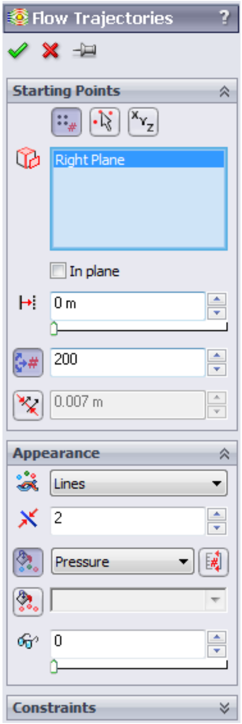
For **Reference**, select the `Right` reference plane of the assembly.

Set the **Number of Points** to **200**.

Click the **Settings** tab.

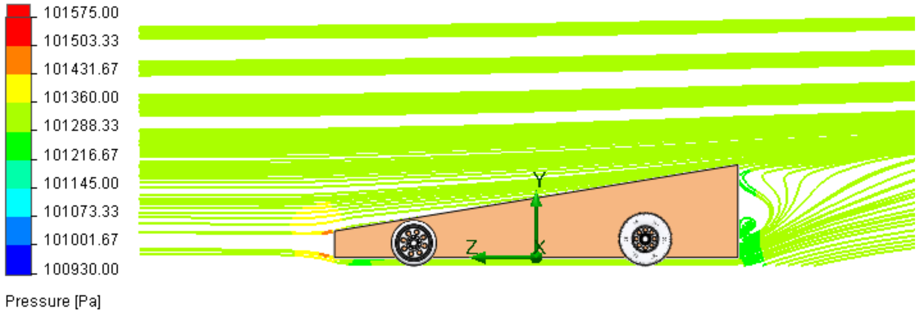
For **Draw trajectories as**, select **Line**.

Click **OK**.

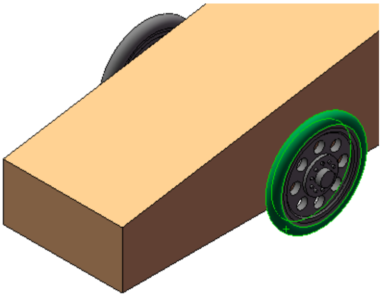


10 Right side view.

Change the view to a right side orientation. Notice the turbulence behind the body of the car.



11 Insert another new flow trajectory.
Hide `Flow Trajectories 2` and then insert a new flow trajectory.



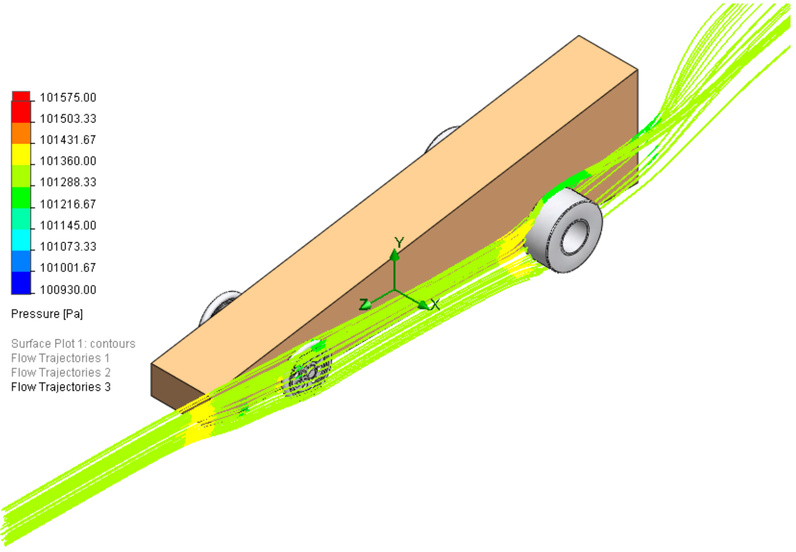
For **Reference**, select the surface that forms the tread of the front wheel.

Set the **Number of Points** to **50**.

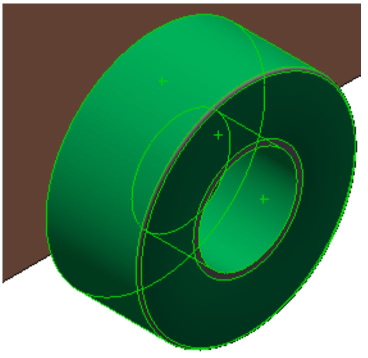
For **Draw trajectories as**, select **Line**.

Click **OK**.

12 Change to an isometric view.
The lower number of trajectory lines makes it easier to see if there is significant turbulence surrounding the front wheel. It appears that the narrow profile of the front wheel does not introduce much turbulence.



13 Repeat for the rear wheel.
Hide `Flow Trajectories 3` and then insert a new flow trajectory.



For **Reference**, select the surfaces that form the tread, sidewall, and hub of the front wheel.

Set the **Number of trajectories** to **75**.

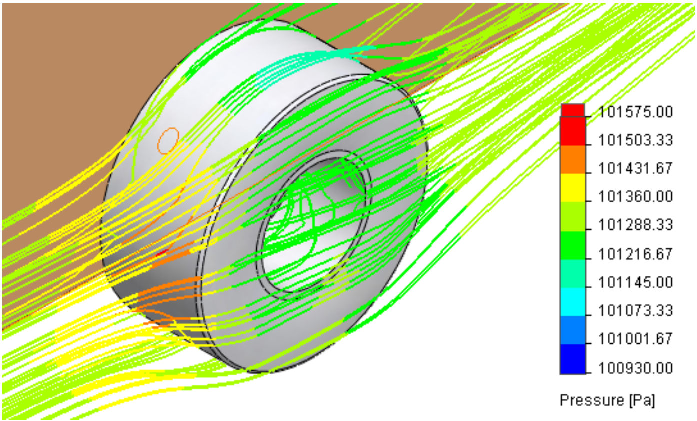
For **Draw trajectories as**, select **Line**.

Click **OK**.

14 Change to an isometric view and zoom in.

The flow trajectories reveal several conditions:

- The red color of the trajectories in front of the wheel indicate an area of high pressure. Perhaps a narrower wheel would reduce drag.
- There is turbulence in the area of the wheel hub.
- The flow trajectories behind the wheel are fairly smooth indicating a lack of turbulence.



15 Edit definition.

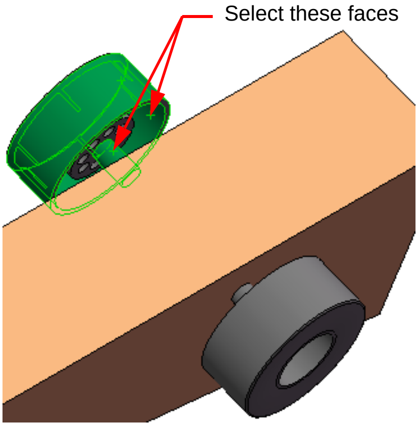
Right-click `Flow Trajectories 4` and select **Hide**.

Rotate the view so you can see the underside of the car body and the interior of the rear wheel.

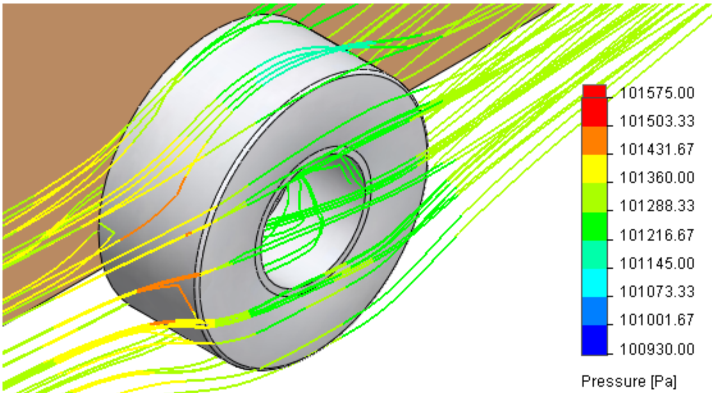
Then, right-click `Flow Trajectories 4` again and select **Edit Definition**.

For **Reference**, select the surfaces that form the inside of the rear wheel and the wheel's axle.

Click **OK**.



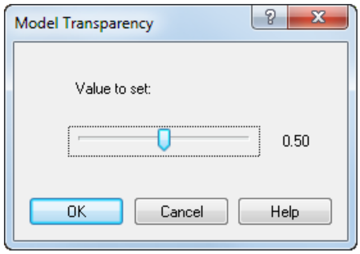
Reset the view to an isometric and zoom in on the rear wheel.



It is hard to see the flow lines between the wheel and the car body because they are obscured by the geometry of the wheel.

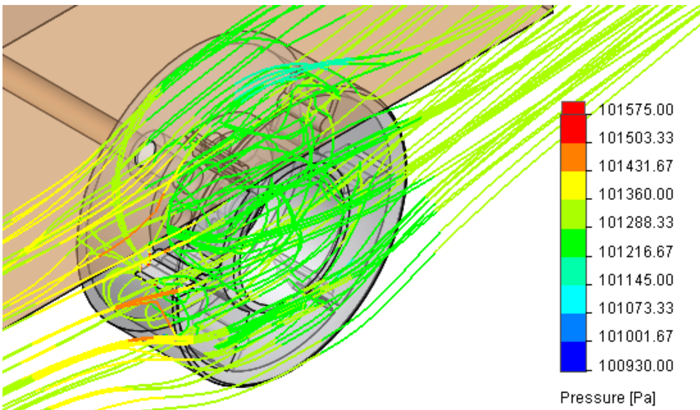
16 Transparency.

Click **Set Model Transparency** on the Flow Simulation Display toolbar, or click **Flow Simulation, Results, Display, Transparency**.



The **Model Transparency** dialog box appears.

Click **OK** to use the default transparency value of **0.50** (50%).



It is now easier to see the turbulence in the airflow between the wheel and car body.

17 Turn off transparency.

Click **Set Model Transparency**, or click **FloWorks, Results, Display, Transparency**.

Drag the slider to set the transparency **Value** to **0.0**.

Click **OK**.

18 Hide the flow trajectories plot.

Right-click `Flow Trajectories 4` and select **Hide**.

Quantitative Results

The preceding examples of surface plots and flow trajectories were excellent tools for visualizing how the air flows around the car. However, they are more qualitative than quantitative. Let's move on to a more quantitative interpretation of results.

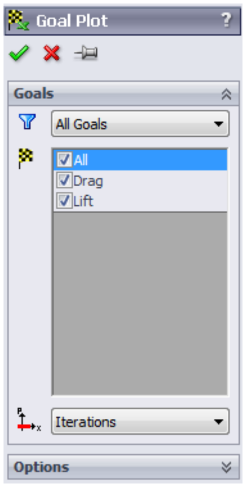
1 Create a goals plot.

In the analysis tree, expand the `Results` listing and right-click `Goal Plots`.

Select **Insert** from the shortcut menu.

Click **All**.

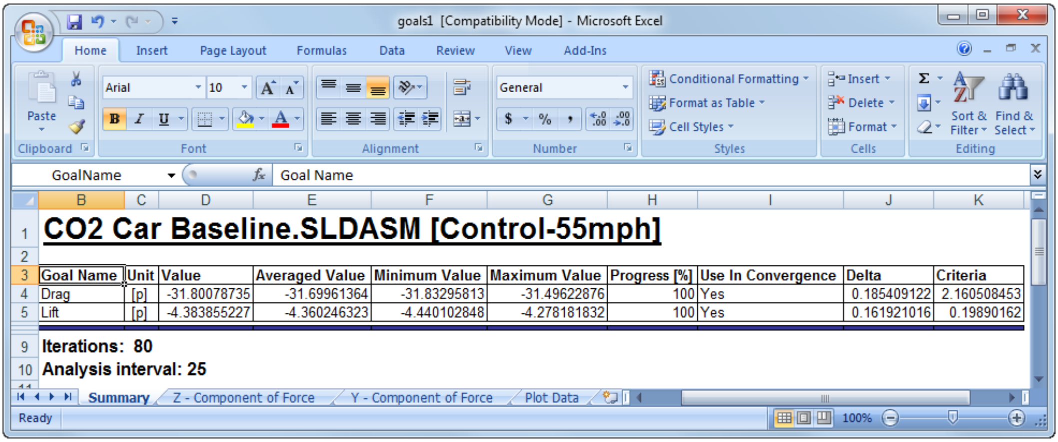
Click **OK**.



2 Excel spreadsheet.

Microsoft® Excel is launched and a spreadsheet opens. Pay particular attention to the first three columns. They show the name of the goal, the units (gram-force, in this case) and the value.

Note: Remember! We used symmetry when doing the analysis so these values have to be doubled to obtain the correct value for the entire car.



CO2 Car Baseline.SLDASM [Control-55mph]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
Drag	[p]	-31.80078735	-31.69961364	-31.83295813	-31.49622876	100	Yes	0.185409122	2.160508453
Lift	[p]	-4.383855227	-4.360246323	-4.440102848	-4.278181832	100	Yes	0.161921016	0.19890162

Iterations: 80
Analysis interval: 25

Note: We reformatted the column headers to make the columns narrower so we could fit the image on the page.

3 Save and close the assembly.

Be sure to save the Excel spreadsheet when prompted.

Units, Values, and Interpreting the Results

As we discussed on page 35, gram-force is a unit of force approximately equal to the weight of a 1-gram mass on Earth. The drag on the car is a force. Grams are a unit of mass. So it is not accurate to say the drag is approximately 31.8 grams.

The correct way to state the results is to say we have a drag force of approximately 31.8 grams-force and a downward lift force of approximately 4.4 grams-force.

Conclusion

From the analysis, it is obvious that the car body blank right out of the box is not very aerodynamic. In the next lesson we will modify the shape of the body and rerun the analysis to determine the effects.

Lesson 4
Making Design Changes

When you complete this lesson, you will be able to:

- Create a configuration in a part;
- Use splines to sketch free-form shapes that cannot be represented using lines and arcs;
- Create extruded cuts using an open sketch contour;
- Use part configurations in an assembly.

Changing the CO_2 Car Design

Based on the analysis of the car using Flow Simulation, we conclude that the shape of the body needs to be redesigned to direct the air around the car with a more gradual and smoother changes in direction. We need to make the car rounder.

Configurations

Configurations allow you to represent more than one version of the part within a single SolidWorks file. For example, by suppressing the features and changing the dimension values of the model, the design can be altered easily without creating another new model. Any configuration may be changed to a dimension of a different value.

Both parts and assemblies can support configuration adjustments.

If a configuration is not created, the model which we create is saved automatically with a configuration named `Default`.

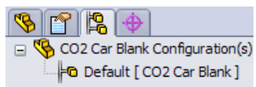
Flow Simulation creates a configuration to store all analysis data. The name of this configuration is the name you entered in the Flow Simulation Wizard. In this case it is `Control-55mph`.

Modifying the Model

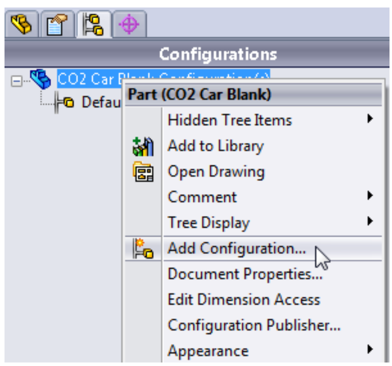
1 **Open the `CO2 Car Blank` part.**
Click **Open**, or click **File, Open**.

Browse to the location of the `CO2 Car Blank`, select it, and click **Open**.

2 **Switch to ConfigurationManager.**
Click the ConfigurationManager tab to change from the FeatureManager design tree to the ConfigurationManager.



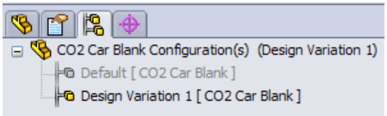
3 **Add a new configuration.**
The current configuration is named `Default`. Right-click on the file name and select **Add Configuration**.



Type `Design Variation 1` as the name.

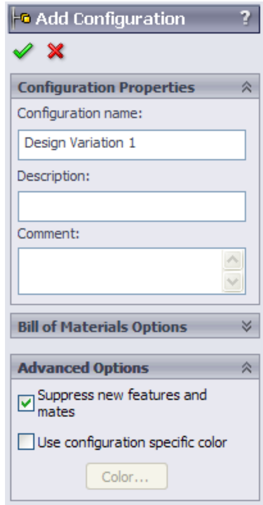
Under the **Advanced Options**, make sure that the **Suppress new features and mates** option is selected.

Click **OK** to add the configuration.

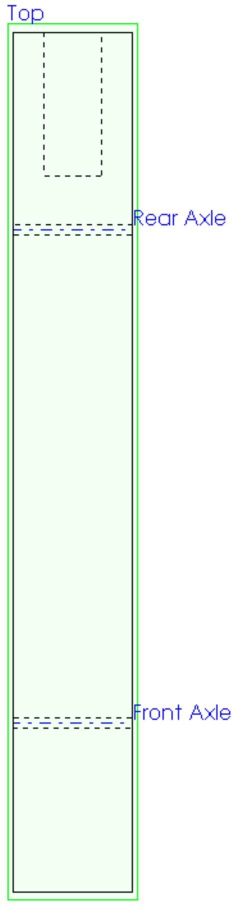


Note: **Suppress** is used to temporarily remove a feature. When a feature is suppressed, the system treats it as if it doesn't exist. This means other features that are dependent on it will be suppressed also. Suppressed features can be unsuppressed at any time. The **Suppress features** option means that as new features are added, they are suppressed in all of the configurations *except* the active one.

The new configuration is active. Any subsequent changes to the part are stored as part of the configuration.



4 Create a new sketch.
Select the `Top` reference plane and click **Sketch**.



5 Top view orientation.

Click **Top** on the Standard Views toolbar to change to the `Top` view orientation.

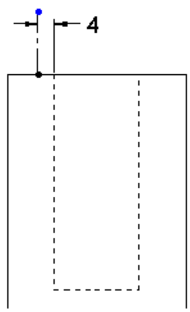
6 View axes.

Click **View, Axes** to display the axes that run through the center of the two axle holes. You will use these for references when sketching.

7 Show hidden lines.

Click **Hidden Lines Visible** on the View toolbar.

This enables you see the location of `Power Plant Chamber` (the hole for the CO_2 cartridge) so you can maintain the minimum thickness surrounding it as required by the specifications.



8 Construction lines.

Zoom in on the area around the `Power Plant Chamber`.

Click **Tools, Sketch Entities, Centerline**, or click **Centerline** on the Sketch toolbar.

Sketch a vertical construction line as shown at the right.

Click **Smart Dimension** on the Sketch toolbar, or click **Tools, Dimensions, Smart**.

Dimension the construction line to be **4mm** from the edge of the hole for the CO_2 cartridge.

9 Centerpoint arc.
Zoom in on the front end of the car.

Click **Tools, Sketch Entities, Centerpoint Arc**, or click **Centerpoint Arc** .

Position the pointer to the right of the car body.

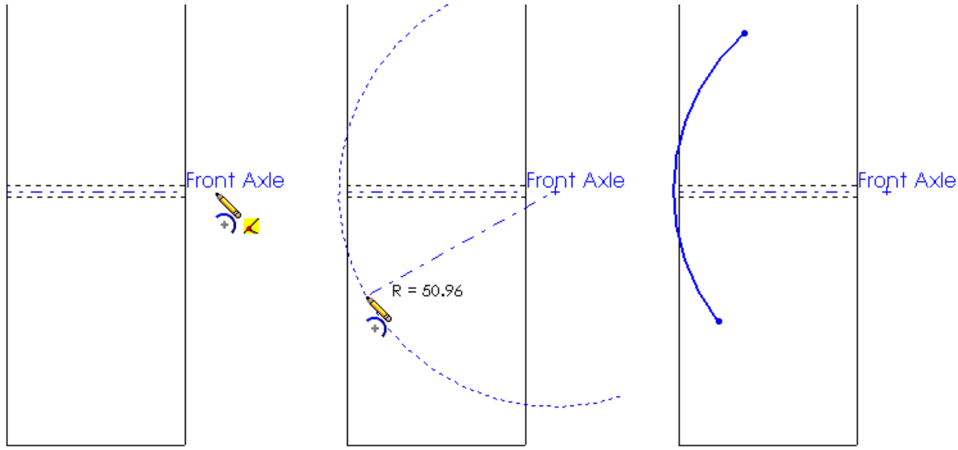
Look for the **Coincident** pointer that indicates you are capturing a **Coincident** relation between the centerpoint of the arc and the axis.

Click the mouse button to establish the centerpoint of the arc.

Move the mouse to establish the starting point of the arc.

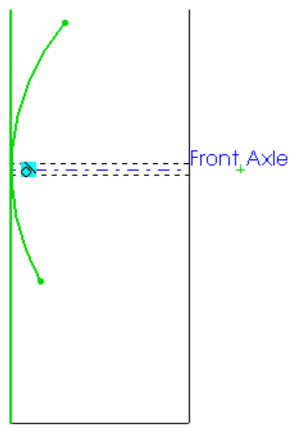
Press and hold the mouse button and drag to define the length of the arc.

The result should look like the arc shown in the rightmost illustration.



Step 1: Locate the center Step 2: Define the radius Step 3: Define the arc length

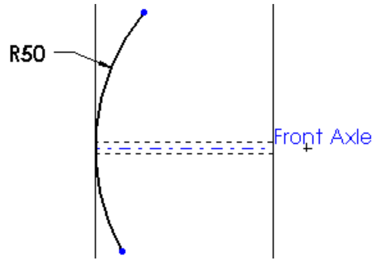
10 Tangent relation.
Press **Ctrl** and select the edge of the car body and the arc. In the PropertyManager, select **Tangent** and click **OK**.



11 Dimension the arc.

Click **Smart Dimension** on the Sketch toolbar, or click **Tools, Dimensions, Smart**.

Dimension the arc with a radius of **50mm**.



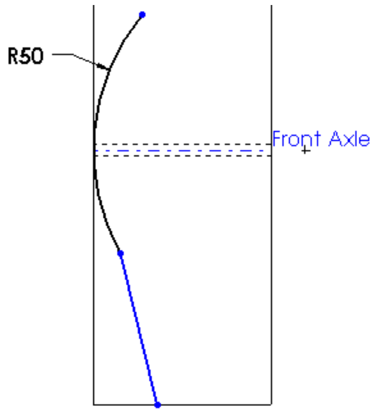
12 Sketch a line.

Click **Tools, Sketch Entities, Line**, or click **Line** on the Sketch toolbar.

Sketch a line from the endpoint of the arc to the edge of the car body as shown.

Be careful not to snap to the midpoint of the car body edge.

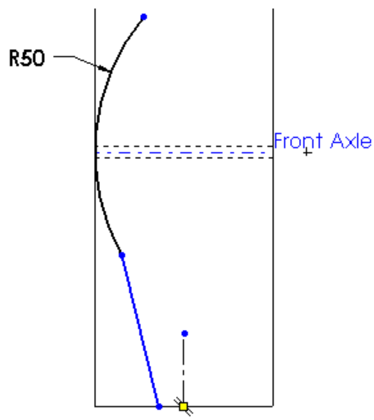
The line should be at an angle as shown. It should *not* be vertical and it should *not* be tangent to the arc.



13 Sketch a construction line.

Click **Tools, Sketch Entities, Centerline**, or click **Centerline** on the Sketch toolbar.

Sketch a short, vertical construction line as shown, starting at the midpoint of the edge of the car body.



14 Mirror.

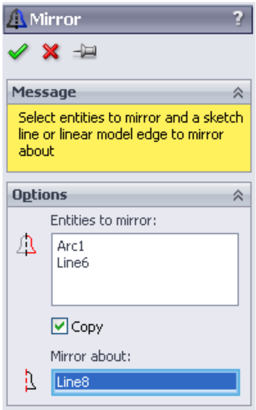
Click **Tools, Sketch Tools, Mirror**, or click **Mirror Entities** on the Sketch toolbar.

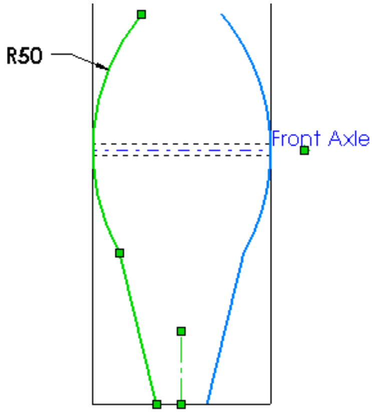
For **Entities to mirror**, select the arc and line.

For **Mirror about**, select the construction line.

Make sure **Copy** is selected. A preview indicates the results of the mirror operation.

Click **OK**.

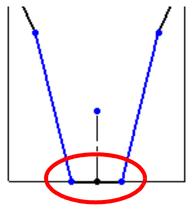




15 Sketch a line.

Click **Tools, Sketch Entities, Line**, or click **Line** on the Sketch toolbar.

Sketch a line between the endpoints of the two angled lines.



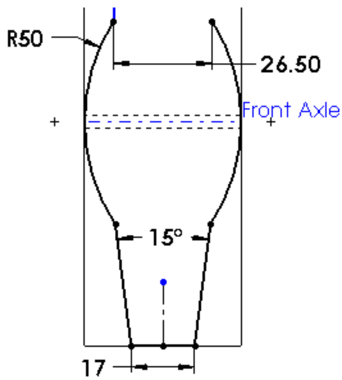
16 Dimensions.

Click **Smart Dimension** on the Sketch toolbar, or click **Tools, Dimensions, Smart**.

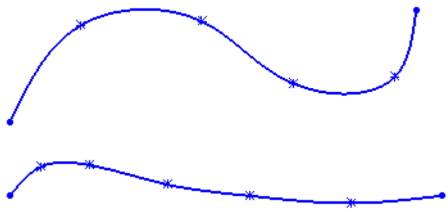
Dimension the length of the line you created in step **15** and set the value to **17mm**.

Dimension the angle between the two angled lines and set the value to **15°**.

Dimension the distance between the ends of the two arcs and set the value to **26.50mm**.



Splines



Splines are used to sketch curves that have continuously changing shape. Splines are defined by a series points between which the SolidWorks software uses equations to interpolate the curve geometry. Splines are very useful for modeling free-form shapes that are smooth and fair. [Fair is a term often used in boat building. A “fair curve” is one that is as smooth as it can be as it follows the path it must take around the hull of a boat; it is free of extraneous bumps or hollows.]

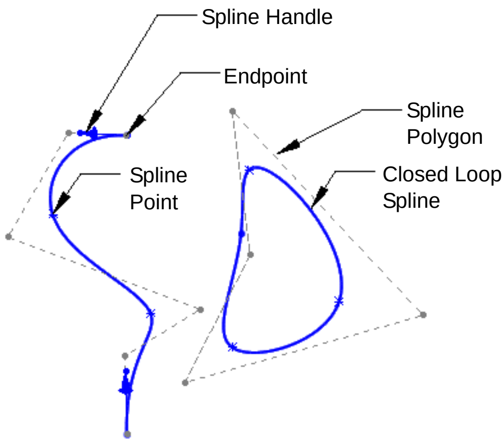
You can modify a spline by adding or deleting points, moving the points, dimensioning the points, changing tangency at the points, or adding geometric relations.

Where to Find It

- Click **Spline** on the Sketch toolbar.
- Or, click **Tools, Sketch Entities, Spline**.

The Anatomy of a Spline

A spline in the SolidWorks software has several components and controls. Understanding what controls and analytical tools are available will help you get the most out of your splines.

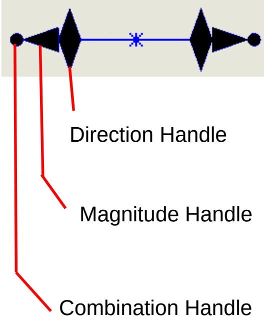


- **Endpoints**
  Every spline has at least one endpoint. A closed loop spline has a single endpoint where the ends are tangent to one another. Open loop splines have two endpoints. An open loop spline can be converted to a closed loop spline by dragging one endpoint onto another, but a closed loop spline cannot be made open except by trimming.
- **Spline Points**
  Most splines use one or more interpolant spline points between the endpoints. Spline points can be added (through the right mouse menu) or deleted.

- **Spline Handles**
  Spline handles are used to change the direction and magnitude of the tangency at a spline point or endpoint. Unless a handle is being used to create tangency other than the default settings, they are not visible unless the spline is selected.

  Spline handles at interior spline points can be dragged asymmetrically (handles to opposite sides of the point are independent), or by holding the **Alt** key, the handles will behave symmetrically.

  Spline handles are composed of magnitude and direction handles. The magnitude handle can be dragged in a direction tangent to the spline, and the direction handle can be dragged in a circle around the point to which it is attached. By dragging the dot at the end of the magnitude handle, you can control both magnitude and direction at the same time. Notice that the cursor changes to indicate which control it is over.



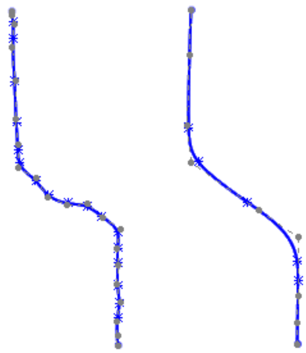
- **Control Polygon**
  The control polygon is the series of dotted lines that go around the spline. It can be used in place of handles to adjust the shape of the spline. To manipulate the control polygon, drag the control points (polygon vertices).

Note: Moving the control polygon will move spline points while moving the spline handles will not.

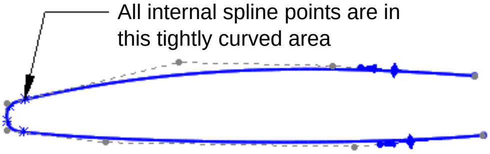
Sketching with Splines

Here are some general guidelines you may find useful for working with splines:

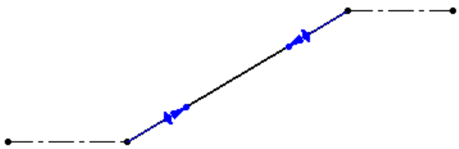
- **Smoothest curves**
  Use as few spline points as possible to give the smoothest curve. Using many spline points usually only works if they are generated by a computer program. Manually tweaking points that are closely spaced can lead to lumpy or uneven splines.



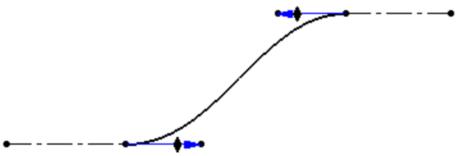
- **Point density**
  You will need more spline points in areas of smaller radius. A long curving area will need relatively fewer points than a tightly curved section.



- **Two-point splines**
  A two-point spline looks just like a straight line unless tangency is applied to the ends, in which case it becomes a very useful and flexible sketch tool. It is particularly useful in situations where a profile must change convexity, which an arc cannot do. Notice that this is much smoother than using a pair of tangent arcs.



Two-point spline without tangency



Two-point spline with tangency

Two tangent arcs

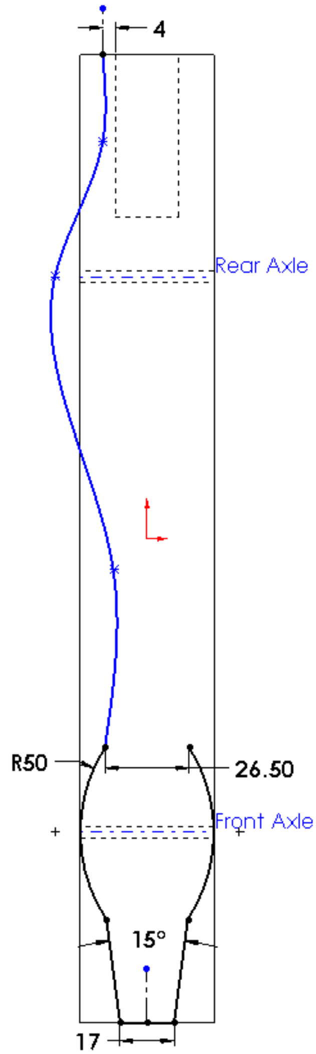
- **Tangency at endpoints**
  By default, splines are created with no tangency at the endpoints. This means that splines tend to be flat or straight at their ends.

Procedure

17 Sketch a spline.

Click **Spline** on the Sketch toolbar, or, click **Tools, Sketch Entities, Spline**.

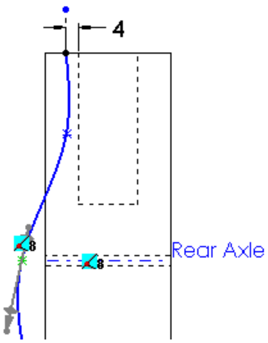
Sketch a spline as shown in the illustration at the right. One end starts at the endpoint of the arc; the other end is at the end of the construction line. You want three internal spline points.



Tip: When you get to the last point in the spline, double-click the mouse the end the spline.

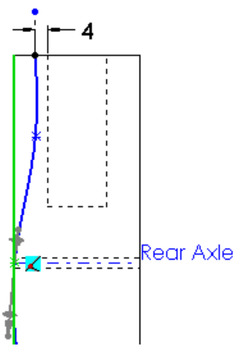
18 Add relation.

Add a **Coincident** relation between the spline point and the axis of the rear axle hole.



19 Add another relation.

Add a second **Coincident** relation between the spline point and the edge of the car body.

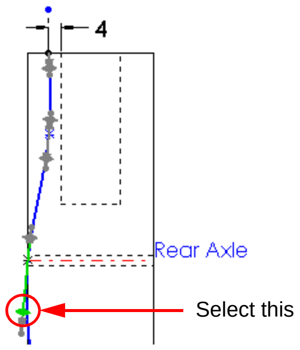


20 Add a relation to spline handle.

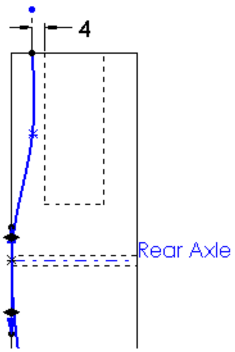
Select the spline. This makes the spline handles appear.

Select the spline handle by clicking the diamond-shaped direction handle.

Add a **Vertical** relation.



Before

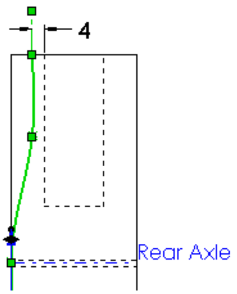


After

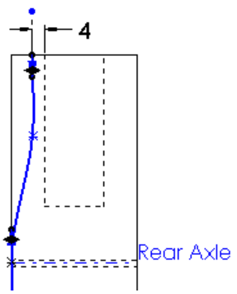
21 Spline tangency.

Press **Ctrl** and select the construction line and the spline.

Add a **Tangent** relation.



Before

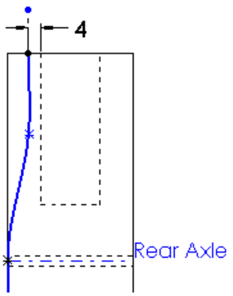


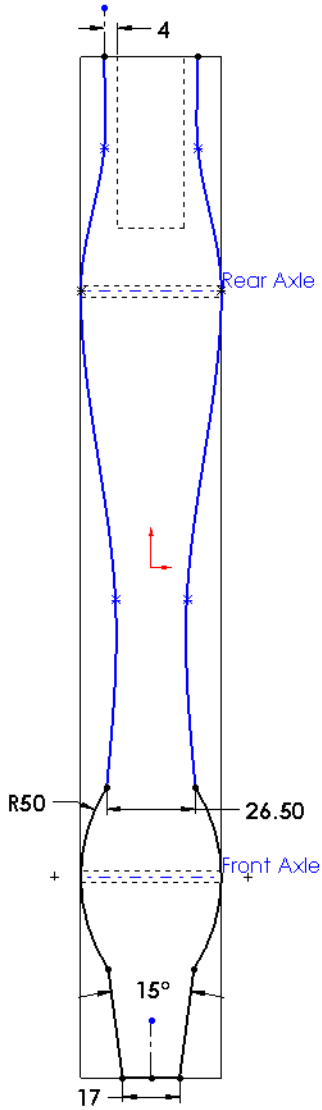
After

Note: You do not need to add any relations to the tangent handles on the other two spline points. Leave them at their default settings.

22 Turn off the display of the spline handles.

Right-click the spline and clear the **Show Spline Handles** option on the short cut menu.





23 Mirror the spline.

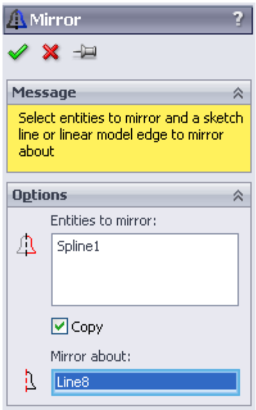
Click **Tools, Sketch Tools, Mirror**, or click **Mirror Entities** on the Sketch toolbar.

For **Entities to mirror**, select the spline.

For **Mirror about**, select the construction line.

Make sure **Copy** is selected. A preview indicates the results of the mirror operation.

Click **OK**.



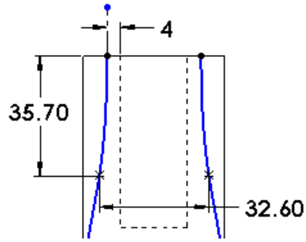
Dimensioning Splines

It is common practice to leave splines under defined. Fully dimensioning splines require two dimensions, or a combination of dimensions and sketch relations, for each spline point. When a spline is fully defined, it is much harder to make changes to it.

However, in this case we *will* fully dimension the splines. Otherwise, the shape of the car you are working on would be different than the one illustrated in the book and your analysis results would be different.

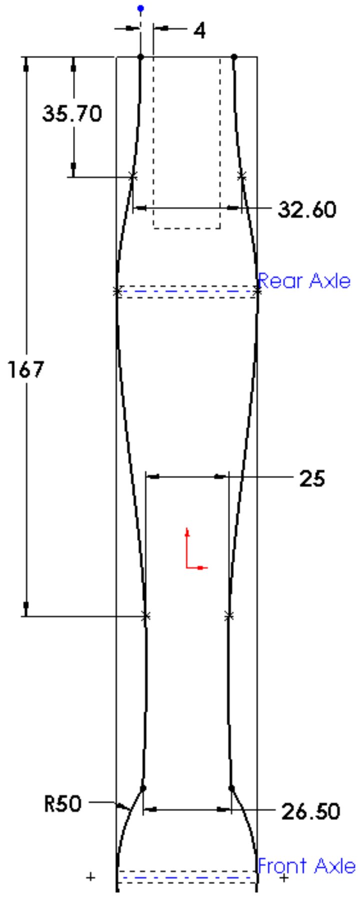
24 Dimension the splines.
Starting at the end of the car where the CO_2 cartridge goes, dimension the splines as shown.

Because of the symmetry imposed by the mirroring operation, you only have to apply the **35.70mm** dimension to one of the splines. The spline point on the other spline will automatically line up.



25 Finish dimensioning the splines.
The spline point at the rear axle is already fully defined by the **Coincident** sketch relations. There is no need to dimension it. In fact, dimensioning it would make the sketch over defined.

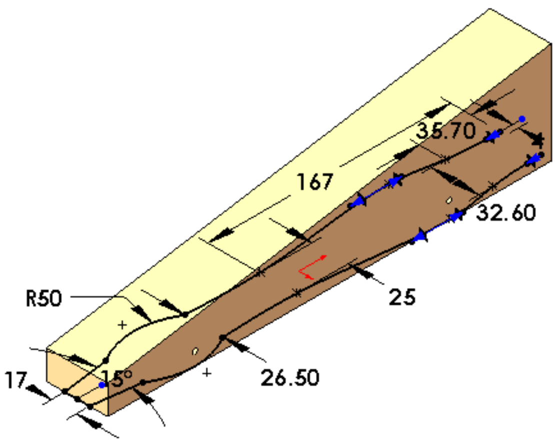
The sketch should now be fully defined.



26 Shaded view.
Click **Shaded with Edges** on the View toolbar.

This will make it easier to see the preview when we extrude a cut feature using the sketch.

27 Isometric view.
Change the view back to the `Isometric` view.

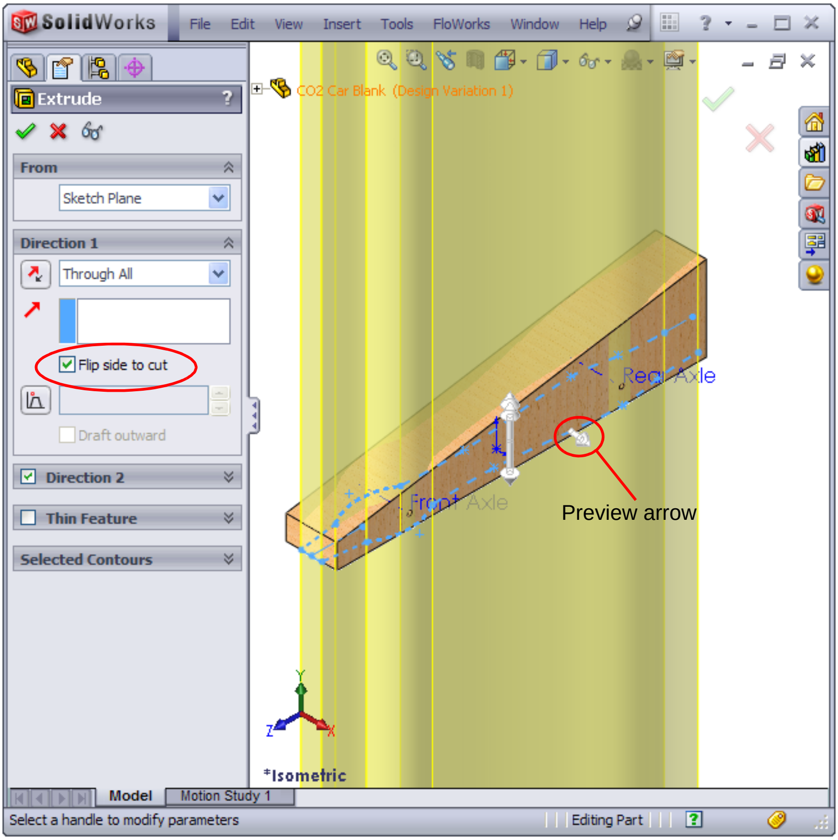


Extruding an Open Contour

The sketch we just completed is an open contour because we did not sketch a line between the endpoints of the two splines.

Extruding an open contour makes the **Extruded Cut** feature behave in a slightly different way than you are used to:

- Both end conditions are automatically set to **Through All**.
- You have to pay attention to which side of the contour is cutting away the solid. A preview arrow in the graphics window points to the side of the contour where the material will be removed. To change the direction either click the arrow, or click **Flip side to cut** in the PropertyManager.



28 Extrude a cut.

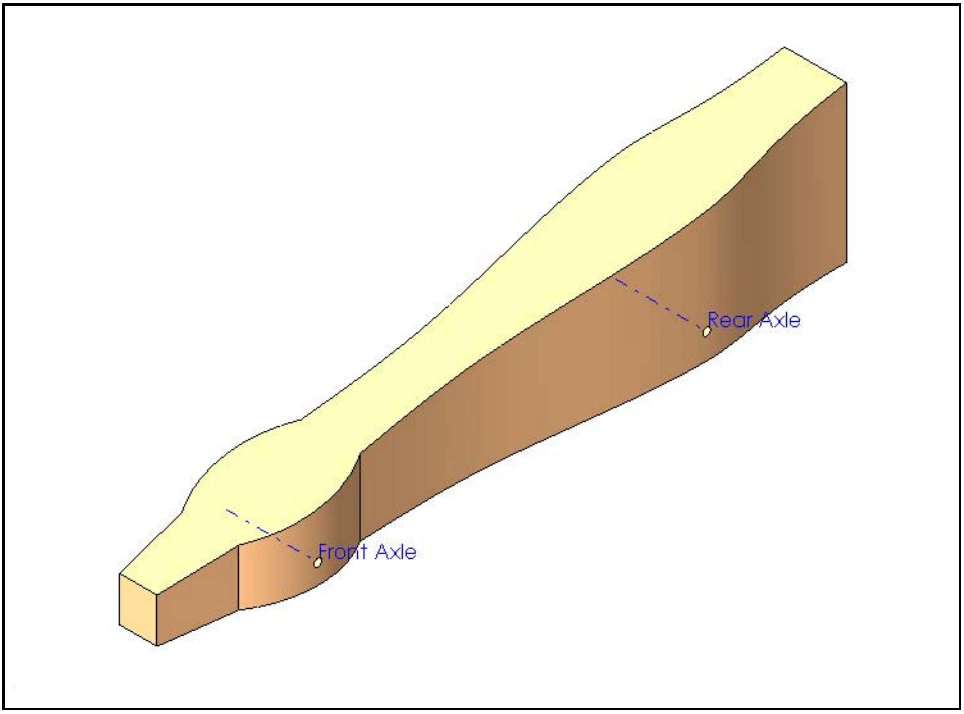
Click **Extruded Cut** on the Features toolbar, or click **Insert, Cut, Extrude**.

Click **Flip side to cut** so the material *outside* the contour is removed.

Click **OK**.

29 Results.

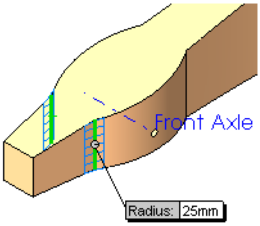
The results are shown below.



30 Add fillet.

Click **Fillet** and set the **Radius** to **25mm** and select the edges as shown.

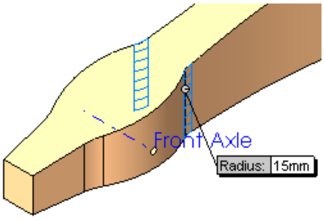
Click **OK**.



31 Add another fillet.

Click **Fillet** again and set the **Radius** to **15mm** and select the edges as shown.

Click **OK**.



32 Save your work.

Click **File, Save**, or click **Save** on the Standard toolbar.

Tip: It is always good practice to save your work after you have created something you want to keep, or just before you try something you aren't sure is going to work.

33 Create a new sketch.

Select the `Right` reference plane and click **Sketch**.

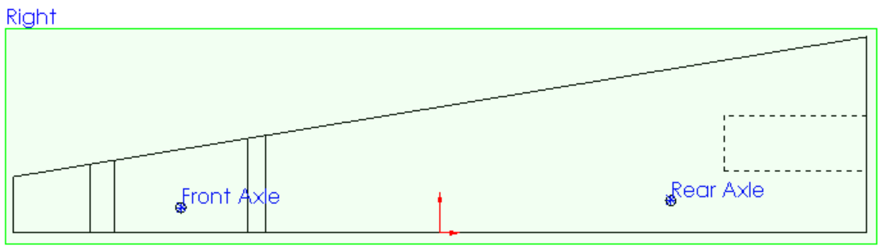
34 Show hidden lines.

Click **Hidden Lines Visible** on the View toolbar.

This enables you see the location of `Power Plant Chamber` (the hole for the CO_2 cartridge) so you can maintain the minimum thickness surrounding it as required by the specifications.

35 Right view orientation.

Click **Right** on the Standard Views toolbar to change to the `Right` view orientation.



36 Construction lines.

Click **Tools, Sketch Entities, Centerline**, or click **Centerline** on the Sketch toolbar.

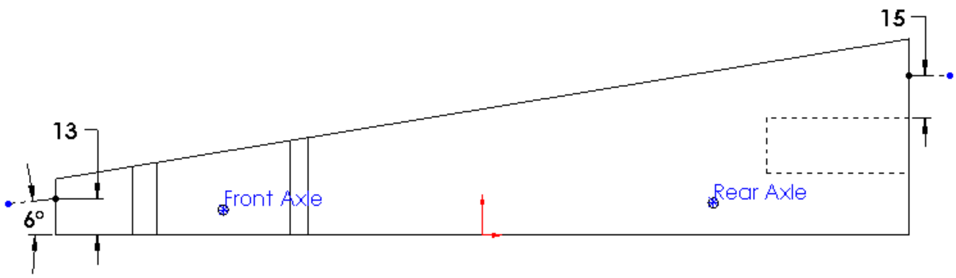
Sketch a horizontal construction line extending from the rear of the car body as shown below.

Sketch a second construction line extending from the front of the car body. This construction line is not horizontal. Rather is extends downward at an angle.

Note: Make sure you to not snap to the midpoint of the front edge of the car.

Click **Smart Dimension** on the Sketch toolbar, or click **Tools, Dimensions, Smart**.

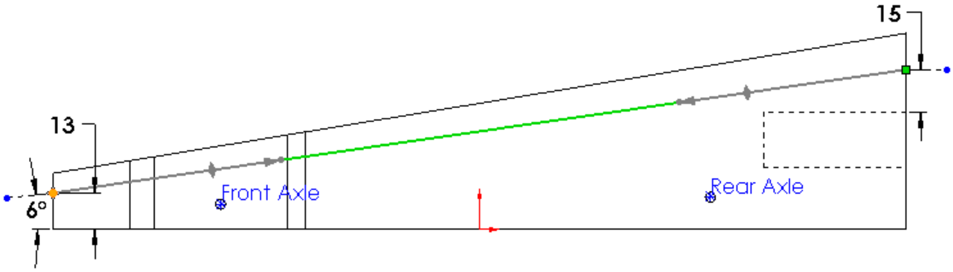
Dimension the construction lines as shown in the illustration below.



37 Two-point spline.

Click **Spline** on the Sketch toolbar, or click **Tools, Sketch Entities, Spline**.

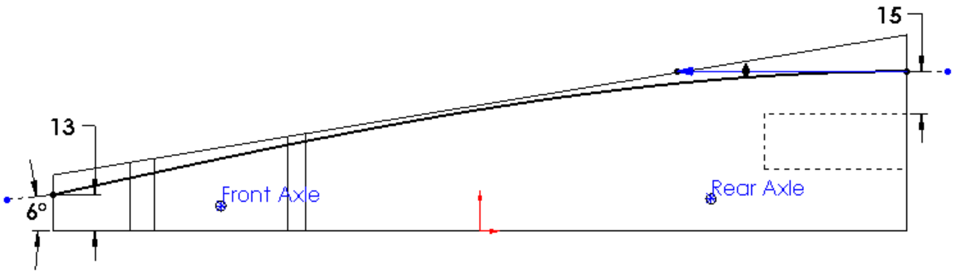
Sketch a two-point spline between the endpoints of the two construction lines. Notice the spline looks exactly like a straight line. That will change when we add relations to the tangency handles.



38 Horizontal relation.

Select the spline. This makes the spline handles appear.

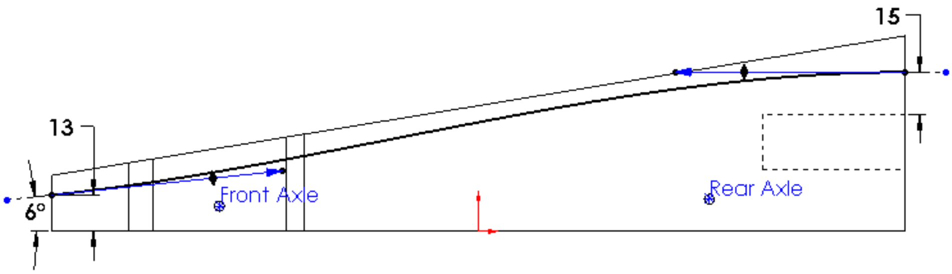
Add a **Horizontal** relation to the spline handle at the rear of the car. Notice how this makes the spline curve.



39 Tangent relation.

Press **Ctrl** and select the construction line at the front of the car body and the spline.

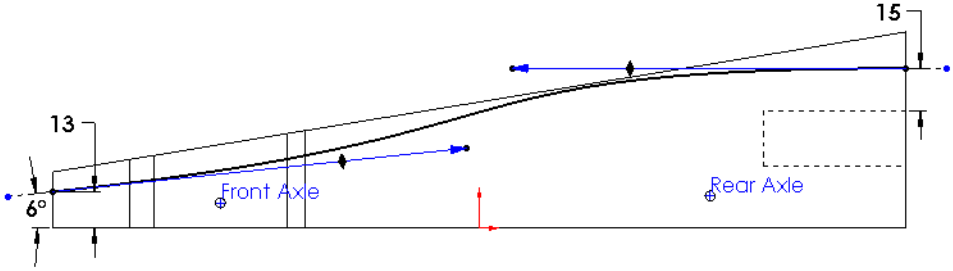
Add a **Tangent** relation.



40 Adjust the length of the spline handles.

Making the spline handles longer increases the effect they have on the shape of the spline.

Experiment by dragging the spline handles. Notice how the shape of the spline changes.

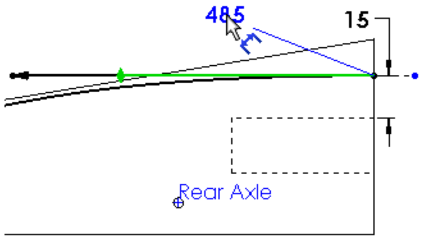


While this is very interactive, it is not precise. There are two ways to precisely control the length of the spline handles:

- Dimension them
- Specify their length in the PropertyManager

41 Dimension the spline handles.

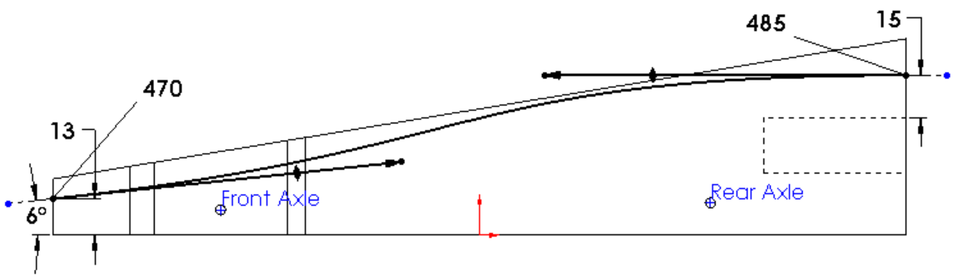
Click **Smart Dimension** on the Sketch toolbar, or click **Tools, Dimensions, Smart**.



Select the diamond-shaped direction handle at the rear of the car body. Then click in the blank area of the graphics window to place the dimension.

Set the value to **485**.

Repeat this process for the tangent handle at the front of the car body making the length value **470**.

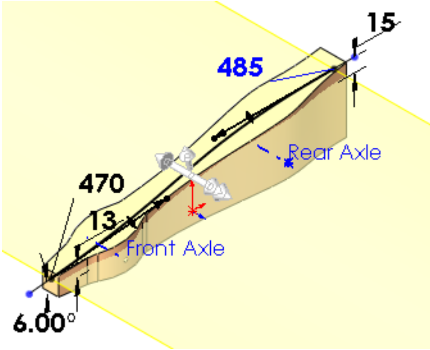


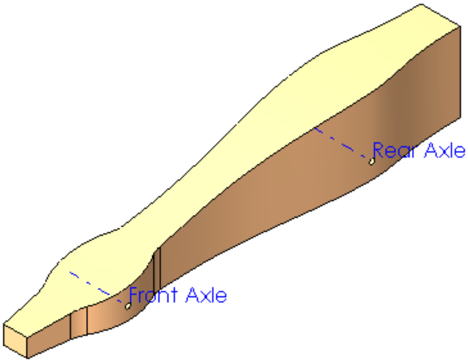
42 Extrude a cut.
Click **Extruded Cut** on the Features toolbar, or click **Insert, Cut, Extrude**.

Verify that the cut is removing material from the correct side.

Click **OK**.

The results are shown below.



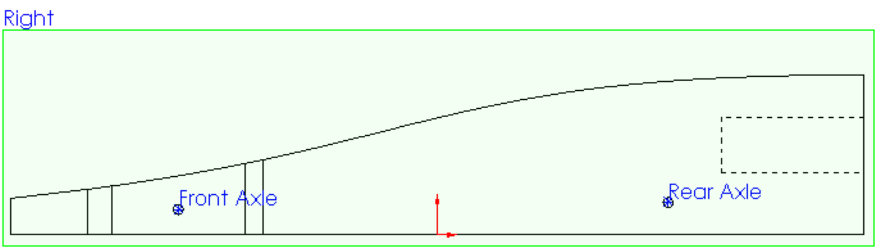


Modify the Underside of the Car

43 Create a new sketch.
Select the `Right` reference plane and click **Sketch**.

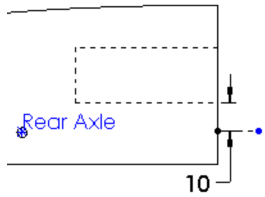
44 View setup.
Click **Hidden Lines Visible** on the View toolbar.

Click **Right** on the Standard Views toolbar to change to the `Right` view orientation.



45 Construction line.

Click **Tools, Sketch Entities, Centerline**, or click **Centerline** on the Sketch toolbar.



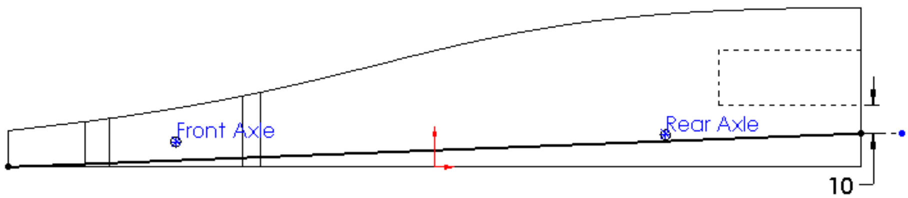
Sketch a horizontal construction line extending from the rear of the car body as shown below.

Dimension the construction line to be **10mm** from the edge of the hole for the CO_2 cartridge.

46 Two-point spline.

Click **Spline** on the Sketch toolbar, or click **Tools, Sketch Entities, Spline**.

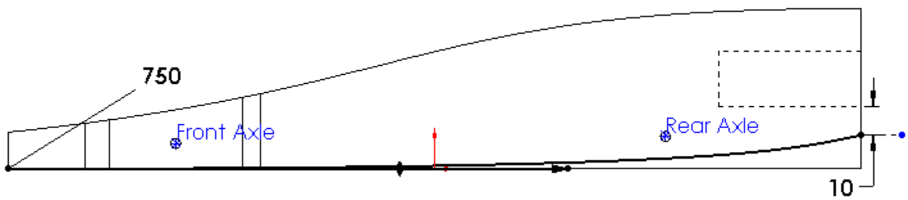
Sketch a two-point spline between the endpoint of the construction line and the front corner of the car body.



47 Tangent relation.

Add a **Tangent** relation between the spline and the bottom edge of the car body.

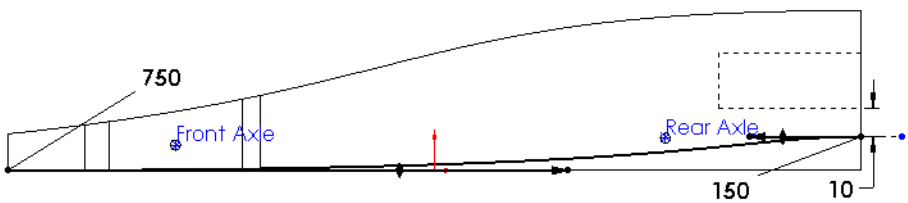
Dimension the length of the handle to be **750**.



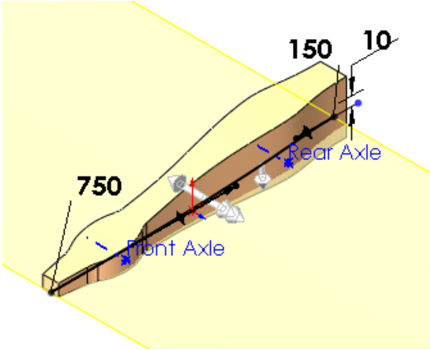
48 Horizontal relation.

Add a **Horizontal** relation to the spline handle at the rear of the car.

Dimension the length of the handle to be **150**.



49 Extrude a cut.

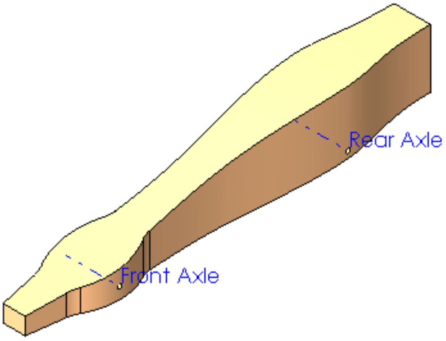


Click **Extruded Cut** on the Features toolbar, or click **Insert, Cut, Extrude**.

Verify that the cut is removing material from the correct side.

Click **OK**.

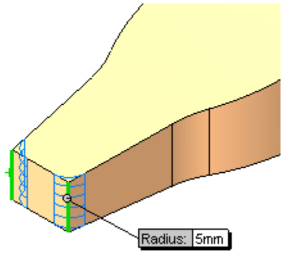
The results are shown below.



50 Fillet.

Click **Fillet** and set the **Radius** to **5mm** and select the edges as shown.

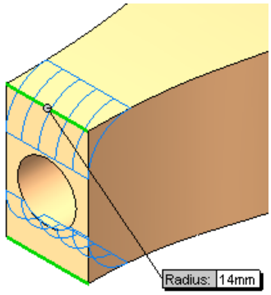
Click **OK**.



51 Fillet.

Click **Fillet** and set the **Radius** to **14mm** and select the edges as shown.

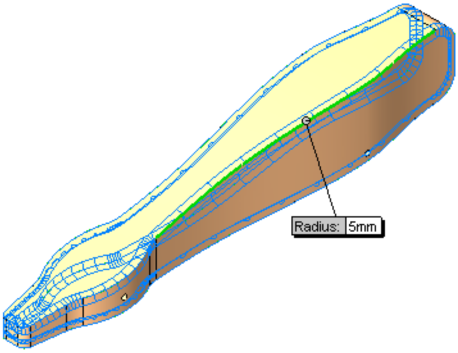
Click **OK**.



52 Fillet.

Click **Fillet** and set the **Radius** to **5mm** and select the edges as shown.

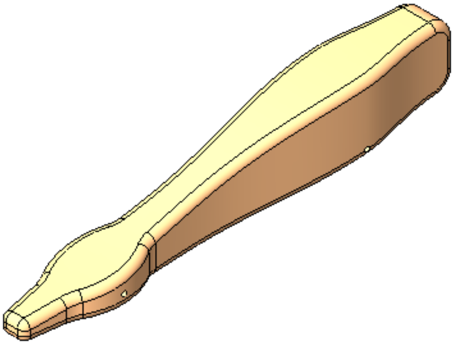
Click **OK**.



Tip: The option **Tangent propagation** in the **Fillet** command will select a chain of connected edges provided they are tangent to each other. This simplifies creating this fillet because you don't have to manually select all the edges.

53 Complete.

The results of the design changes are shown at the right.



Reduction in Mass

What is the mass of the new body design? Remember, not only do we want to improve the aerodynamics, we want to reduce the mass.

1 Mass properties calculations.

What is the mass of the new body design?

Click **Tools, Mass Properties**, or click **Mass Properties** on the Tools toolbar.

The mass is 51.17 grams. How does that compare to the mass of the out-of-the-box blank?

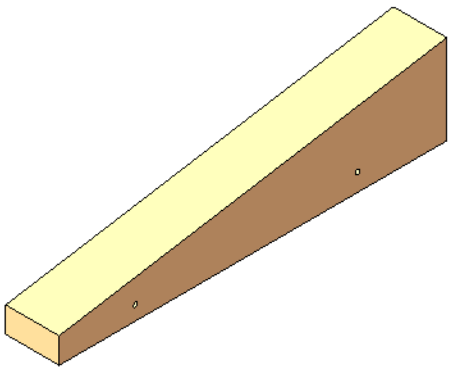
Most likely, this is not the exact mass of your design. Why is that?

2 Switch configurations.

In the ConfigurationManager, double-click the configuration named `Default`.

Click **Tools, Mass Properties**, or click **Mass Properties** on the Tools toolbar.

The mass is 89.53 grams.



Percentage Improvement

To find the percentage of improvement use this formula:

$$\left(\frac{InitialValue - FinalValue}{InitialValue}\right) \times 100 = PercentageChange$$

$$\frac{89.53 - 51.17}{89.53} \times 100 = 42.85$$ The change yielded about a 43% reduction in mass.

3 Save and close the part file.

Assembly Configurations

We have created a configuration in the part. Next we will create a configuration in the assembly to show the car before and after the changes.

1 Reopen the assembly.
When you reopen the assembly, the latest version of the car will not be referenced; all of the changes you just made will not be present.

Note: If you did *not* close the assembly at the end of Lesson 3, you will get a message when the assembly window become visible. The changes to the car are detected and SolidWorks asks if you want to rebuild the assembly.

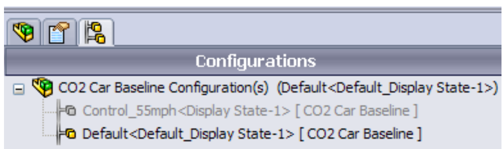
`Models contained within the assembly have changed. Would you like to rebuild the assembly now?`

Click **Yes**.

2 Switch to the ConfigurationManager.
Click the ConfigurationManager tab to change to the ConfigurationManager.

The active configuration is the one we created when doing the initial Flow Simulation analysis: `Control-55mph`.

3 Switch back to the default configuration.
Position the cursor over the `Default` configuration and double-click it to make it active.



The Flow Simulation analysis tree tab disappears because the default configuration does not have any analysis data associated with it.

4 Add a new configuration.

Right-click on the file name and select **Add Configuration**.

The **Add Configuration** PropertyManager appears.

For **Configuration name**, type the name `Car Design Version 1`.

Expand the **Advanced Options** list.

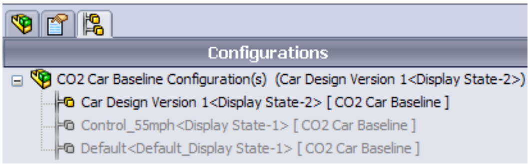
Click the **Suppress new components** option.

When this option is selected, new components added to *other* configurations are suppressed in *this* configuration.

Click **OK**.

5 Results.

A new configuration is added to the assembly.



However, the new configuration is identical to the `Default` configuration.

We need to change it so that this configuration shows the modified car body.

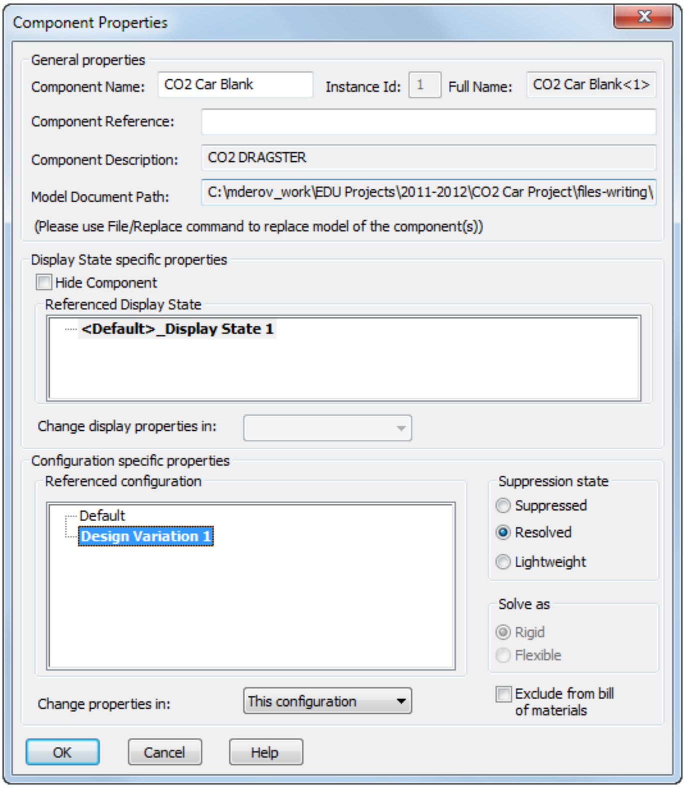
6 Referencing a different configuration.

Switch back to the FeatureManager design tree.

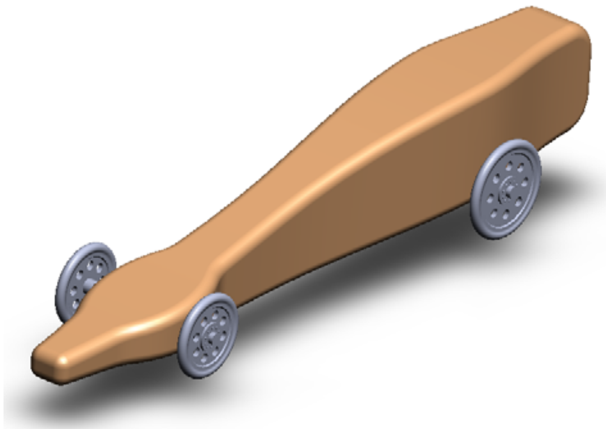
Right-click on the `CO2 Car Blank` component in the FeatureManager design tree and select **Component Properties**.

In the **Referenced configuration** area, select `Design Variation 1`.

Click **OK**.



7 Results.
The assembly now shows the modified design of the car body.



Lesson 5
Analyzing the Modified Design

When you complete this lesson, you will be able to:

- Clone a Flow Simulation project thus creating a template that can be used throughout the design process;
- Create assembly
- Reanalyze the model.

Analyze the Modified Design with Flow Simulation

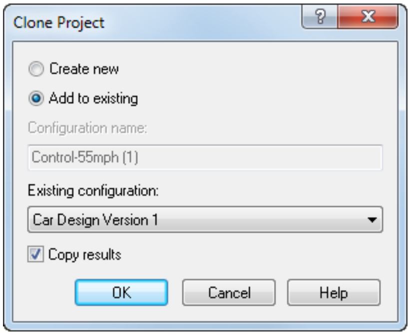
The easiest way to redo the analysis is to clone the Flow Simulation project we created for the initial design. This way we don't have to repeat the work of adding the goals, defining the computational domain, and adding the various results plots.

1 Activate the analysis configuration `Control-55mph`.
In the ConfigurationManager, double-click the configuration named `Control-55mph`. This is the fastest and easiest way to switch between configurations.

The Flow Simulation analysis tree tab reappears.

2 Switch to the analysis tree.
Click the Flow Simulation analysis tree tab to access the analysis features.

3 Clone project.
Right-click the uppermost feature, `Control-55mph`, and select **Clone Project** from the shortcut menu.



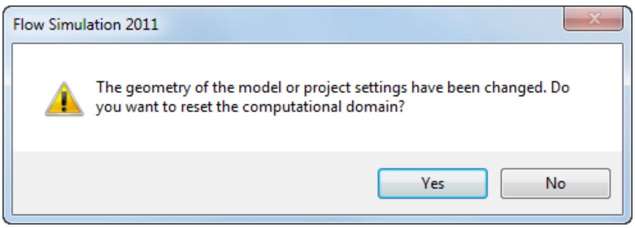
In the **Clone Project** dialog box, click **Add to existing**.

From the **Existing configurations** list, select `Car Design Version 1`.

Select the **Copy results** option and click **OK**.

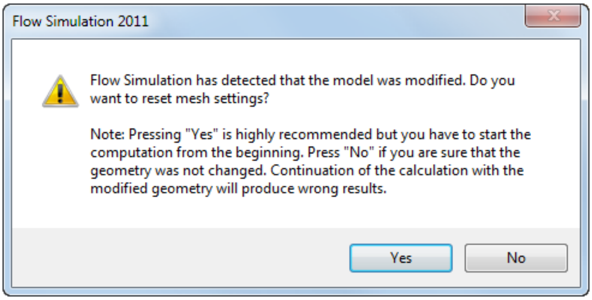
4 Messages.
When you click **OK**, you will get a couple of messages:

- **Computational Domain**



The system will ask you if you want to reset the computational domain. Click **No**. To make it easier to do meaningful comparisons between the two sets of results, we want to use the same size computational domain. Also, resetting the domain would require us to redefine the symmetry conditions. That would be extra work.

- **Mesh Settings**



The geometry of the car body has changed. We've rounded the nose and made other changes. The mesh should be reset.

Click **Yes**.

5 Run the solver.
In the Flow Simulation analysis tree , right-click the uppermost feature, `Car Design Version 1`, and select **Run** from the shortcut menu.

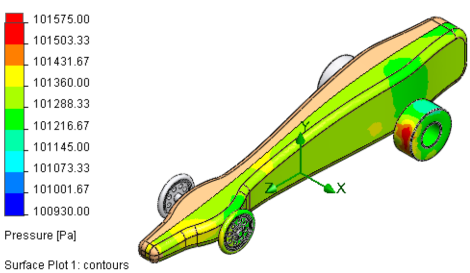
Examine the Results

1 Load the results.
If the results were not loaded automatically, right-click `Results` in the Flow Simulation analysis tree and select **Load Results** from the shortcut menu.

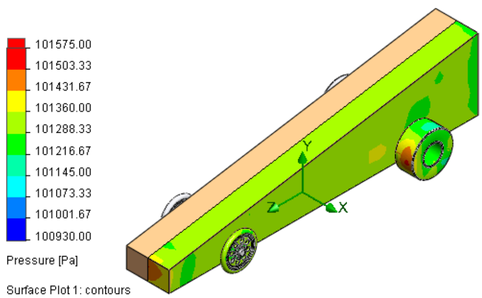
2 Show the surface plot.
Expand `Surface Plots`.

Right-click `Surface Plot 1`, and select **Show**.

3 Surface plot results.
For comparison purposes in this lesson, we have also shown the surface plot for the initial design.



Changed Design



Baseline Design

q

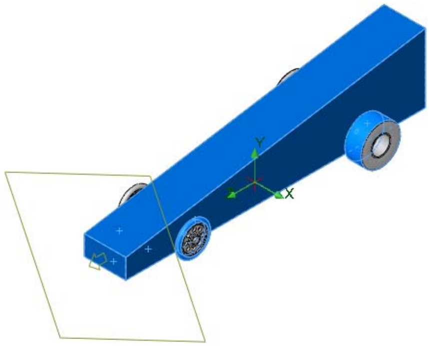
The drag force is equal to the pressure multiplied by the area. You can see in the surface plots of the two designs that rounding off the nose of the body results in a much smaller area of high pressure. This means we have reduced the drag force on the body of the car. However, we have areas of high pressure on the front portions of the wheels, particularly the rear wheels. We will discuss this in more detail later in this lesson.

Flow Trajectories

Now let's look at the flow trajectories.

4 Hide the surface plot.
Right-click `Surface Plot 1` and select **Hide**.

5 Missing face for the first set of flow trajectories.
The first set of flow trajectories referenced faces of the car body don't exist in this configuration. They were eliminated when the cut features and fillets were applied to the body. Therefore, we must redefine the reference before we can display the plot.

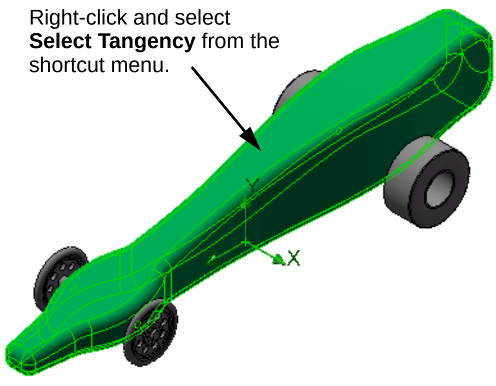


6 Edit definition.
Right-click `Flow Trajectories 1` and select **Edit Definition**.

For **Reference**, select the faces of the car by following this procedure:

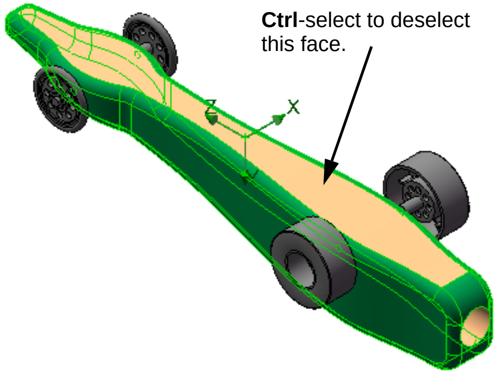
- In the graphics area, right-click one of the faces of the car body and select **Select tangency** from the shortcut menu.

  This selects *all* the tangent faces on the car body, including the bottom. The flow trajectories plot for the initial design did not include the bottom of the car. So, to keep the comparison valid, we need to deselect the bottom face.



- Press and hold the **Shift** key. Press the up arrow on the keyboard twice. This flips the view over 180° so you can see the bottom.

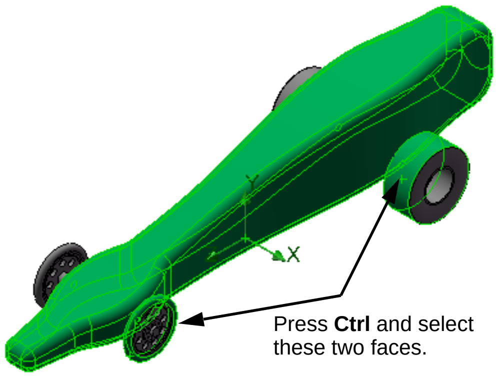
  Now press **Ctrl** and click the bottom face of the car, deselecting it.



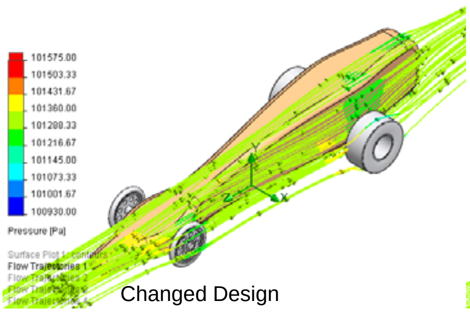
- Press and hold the **Shift** key and press the up arrow twice again. This flips the view another 180°, back to its original orientation.

  Now we have to include the faces tread of the wheels.

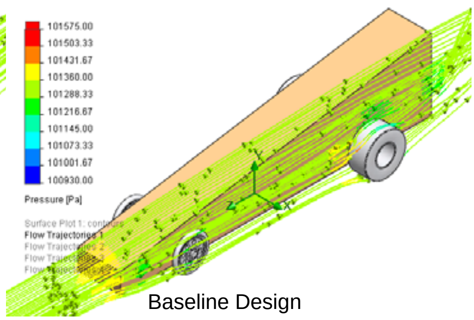
  Press **Ctrl** and select the faces that form the tread portion of the front and rear wheels.



Click **OK**.



Changed Design

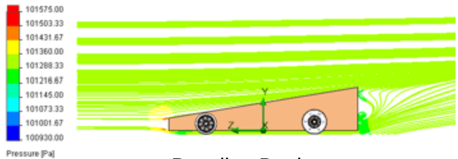


Baseline Design

7 Display the other flow trajectories.

Display the flow trajectory plot that references the `Right` plane.

Changed Design

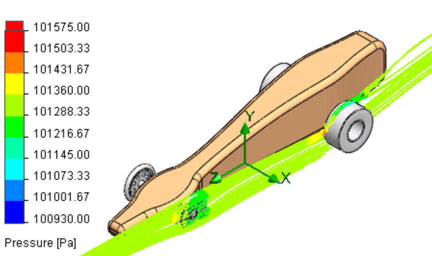


Baseline Design

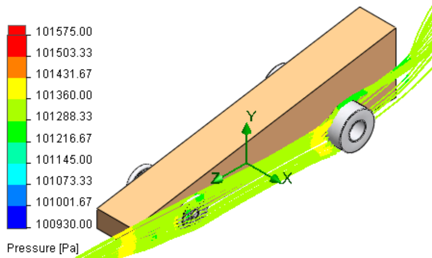
Notice the reduction in turbulence behind the car in the modified design.

8 Flow trajectory for front wheel.

Look closely at how the flow trajectories interact with the rear wheel. It appears the change in the shape of the car body is causing increased drag on the rear wheel. It is as if the car body's shape is deflecting the air flow so it hits the rear wheel more directly. We will explore this in more detail later in this lesson.



Changed Design

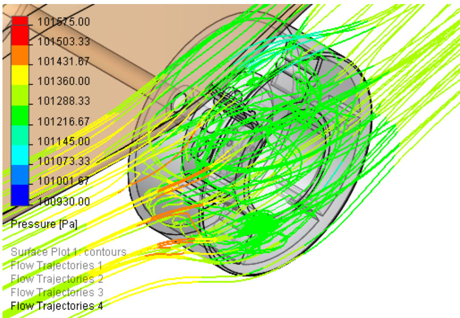


Baseline Design

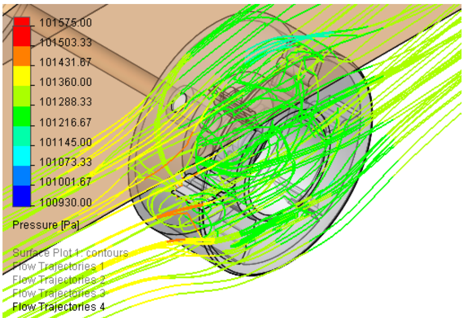
9 Flow trajectory for rear wheel.

If you don't remember how to turn on transparency, review the procedure in step **16** on page 59.

Looking at the flow trajectories, it appears the modified car body is causing even more turbulence and eddies inside the rear wheel.



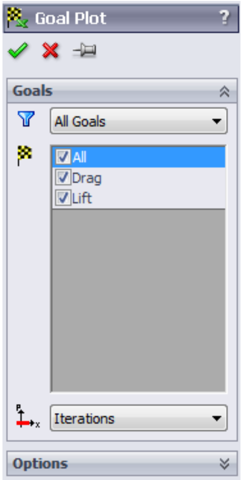
Changed Design



Baseline Design

Quantitative Results

The surface plots and flow trajectories don't give us the full story. It appears that overall the new design is more aerodynamic, but we don't know how much of an improvement we've achieved. To take a more quantitative approach we will first look at the goals.



1 Create a goals plot.
In the analysis tree, expand the `Results` listing and the `Goals` listing.

Double-click `Goals Plot 1`.

The **Goals** dialog box appears.

Check that both **Drag** and **Lift** are selected.

If they are not, click **All**.

Click **OK**.

2 Excel spreadsheet.
Microsoft® Excel is launched and a spreadsheet opens.

Note: To reduce the size of the image and make it more readable, we are only showing the first three columns, which are the only ones we are interested in.

CO2 Car Baseline.SLDASM [Car Design Version 1]

Goal Name	Unit	Value
Drag	[p]	-23.94611392
Lift	[p]	-1.073978929

Iterations: 77
Analysis interval: 26

CO2 Car Baseline.SLDASM [Control-55mph]

Goal Name	Unit	Value
Drag	[p]	-31.80078735
Lift	[p]	-4.383855227

Iterations: 80
Analysis interval: 25

The drag value for the new design is 23.95 grams-force. The drag value for the original design is 31.80 grams-force.

Percentage Improvement

To find the percentage of improvement use this formula:

$$\left(\frac{InitialValue - FinalValue}{InitialValue}\right) \times 100 = PercentageChange$$

For simplicity we will round to 2 decimal places. Substituting we get:

$\frac{(31.80 - 23.95)}{31.80} \times 100 = 24.69$ The changes yielded about a 24.69% improvement.

What About Lift?

It is interesting to note that the original design had a *downward* lift force of approximately 4.38 grams-force. The modified design has an nearly zero lift force. In fact, with further modifications, this lift force could change sign and become *upward*. As long as this force is less than the weight of the car (which it is), it could be beneficial. This type of car does not rely on traction for acceleration and an upward lift force may reduce rolling friction between the wheels and the track.

CO2 Car Baseline.SLDASM [Car Design Version 1]

Goal Name	Unit	Value
Drag	[p]	-23.94611392
Lift	[p]	-1.073978929

Iterations: 77
Analysis interval: 26

CO2 Car Baseline.SLDASM [Control-55mph]

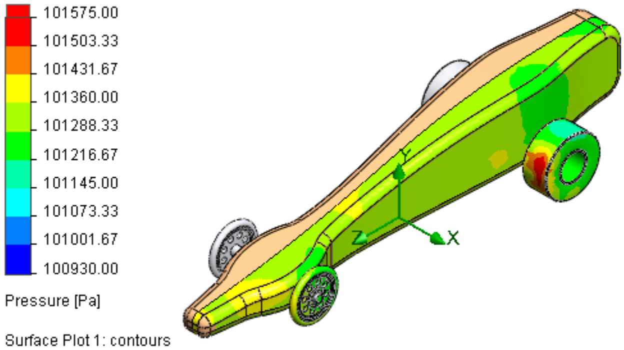
Goal Name	Unit	Value
Drag	[p]	-31.80078735
Lift	[p]	-4.383855227

Iterations: 80
Analysis interval: 25

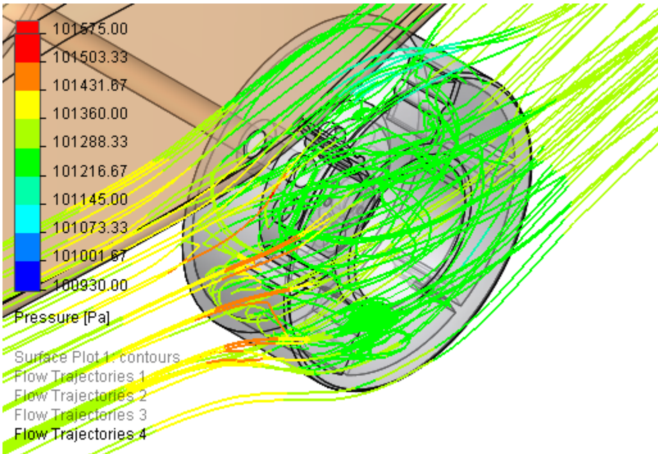
Of course, if the lift force is greater than the weight of the car, then the car will tend to become airborne.

Why Didn't We See a Greater Reduction in Drag?

Looking at the wheels in the surface plot gives us a clue. The red color in the surface plot indicates that there are areas of high pressure on the leading portions of the front and rear wheels, particularly the rear wheels.



The flow trajectories also provide a clue. There is a significant amount of turbulence surrounding the rear wheels.



Let's look at this data quantitatively.

Surface Parameters

Surface Parameters are results that allow you to examine the forces on selected faces in the model. We will create a report for the rear wheels of the assembly because they appear to be having the biggest impact on the aerodynamics.

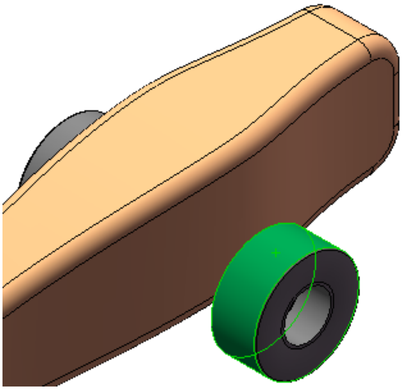
3 Insert surface parameters.
In the Flow Simulation analysis tree, expand the `Results` listing.

Right-click `Surface Parameters` and select **Insert** from the shortcut menu.

Select the face that represents the tread of the rear wheel.

Select **All** under the **Parameters** list.

Click **Export to Excel**.



4 Excel spreadsheet.
An Excel spreadsheet opens.

Surface Parameters 1[Car Design Version 1]

Face<1> <Rear Wheel-1@Revolve1>

Global Coordinate System

Iteration [ ] 77

Local parameters

Parameter	Minimum	Maximum	Average	Bulk Average	Surface Area [m^2]
Pressure [Pa]	101120.052	101706.73	101313.099		0.001528992
Density [kg/m^3]	1.20063573	1.2080885	1.20308439		0.001528992
Velocity [mile/h]	0	0	0		0.001528992
X - Component of Velocity [mile/h]	0	0	0		0.001528992
Y - Component of Velocity [mile/h]	0	0	0		0.001528992
Z - Component of Velocity [mile/h]	0	0	0		0.001528992
Mach Number [ ]	0	0	0		0.001528992
Heat Transfer Coefficient [W/m^2/K]	0	0	0		0.001528992
Shear Stress [Pa]	0	7.18629565	1.52149944		0.001528992
Surface Heat Flux [W/m^2]	0	0	0		0.001528992
Fluid Temperature [K]	293.061754	293.513865	293.3126		0.001528992

Integral parameters

Parameter	Value	X-component	Y-component	Z-component	Surface Area [m^2]
Heat Transfer Rate [W]	0				0.001528992
Normal Force [p]	10.6807547	-0.270537514	2.13285486	-10.4621346	0.001528992
Shear Force [p]	0.176039005	0.015450754	0.023923231	-0.173720133	0.001528992
Force [p]	10.8553289	-0.25508676	2.15677809	-10.6358547	0.001528992
Torque [N*m]	0.004452503	0.000282151	0.004355704	0.000879213	0.001528992
Surface Area [m^2]	0.001528992	-5.69846E-05	2.31627E-05	-2.06194E-06	0.001528992
Torque of Normal Force [N*m]	0.004394965	0.000285784	0.00429792	0.000872887	0.001528992
Torque of Shear Force [N*m]	5.82432E-05	-3.63346E-06	5.77845E-05	6.3263E-06	0.001528992
Uniformity Index [ ]	1				0.001528992
CAD Fluid Area [m^2]	0.002067829				0.002067829

The information we are interested in is in the section labeled *Integral Parameters*. Specifically we want the Z-component of the Force [p].

For this example it is 10.636 grams-force.

What Does This Tell Us?

About 10.64 grams-force of drag is attributable to the just the rear wheels. Comparing that to the total drag force of 23.95 grams-force, we see that the rear wheels are a major contributor to drag. In fact, they represent over 44% of the drag! $(10.64 \div 23.95) \times 100 = 44$

Note: In the note on page 61, we said that to find the total drag on the car you would have to double the drag force values because we used symmetry during the calculations. Why didn't we double the values for this comparison? The answer is we displayed the surface parameters for only one rear wheel. That's half of the number of rear wheels. Since the total drag force data is based on analyzing half of the car (symmetry) and we are calculating the relative contribution of half of the rear wheels, the proportions are correct.

5 Initial design.

In step **3** on page 91 we speculated that the modified shape of the car body might have increased the air flow to the rear wheels, thereby increasing their contribution to the drag. Let's see if that is actually the case.

Switch to the ConfigurationManager and activate the configuration named `Control-55mph`.

Insert the surface parameters on the face of the rear wheel following the procedure described in step **3** on page 98. Although we won't show the spreadsheet here, the Z-component of the force is 9.85 grams-force. This is very close to the 10.64 grams-force we got in the modified design, therefore a significant improvement is not seen in the drag on the rear wheels.

Did Changing the Body Really Help?

We have already seen that redesigning the body gave us nearly a 25% reduction in drag. Since the wheels represent such a significant portion of the total drag, we must have made a major improvement in the body. Without running an analysis on just the body in its original and redesigned configurations, we can make some approximations using the data we already have.

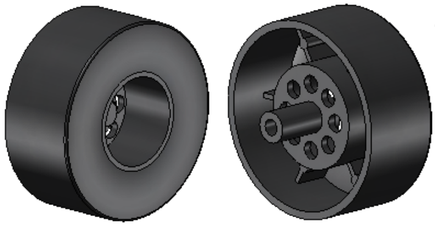
If we switch back to the original design and display the surface parameters for the front and rear wheels we get the following values as shown in the table below:

Drag Source	Initial Design	Modified Design
Entire Model	31.80	23.95
Rear Wheel	9.85	10.64
Approximate amount attributable to body: (Entire Model - Rear Wheels)	21.95	13.31

Once again, we can calculate the percentage reduction in the drag force on the body:

$\frac{(21.95 - 13.31)}{21.95} \times 100 = \ 39.36$ or about 39%.

Note: This is only an approximation because there are a number of factors we did not take into account. For example, the drag force on the wheels was estimated using only the large surface that represents the tread. We did not include the other faces on the wheel, particularly the inside faces that seem to contribute so much to the turbulence.



So yes, changing the body had a major impact on the aerodynamics. It resulted in approximately a 39% reduction in drag on the body. And even with the wheels contributing so much drag, it resulted in about a 25% reduction in drag on the car as a whole.

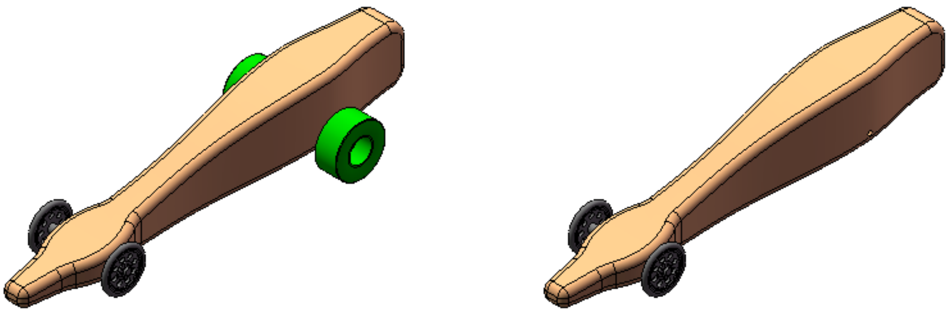
Replace the Rear Wheels

We are going to replace the rear wheels with narrower ones to see what impact it has on the overall drag forces.

1 **New configuration.**
In the ConfigurationManager, double-click the configuration named `Car Design Version 1`. Create a new configuration named `Narrow Rear Wheels`.

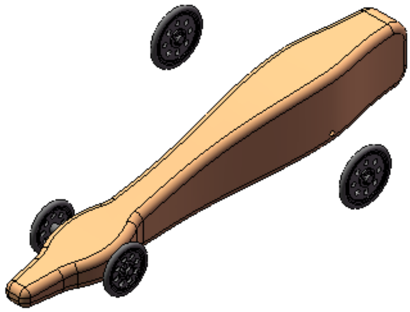
2 **Suppress the rear wheels.**
Select the two rear wheels either in the graphics window or the FeatureManager design tree.

Click **Edit, Suppress, This configuration**.

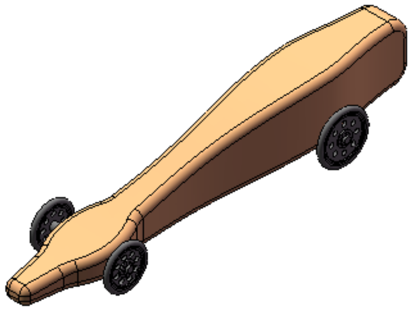


The wheels along with their mates are suppressed. They aren’t deleted from the assembly, they are just “turned off”. They will not be included in any system calculations.

3 Add the narrow wheels.
Drag and drop two instances of the `Narrow Wheel` from the Design Library window into the assembly window.



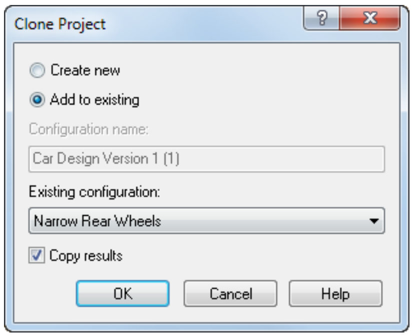
4 Mate the wheels.
Following the same procedure you used in Lesson 2 starting on page 24, mate the narrow wheels to the car body using a combination of **Concentric** and **Distance** (21.25mm) mates.



5 Activate the analysis configuration `Car Design Version 1`.
In the ConfigurationManager, double-click the configuration named `Car Design Version 1`.

6 Switch to the analysis tree.
Click the Flow Simulation analysis tree tab to access the analysis features.

7 Clone project.
Right-click the uppermost feature, `Car Design Version 1`, and select **Clone Project** from the shortcut menu.



In the **Clone Project** dialog box, click **Add to existing**.

From the **Existing configurations** list, select `Narrow Rear Wheels`.

Select the **Copy results** option and click **OK**.

8 Messages.
When you click **OK**, you will get a couple of messages.

Click **No** when asked if you want to reset the computational domain.

Click **Yes** when asked if you want to reset the mesh.

9 Run the solver.
In the Flow Simulation analysis tree, right-click the uppermost feature, `Narrow Rear Wheels`, and select **Run** from the shortcut menu.

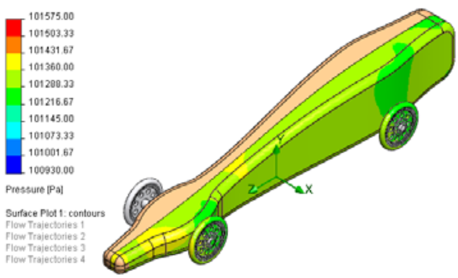
Examine the Results

1. **Load the results.**
   If the results were not loaded automatically, right-click `Results` in the Flow Simulation analysis tree and select **Load Results** from the shortcut menu.

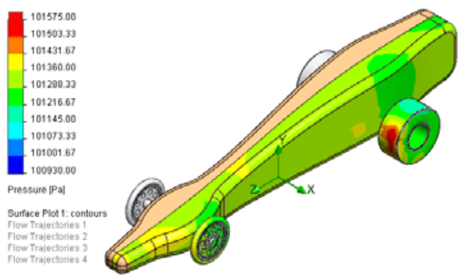
2. **Show the surface plot.**
   Expand `Surface Plots`.

   Right-click `Surface Plot 1`, and select **Show**.

3. **Surface plot results.**
   For comparison purposes, we have also shown the surface plot for the previous design iteration. It looks like replacing the rear wheels had the desired effect.



Narrow Rear Wheel Design



Wide Rear Wheel Design

4. **Display the goals plot.**
   In the analysis tree, expand the `Results` listing and the `Goals` listing.

   Double-click `Goal Plot 1`.

   The **Goals** dialog box appears.

   Check that both **Drag** and **Lift** are selected.

   If they are not, click **All**.

   Click **OK**.

5 Excel spreadsheet.
Microsoft® Excel is launched and a spreadsheet opens.

Note: To reduce the size of the image and make it more readable, we are only showing the first three columns, which are the only ones we are interested in.

CO2 Car Baseline.SLDASM [Narrow Rear Wheels]

Goal Name	Unit	Value
Drag	[p]	-11.9897825
Lift	[p]	-2.585315667

Iterations: 56
Analysis interval: 26

CO2 Car Baseline.SLDASM [Car Design Version 1]

Goal Name	Unit	Value
Drag	[p]	-23.94611392
Lift	[p]	-1.073978929

Iterations: 77
Analysis interval: 26

The drag value for the narrow wheel design is 11.99 grams-force. The drag value for the original, baseline design was 31.70 grams-force and the modified design was 23.95 grams-force.

Percentage Improvement

The table below shows the improvement compared to the original, baseline assembly and the two design iterations:

Design Iteration	Total Drag	Percent Improvement
Original design; out-of-the-box blank	31.70	
Modified body, wide rear wheels	23.95	24.45% compared to baseline
Modified body, narrow rear wheels	11.99	62.18% compared to baseline 49.94% compared to just modifying the body

Just as we predicted, replacing the wide rear wheels with narrow ones had a significant impact on reducing the overall drag.

More to Explore

Using what you have learned, explore some additional design modifications. Or, better yet, start developing your own car body design. Using Flow Simulation as a virtual wind tunnel, you can experiment with many different ideas and approaches before you ever commit to cutting wood.

Browse the Internet for ideas about designing your car. One excellent source is:

http://www.science-of-speed.com

Click on **Showroom**.

With SolidWorks and Flow Simulation together you can easily explore many design variations.

Lesson 6
Making Drawings of the Car

When you complete this lesson, you will be able to:

- Create a B-size drawing of the car body;
- Add different drawing views of parts and assemblies.

Drawings

SolidWorks enables you to easily create drawings of parts and assemblies. These drawings are fully associative with the parts and assemblies they reference. If you change a dimension on the finished drawing, that change propagates back to the model. Likewise, if you change the model, the drawing updates automatically.

Drawings communicate three things about the objects they represent:

- **Shape** – *Views* communicate the shape of an object.
- **Size** – *Dimensions* communicate the size of an object.
- **Other information** – *Notes* communicate nongraphic information about manufacturing processes such as drill, ream, bore, paint, plate, grind, heat treat, remove burrs and so forth.

Creating a Drawing and Views

1 Open the assembly `CO2 Car Baseline`.
Click **File, Open**, or click **Open** on the Standard toolbar.

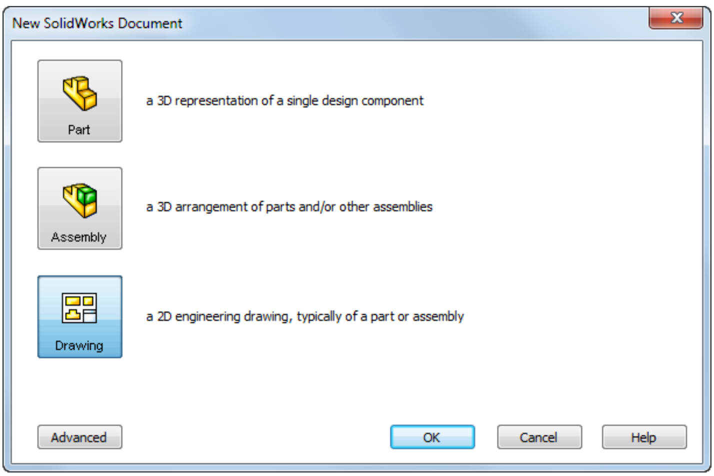
Browse to the folder where you saved the `CO2 Car Baseline` assembly and open it.

2 Open the part `CO2 Car Blank`.
In the graphics window of the assembly, right-click the car body and select **Open Part** from the shortcut menu.

3 Open a new drawing.
Click **File, New**, or click **New** on the Standard toolbar.

The **New SolidWorks Document** dialog box appears.

Select **Drawing** and click **OK**.



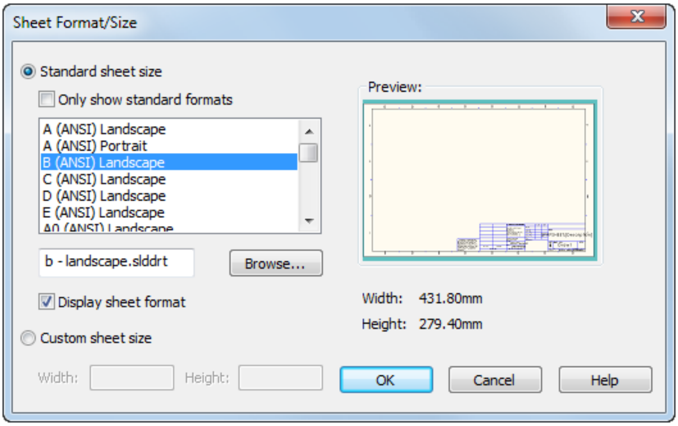
4 Select the sheet size.

The **Sheet Format/Size** window appears.

Deselect **Only show standard formats**.

For **Standard sheet size**, select **B - Landscape**.

Select the **Display sheet format** option and click **OK**.

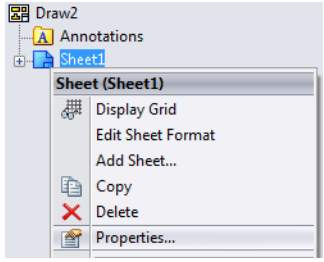


The drawing sheet appears.

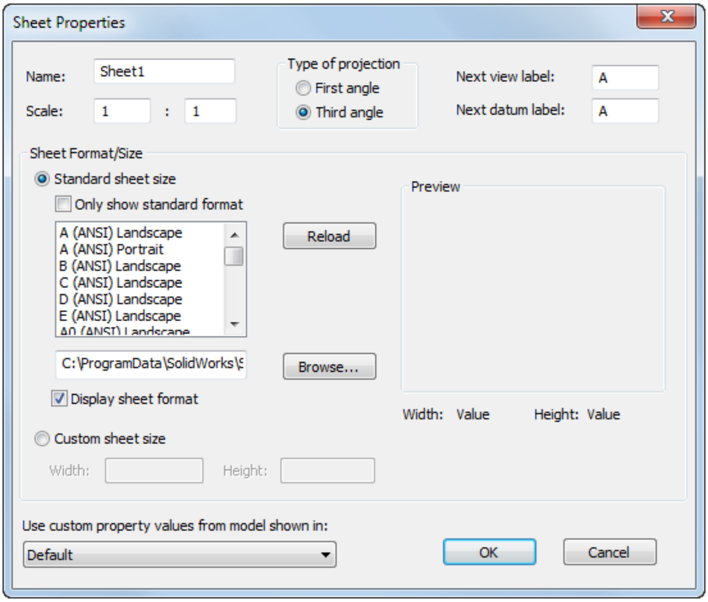
Cancel the **Model View** dialogue to bring you to the drawing's FeatureManager. design tree.

5 Sheet properties.

In the drawing's FeatureManager design tree, right-click `Sheet` and select **Properties** from the shortcut menu.



On the **Sheet Properties** dialog box, under **Type of projection**, select **Third angle** and click **OK**.



6 Setting drawing options.
Click **Tools, Options**, or click **Options** on the Standard toolbar.

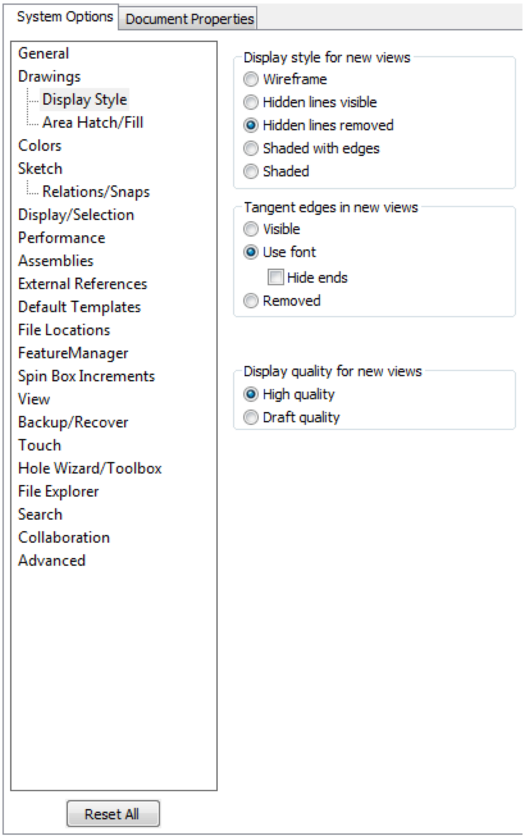
On the **System Options** tab, click **Drawings, Display Style**.

Under **Display style for new views**, click **Hidden lines removed**.

Under **Tangent edges in new views**, click **Use font**.

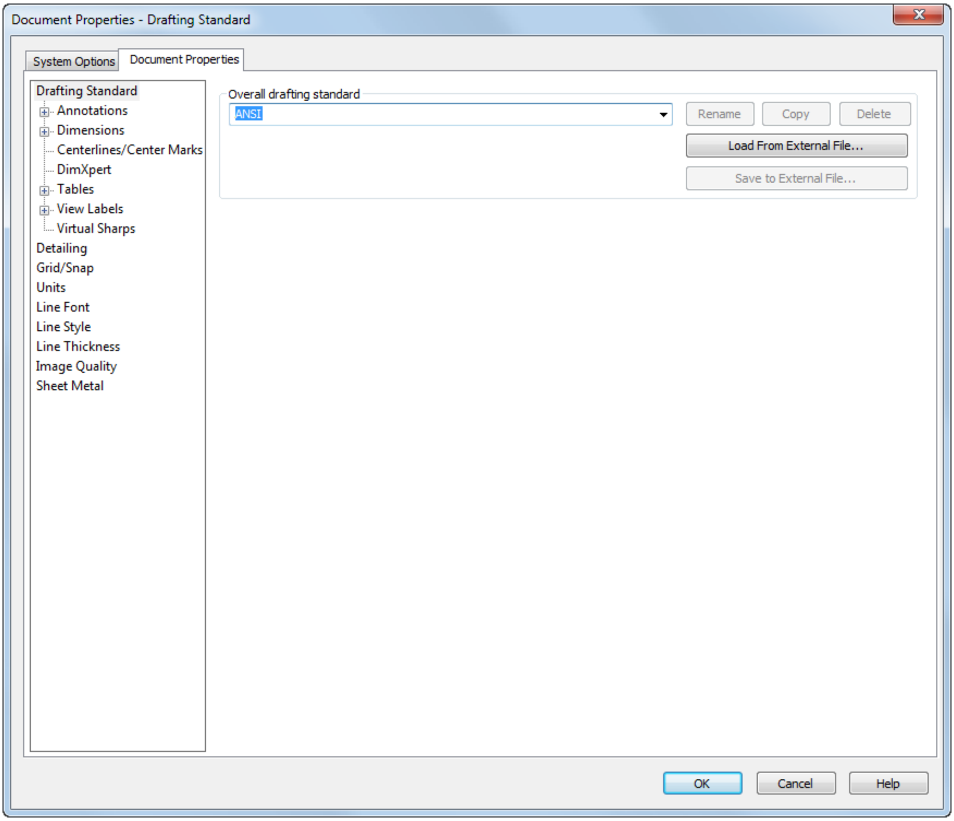
Under **Display quality for new views**, click **High quality**.

Do not click **OK** yet. Continue with the next step.



7 Document properties.
Click the **Document Properties** tab and select **Drafting Standard**.

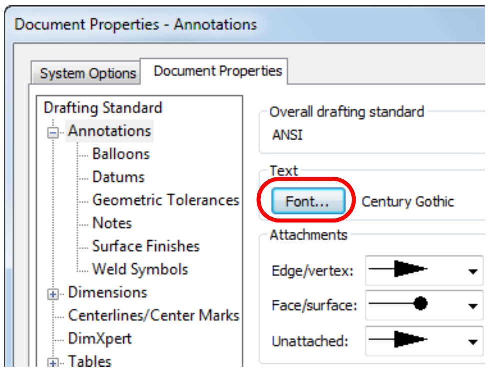
For **Overall drafting standard**, select **ANSI**.



Do not click **OK** yet. Continue with the next step.

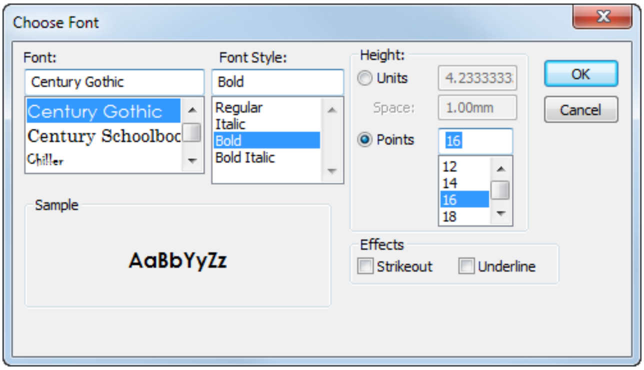
8 Annotations font.
Under **Drafting Standard**, click **Annotations**.

Under **Text** select **Font**.



9 Choose the font.

The **Choose Font** dialog box opens.



Under **Font**, select **Century Gothic**.

For **Font Style**, select **Bold**.

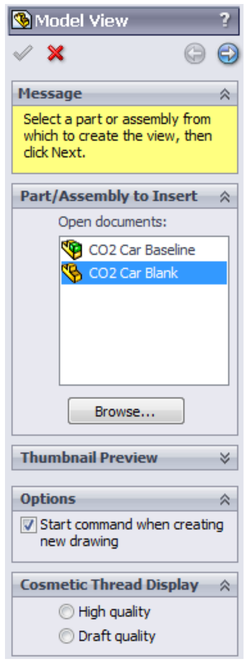
For **Height**, click **Points** and then select **16**.

Click **OK** to close the **Choose Font** dialog box.

Then click **OK** again to close the **Options** dialog box.

10 Model views.

Click **Insert, Drawing View, Model**, or click **Model View** on the Drawing toolbar.



The **Model View** PropertyManager is a wizard-like tool that helps guide you through the process of creating views on the drawing.

The **Open documents** list shows all the parts and assemblies that are currently open. If you wanted to make a drawing of a part or assembly that was *not* open you would click **Browse**.

The first drawing we want to make is of the car body.

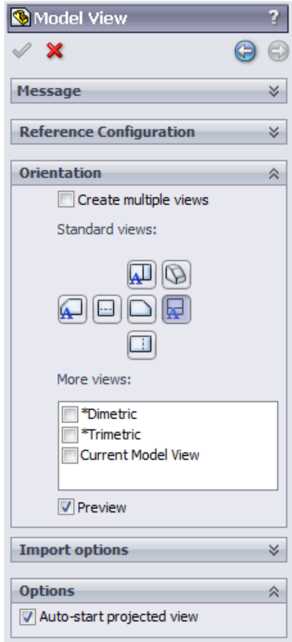
Select the `CO2 Car Blank`, and then click **Next** .

11 View orientation.
In the **Orientation** section of the PropertyManager, select **View orientation**.

Under **Standard views**, click **Right**.

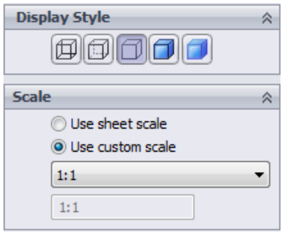
Click **Preview**.

Under **Options**, click **Auto start projected view**.



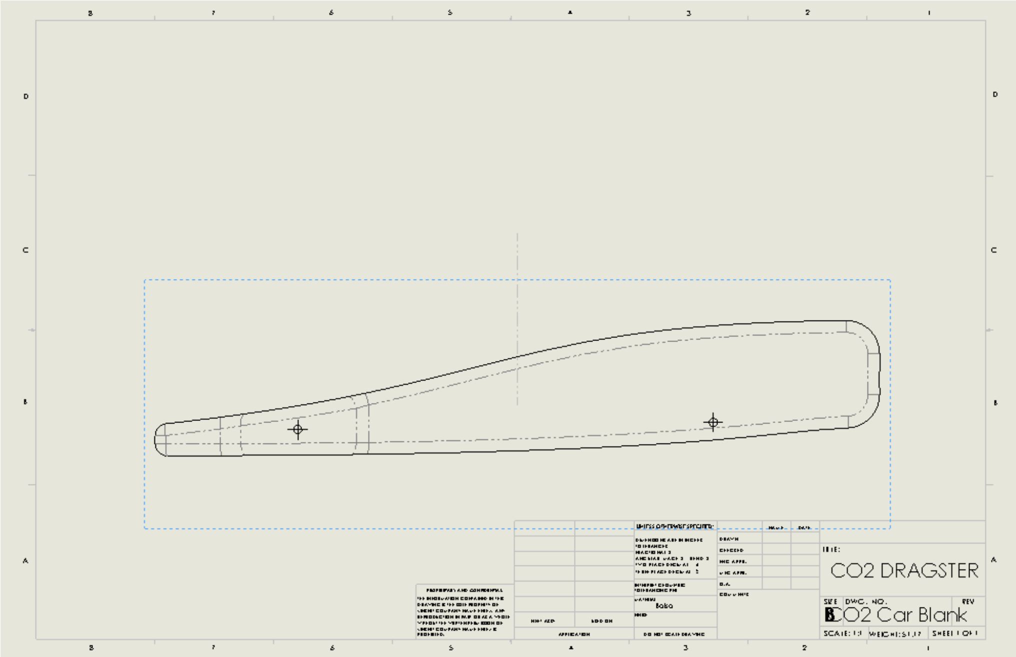
12 Display style and scale.
For **Display Style**, click **Hidden Lines Removed**.

For **Scale**, click **Use custom scale** and select **1:1**.



13 Position the view on the drawing.
Move the cursor into the graphics area. A preview of the view appears.

Click to position the view as shown in the illustration below.

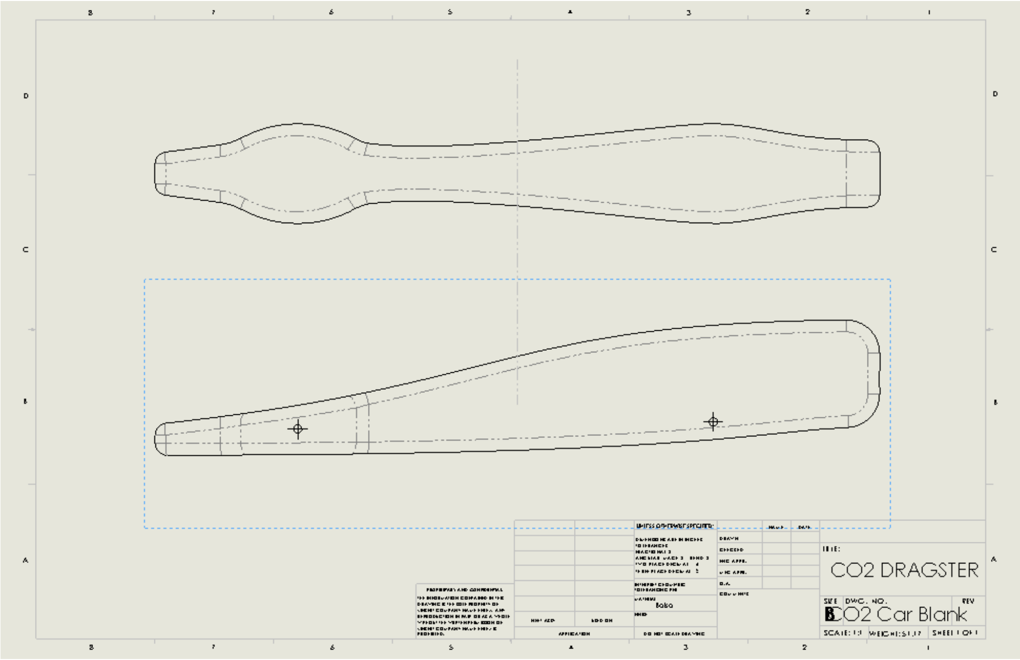

CO2 DRAGSTER
CO2 Car Blank
Balsa

14 Create the top view.

If you recall, in step **11** on page 112 we selected the **Auto-start projected view** option. This means that once we have placed the first view on the drawing, we can automatically create additional views projected from it.

Move the cursor above the existing side view. Another preview appears, this time of the top. Click to position the view as shown.

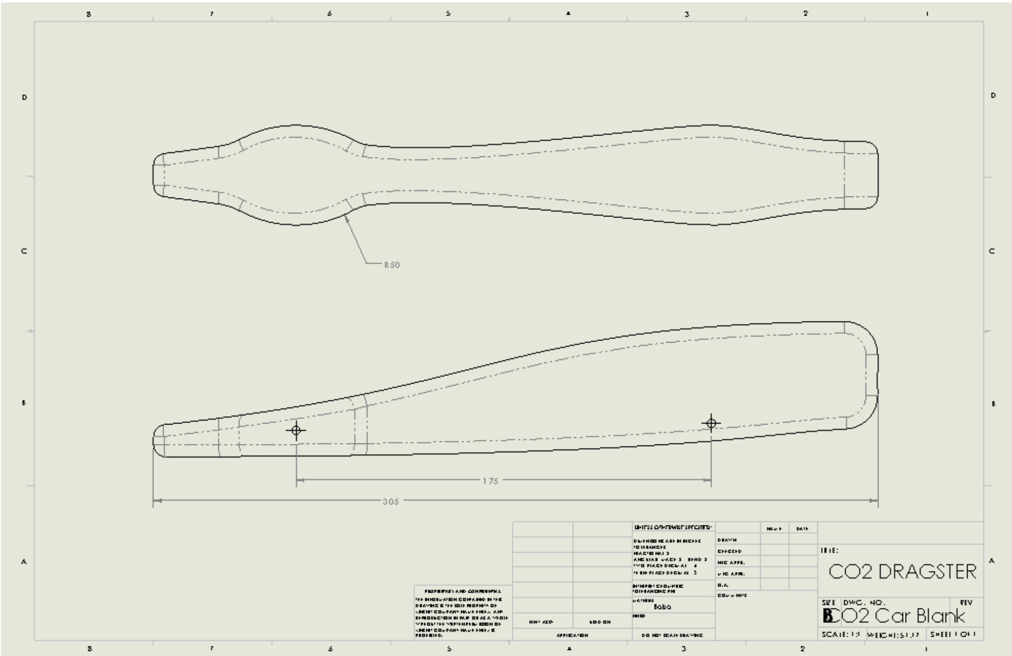
Click **OK** to end the command.



15 Add dimensions.

Click **Smart Dimension** on the Sketch toolbar. Add some dimensions to the drawing.

Note: The objective of this lesson is not to produce a completely dimensioned engineering drawing. Rather it is to introduce some of the basic steps engineers go through when producing documentation for a product.



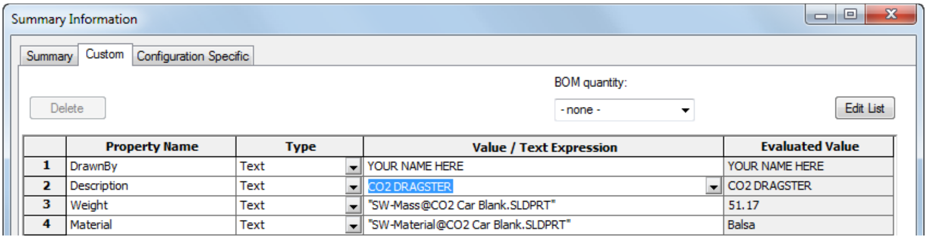
16 Editing the title block.

The title of the drawing sheet is automatically filled in with information that is in the file properties of the car body.

Click **Windows, CO2 Car Blank *** to switch back to the part document.

Click **File, Properties**. The **Summary Information** dialog box appears.

Click the **Custom** tab. Enter the name of your dragster in the **Value** column of the **Description** and click **OK**.



17 Switch back to the drawing window.

The title block is automatically updated with the new information.

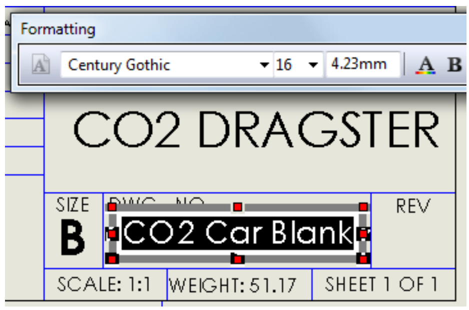
18 Edit sheet format.

The `DWG. NO.` text is too large for the block. Right-click on the drawing sheet and select **Edit Sheet Format**.

Double-click the text `CO2 Car Blank` and set the font size to **16**.

Click off the menu.

Right-click on the drawing sheet and select **Edit Sheet**.



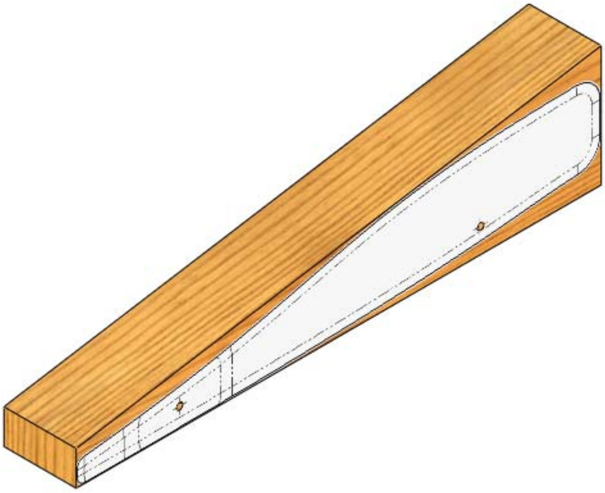
19 Save your work.

Click **File, Save**, or click **Save** on the Standard toolbar.

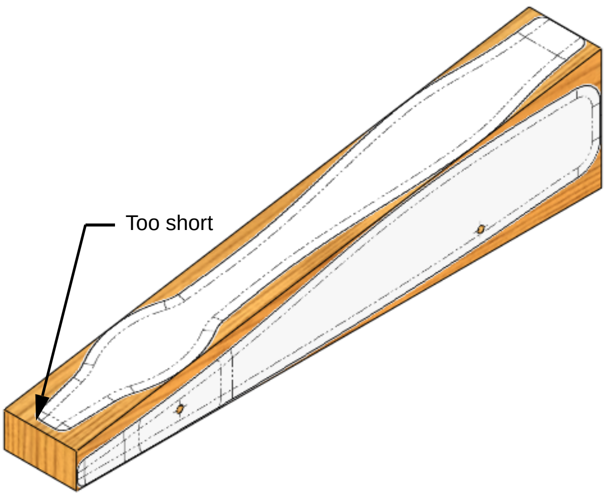
Using the Drawing to Build the Car

If your school has a B-size (11” x 17”) printer or plotter you can make full size prints of your car body design. You can use the drawings to make templates for cutting out the shape of the body.

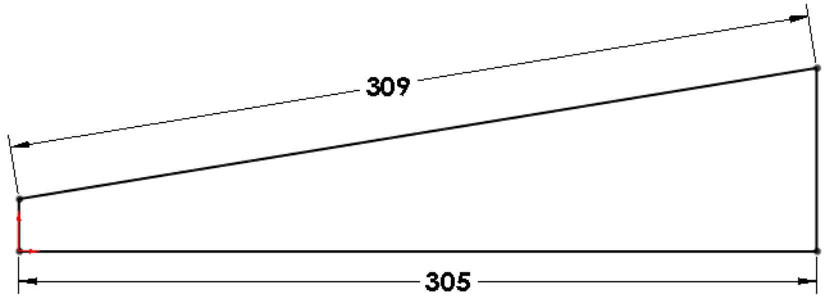
Use scissors to cut out the side view of your design. You can either trace around the template, or attach it to the blank using tape or spray adhesive.



However, if you cut out the top view and do the same thing, you will notice it doesn’t fit correctly. It is too short. Why is that?

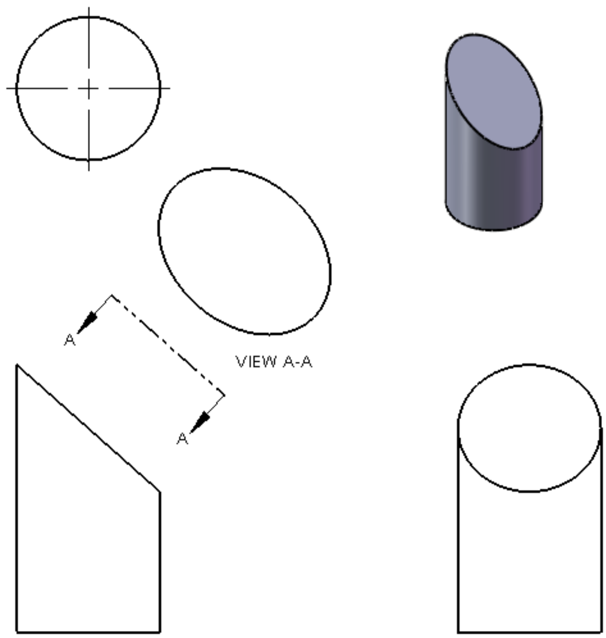


The answer can be found by measuring the blank. The overall length of the blank is 305mm. However, the length of the angled face is about 309mm – about 4mm longer. In order to make a template that is the right size, we need a view that "looks" perpendicular to the angled face. This is called an auxiliary view.



Auxiliary Views

The purpose of an auxiliary view is to show the true size and shape of an inclined surface which is not correctly represented in the front, top, or side views. Auxiliary views are taken from a direction of sight other the standard orthographic directions. The direction of sight is perpendicular to the inclined face you want to represent. Complete auxiliary views are usually unnecessary and often confusing. Generally you want to show only the inclined face and that face alone. Arrows indicate the direction of sight. View A-A, below, is an auxiliary view.



Adding Sheets to Drawings.

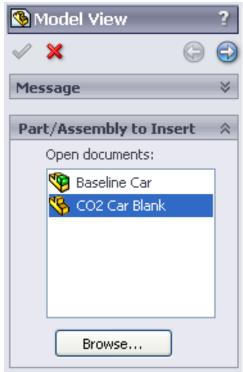
SolidWorks drawings can contain multiple sheets. Each sheet can contain multiple drawing views. New sheets are added as needed to fully document the project.

1 **Add sheet.**
Click on **Add Sheet** (near the `Sheet1` tab on the bottom of the screen).

A new `B-Landscape` sheet is added.

2 **Model views.**
Click **Insert, Drawing View, Model**, or click **Model View** on the Drawing toolbar.

Select the `CO2 Car Blank`, and then click **Next**.



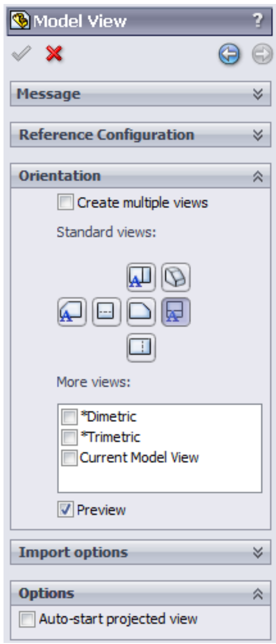
3 **View orientation.**
In the **Orientation** section of the PropertyManager, select **View orientation**.

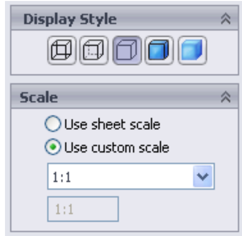
We want to make a drawing of the car body with an auxiliary view. To do that we first need the side view.

Under **Standard views**, click **Right**.

Click **Preview**.

Under **Options**, clear **Auto start projected view**.





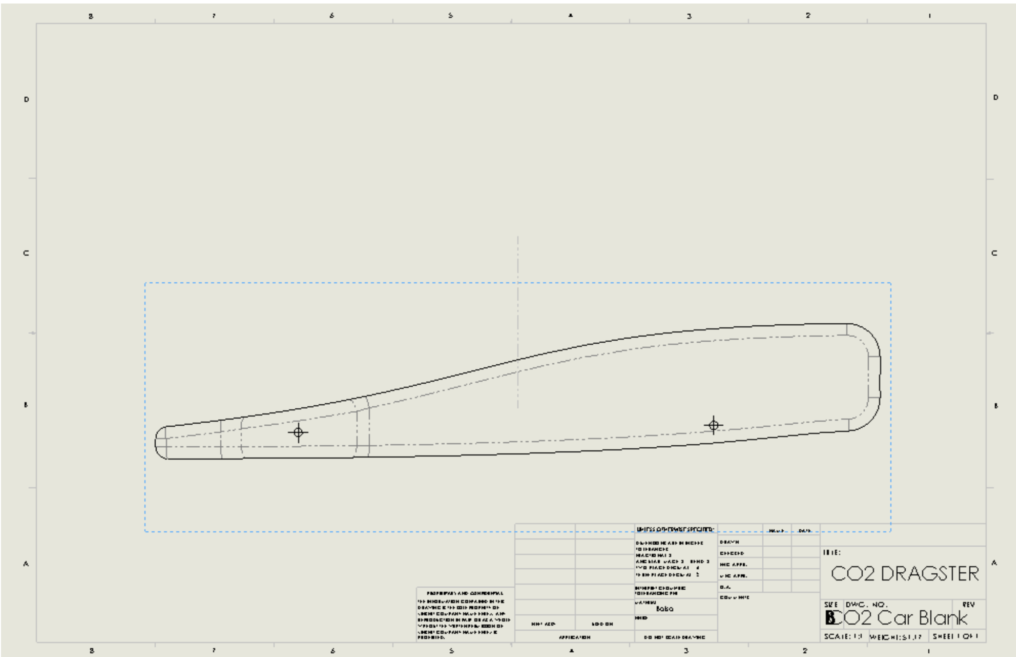
4 Display style and scale.
For **Display Style**, click **Hidden Lines Removed**.

For **Scale**, click **Use custom scale** and select **1:1**.

5 Position the view on the drawing.
Move the cursor into the graphics area. A preview of the view appears.

Click to position the view as shown in the illustration below.

Click **OK**.



Reference Edge for Auxiliary View

In order to create an auxiliary view, we need a reference edge that defines the angle of the projection. Currently the side view only shows the curved edges of the car body. To get a reference edge at the correct angle, we need to see the original blank. This is done by changing the view so it shows the original configuration named `Blank` which we saved before making the modifications to the design. [See step **3** on page 63.]

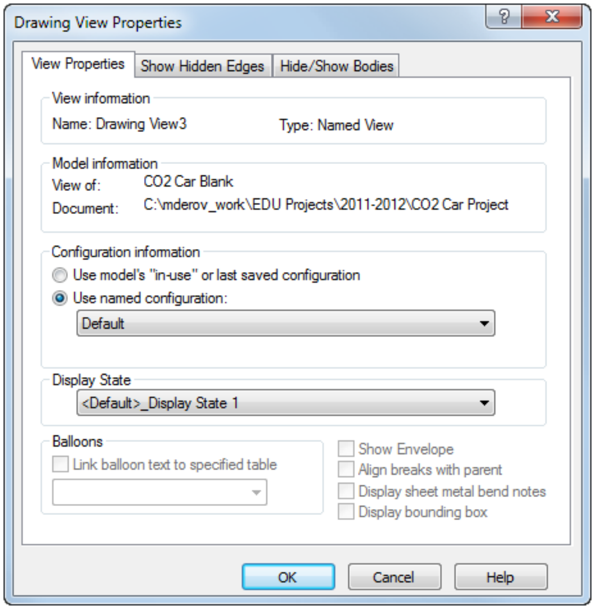
6 View properties.

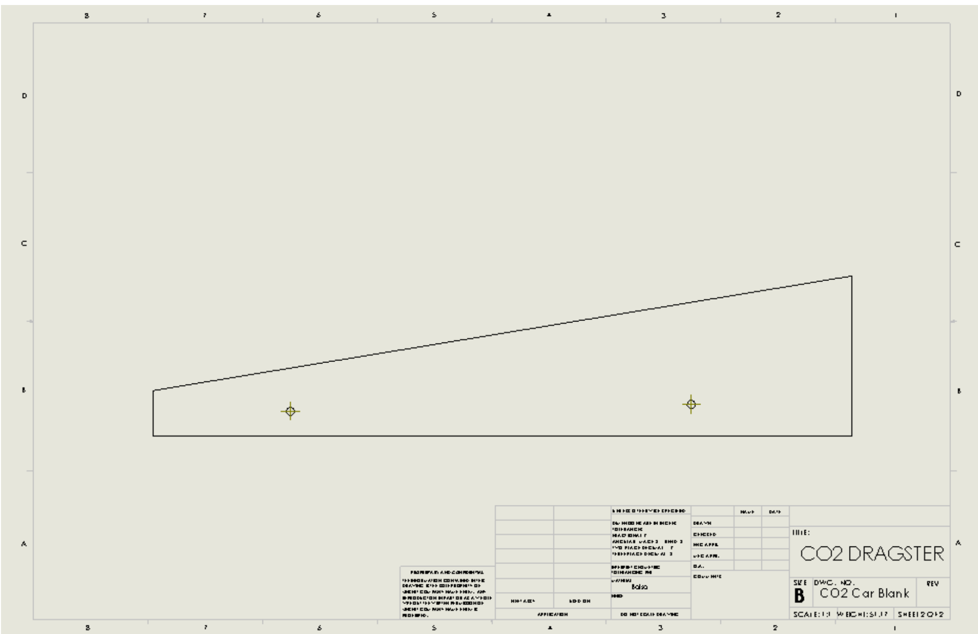
Right-click inside the drawing view and select **Properties** from the shortcut menu.

Under **Configuration information**, select **Use named configuration** and select `Default` from the list.

Click **OK**.

The view now shows the original blank, before we made the design modifications.

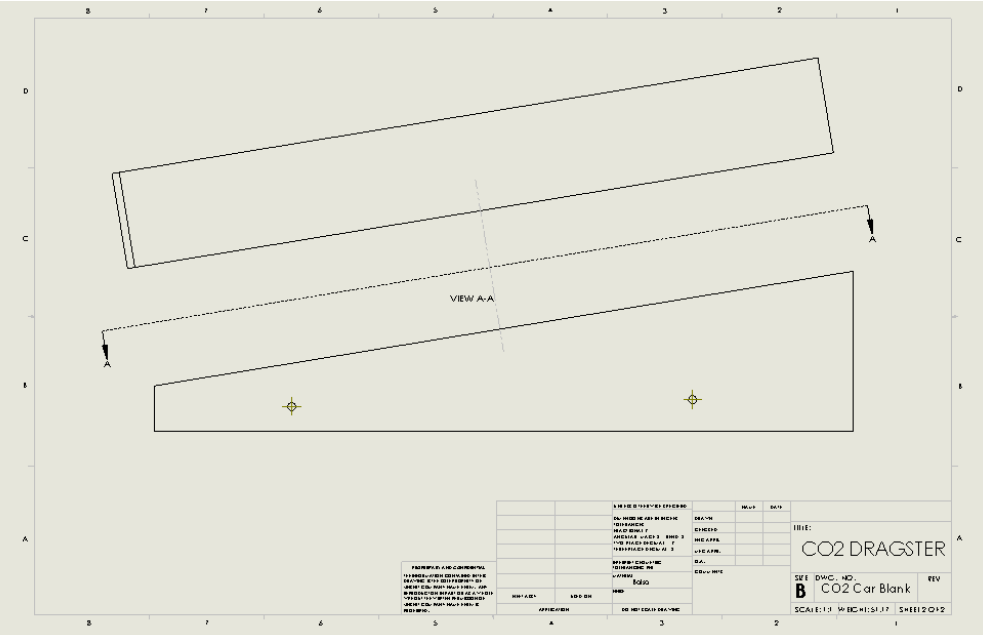




7 Auxiliary view.

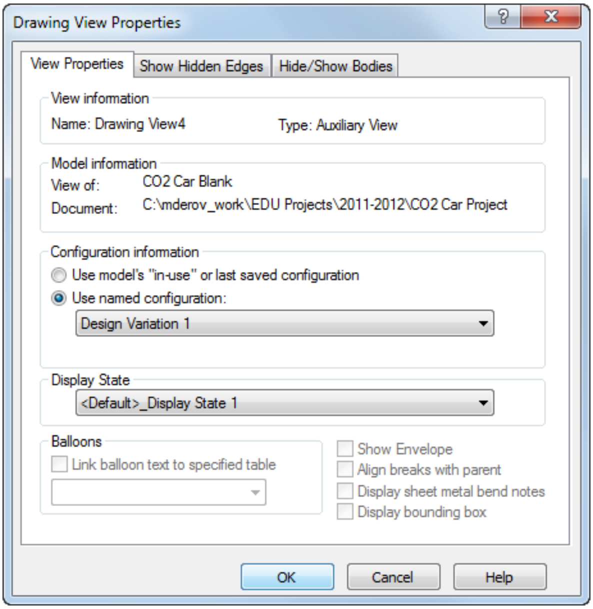
Click **Insert, Drawing View, Auxiliary**, or click **Auxiliary View**  on the Drawing toolbar.

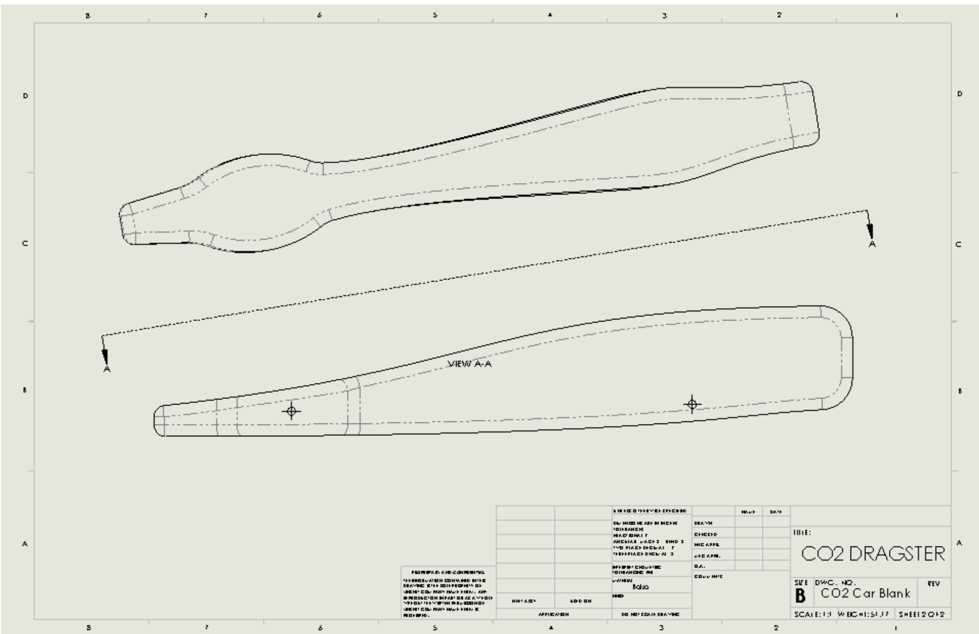
Select the angled line in the side view. Move the cursor above the existing view. The preview appears. Click to position the view as shown below.



8 Change the drawing view configurations.

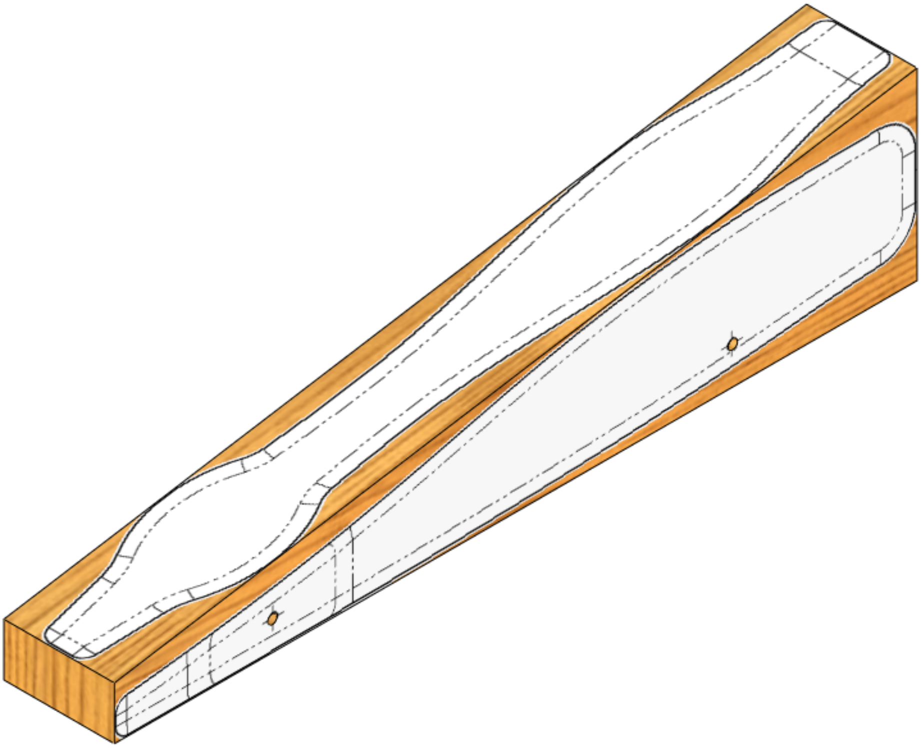
Repeat the procedure you followed in step **6** on page 121 and change both views so they reference the configuration `Design Variation 1`.





Design Template

You can now print and cut out the auxiliary view and fasten it to the blank as a template. Because it is an auxiliary view, it shows the outline true size and shape.



Assembly Drawing

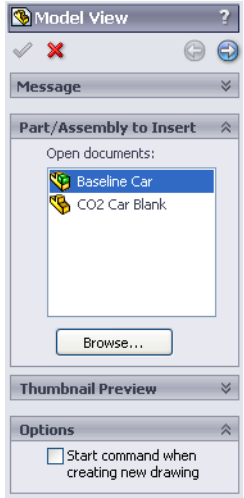
We want to make a drawing of the assembled car, complete with wheels. This will go on a separate sheet.

1 **Add another drawing sheet.**
Click on **Add Sheet** (near the `Sheet1` tab on the bottom of the screen).

A new `B-Landscape` sheet is added.

2 Model views.
Click **Insert, Drawing View, Model**, or click **Model View** on the Drawing toolbar.

Select the `CO2 Car Baseline`, and then click **Next**.

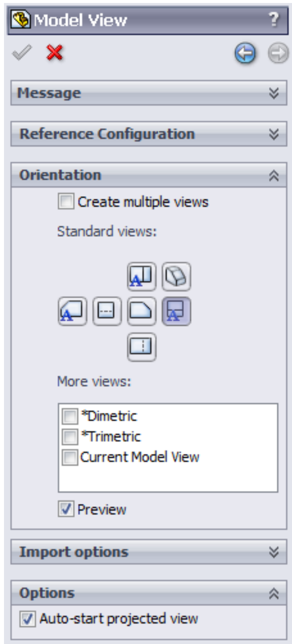


3 View orientation.
In the **Orientation** section of the PropertyManager, select **View orientation**.

Under **Standard views**, click **Right**.

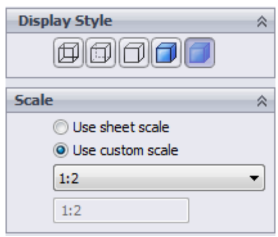
Click **Preview**.

Under **Options**, click **Auto start projected view**.



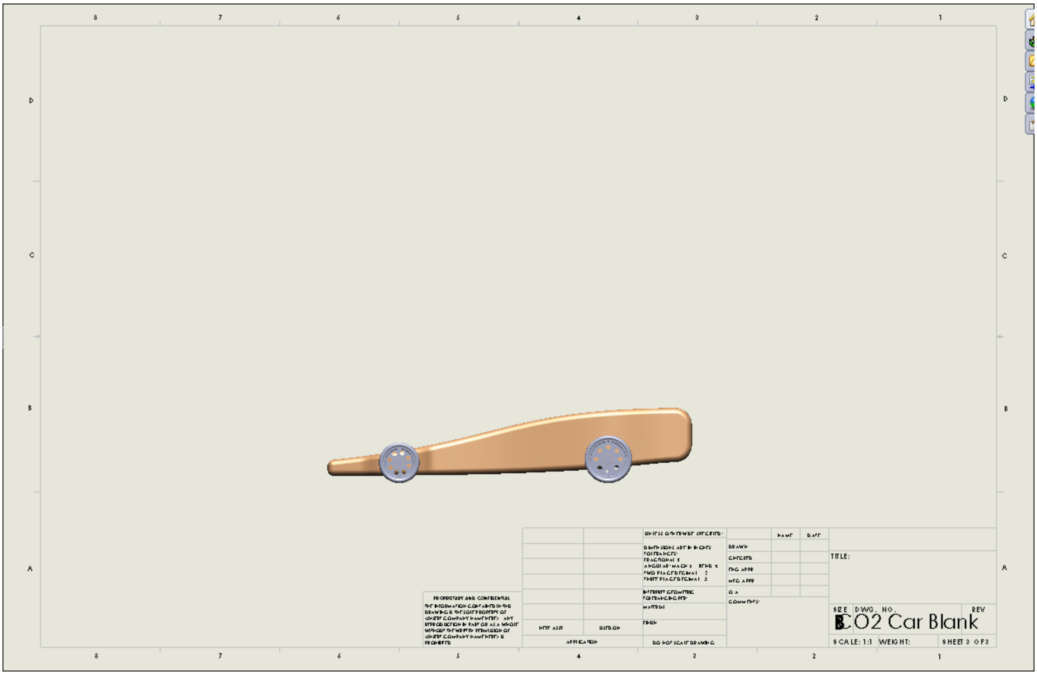
4 Display style and scale.
For **Display Style**, click **Shaded**.

For **Scale**, click **Use custom scale** and select **1:2**.



5 Position the view on the drawing.
Move the cursor into the graphics area. A preview of the view appears.

Click to position the view as shown in the illustration below.



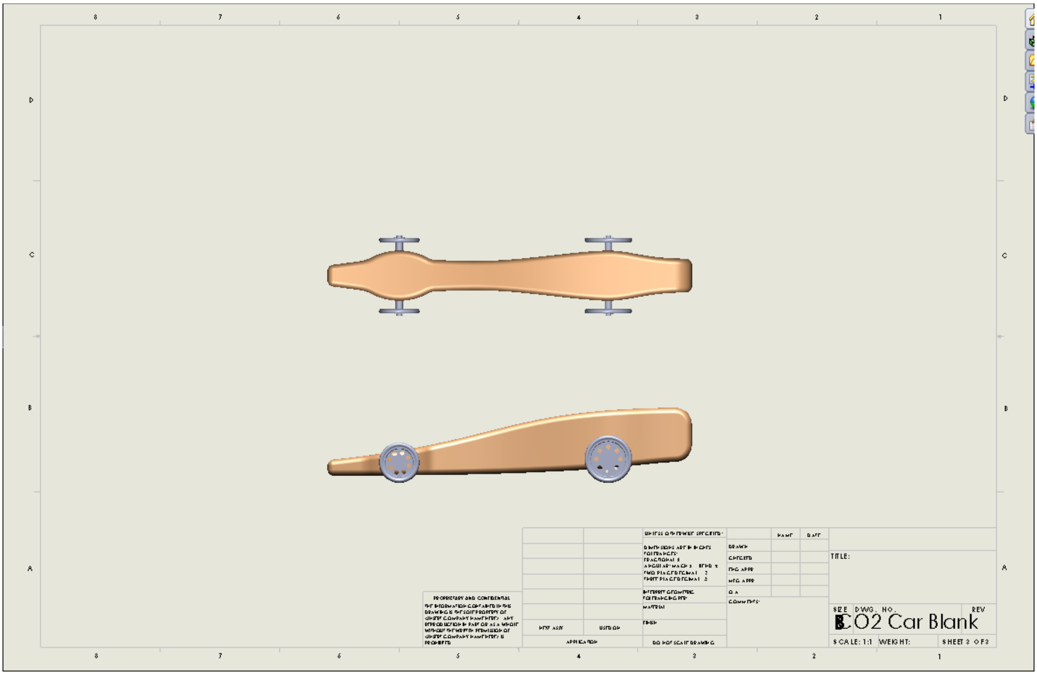
6 Create the top view.

In step **3** on page 125 we selected the **Auto start projected view** option.

This means that once we have placed the first view on the drawing, we can automatically create additional views projected from it.

Move the cursor above the existing front view. Another preview appears, this time of the top. Click to position the view as shown.

Do not click **OK** yet. Continue with the next step.

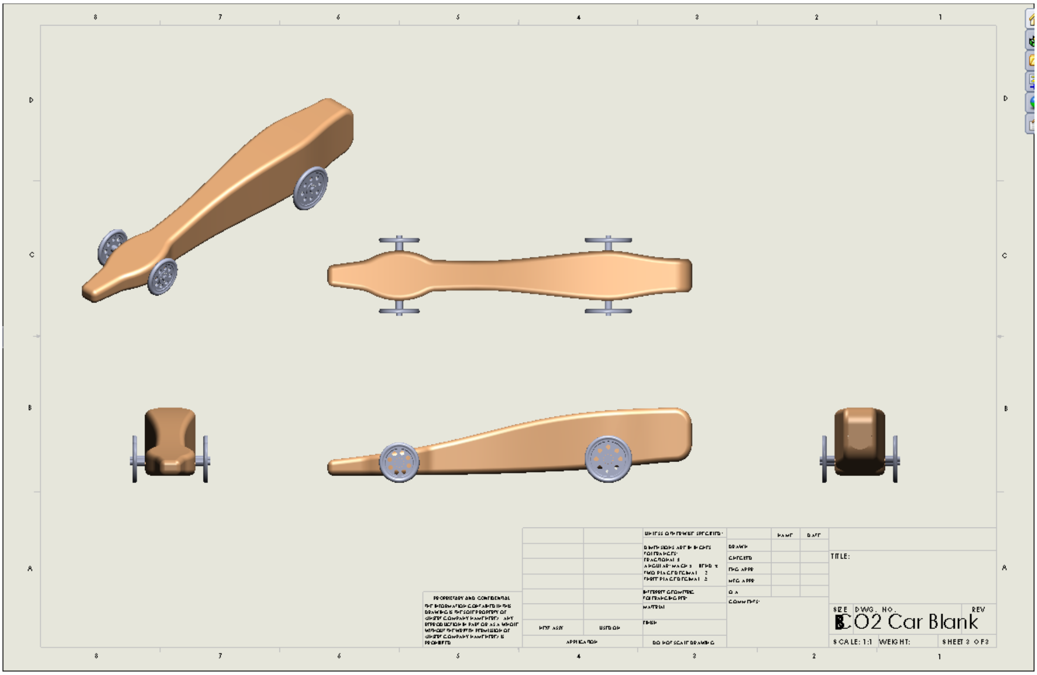


7 Create other projected views.

Continuing with the process you used in step **6**, create three more views:

- One showing the rear of the car
- One showing the front of the car
- An isometric view

Click **OK** to end the command.



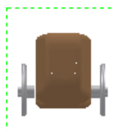
Fine Tuning the Drawing

Although the drawing shown above looks nice, we're going to make a couple of changes to improve it:

- Use a larger view scale of 2:3 for the front, top, and side views, but keep the isometric view at 1:2
- Eliminate the rear view of the car
- Rearrange the views on the drawing sheet

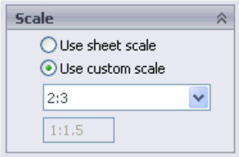
8 Delete view.

Select the view that shows the rear of the car and press **Delete** on the keyboard.



A message will ask you to confirm that you want to delete the view. Click **Yes**.

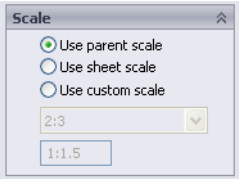
9 Change the view scale.



Click inside the view that shows the side of the car. This is the first view you created on this drawing, in step **5** on page 126.

Select **User Defined** from the list and set the scale to **2:3**.

10 Use parent scale.



Notice that all of the other views also change scale. If you click inside the top view and look at the PropertyManager, you will see why.

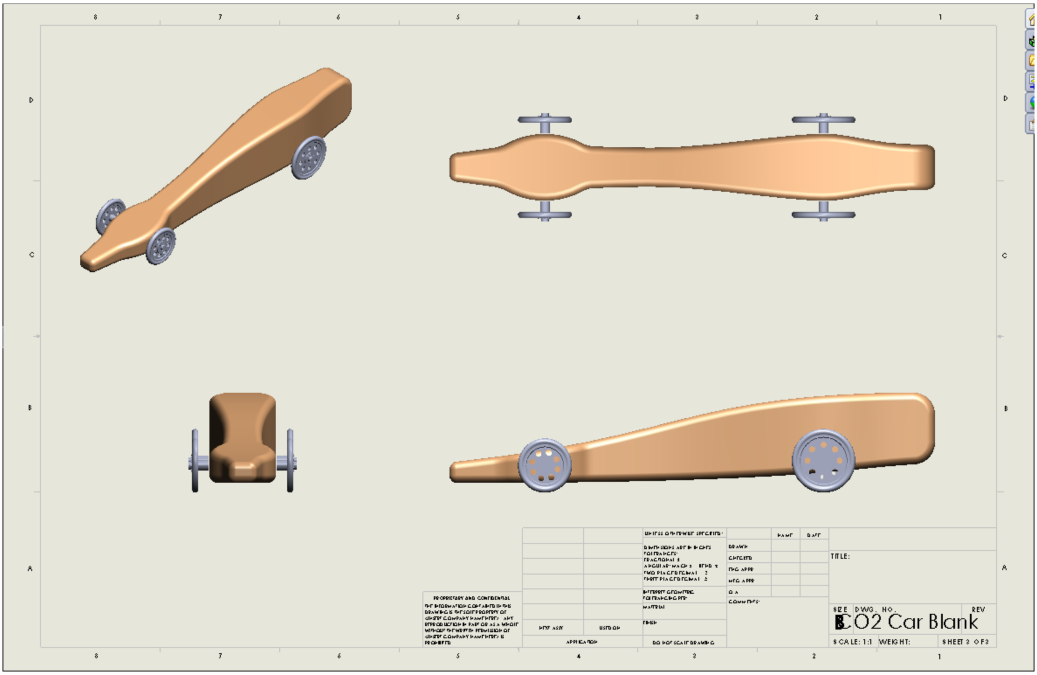
The views that were projected off of the first view are linked to the scale of the source or parent view.

11 Change the scale of the isometric view.

Click inside the isometric view. In the PropertyManager, under **Scale**, click **Use custom scale** and set the scale to **1:2**.

12 Rearrange the views.

Drag the views to position them better on the drawing sheet. Notice that when you drag the parent view, the two projected views maintain their alignments. You can drag the isometric view independently.



13 Save your work.

Lesson 7
Visualization and Rendering

When you complete this lesson, you will be able to:

- Load the PhotoView 360 add-in;
- Apply appearances to a model;
- Apply a scene;
- Create and apply decals to the model;
- Adjust the lighting for best effect;
- Understand what makes an image look realistic and make changes to improve the realism of the rendering;
- Save a rendered image.

Rendered Images

In the previous lessons, we viewed our model through either OpenGL or RealView. If we were not viewing the model in RealView, then we were in OpenGL.

OpenGL

OpenGL (Open Graphics Language) is the default visualization method in SolidWorks. In OpenGL, models can be displayed as shaded, wireframe or a combination of the two. All modern graphics cards, regardless of cost, have OpenGL capabilities.

The visualization capabilities with OpenGL include:

- Surface shading including color, ambience, diffusion, specularity and transparency.
- Basic texture mapping techniques.
- Diffused ground shadow.

RealView

RealView incorporates OpenGL but takes it to another level. More realistic effects can be achieved such as dynamic environmental reflections. All of these effects are handled by the graphics card. Most modern video cards, in the professional workstation class, can display all the effects of RealView.

The visualization capabilities with RealView include:

- Advanced shading
- Reflections (environmental)
- Self-shadows
- Ground reflection
- Advanced texture mapping and bump maps

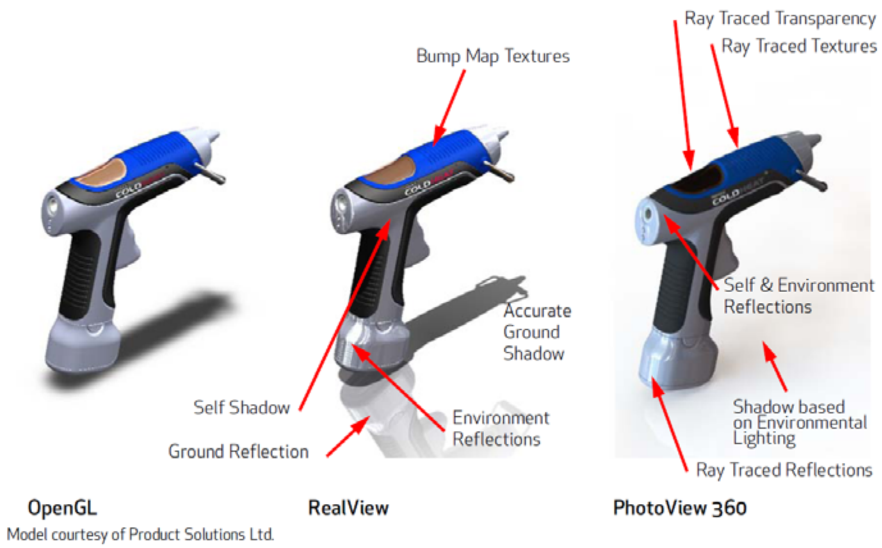
PhotoView 360

PhotoView 360 is distinguished from RealView in that it is not a process of the video card and photorealistic renderings can be achieved regardless of the graphics card used. PhotoView 360 renderings are more photorealistic than RealView because light rays and reflections are more accurately calculated. The use of High Dynamic Range (HDR) environmental images allows for greater realism. PhotoView 360 supports:

- Self-reflection (reflection of objects in one another)
- HDRI (environmental lighting)

Display Comparison

The following image shows a comparison of the same model in OpenGL, RealView and PhotoView 360. You can see the different aspects discussed in the previous paragraphs.



The Visualization/Rendering Process

Visualizing our models, in any of the three modes, involves the same elements.

- **Appearances**

Appearances specify the surface properties of the model such as color, texture, reflectance and transparency. Appearances can be attached to a part, feature, body or face. Appearances are different from materials. Materials are the physical properties of the model, so properties such as density and yield strength come from the assigned material. Appearances only define the way the model looks. Think of an automobile, the fender may be made of steel (material) which defines its strength and stiffness, but what we see is the paint (appearance) which is the color and shininess.

- **Decals**

Decals are images, such as company logos that are applied to the model.

- **Scenes**

Each SolidWorks model is associated with a scene, for which you can specify properties such as environments and backgrounds. Scenes help to put the model in context.

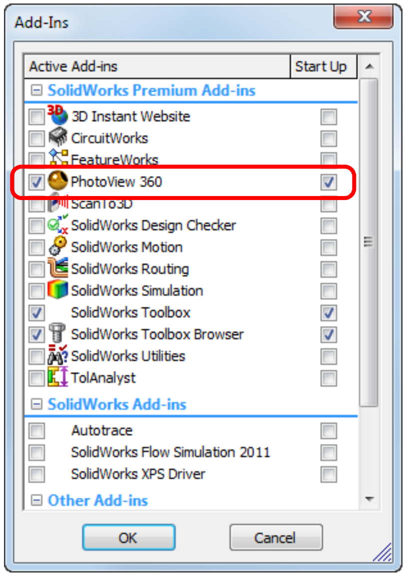
- **Lights**

When in OpenGL or RealView, lighting is provided by discrete lights. Every scene applied to the model has its own set of lights, but you can also add and adjust the lights to achieve a desired effect.

When rendering using PhotoView 360, the primary light source is from the environmental image. You can however add discrete light as desired.

Creating a Rendering

Rendering is the process of applying the appearance, scene, lighting and decal information to the model and then having the light rays traced and accurately portray the way the model would look if it were real. Many of the steps in the process are done in core SolidWorks with only the final setup and processing done by PhotoView 360.



1 **Load the add-in.**
Click **Tools, Add-ins…** and select **PhotoView 360**.

Since we are done using Flow Simulation, clear that check box to turn off the add-in.

Click **OK**.

2 **Open the file.**
Click **Open** on the Standard toolbar, and open the `CO2 Car Baseline.sldasm` which you built earlier.

3 **View orientation.**
One key to creating a realistic rendering is setting up the view. Both view orientation and perspective are important ingredients.

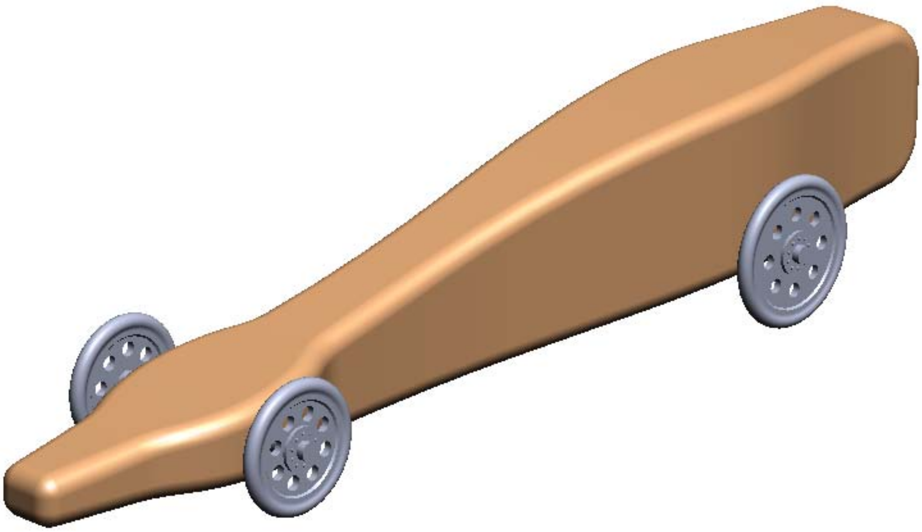
Set the view orientation to **Isometric** and select **Shaded** view mode from the View toolbar.

Press the **Up Arrow** key once.

Click **Perspective** on the View toolbar.

Click **View, Modify, Perspective** and set the **Observer Position** to **4**.

Your model should look like the illustration below.



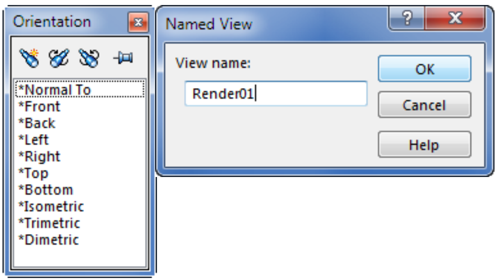
4 Save the view state.

Press the **Spacebar**. The **View Orientation** menu appears.

Click **New View** . The **Named View** dialog box appears.

For **View name**, type `Render01` and click **OK**.

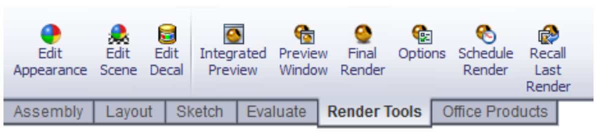
This saves the view state so you can return to it easily.

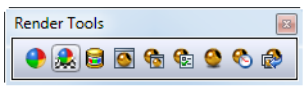


Render Tools toolbar.

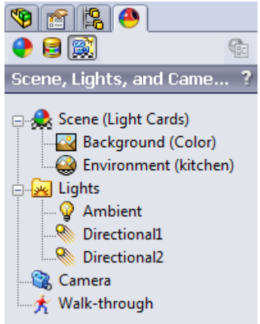
When PhotoView 360 has been added in, a dedicated toolbar and CommandManager tab becomes available.

This Render Tools toolbar contains the tools required to render the model.





DisplayManager



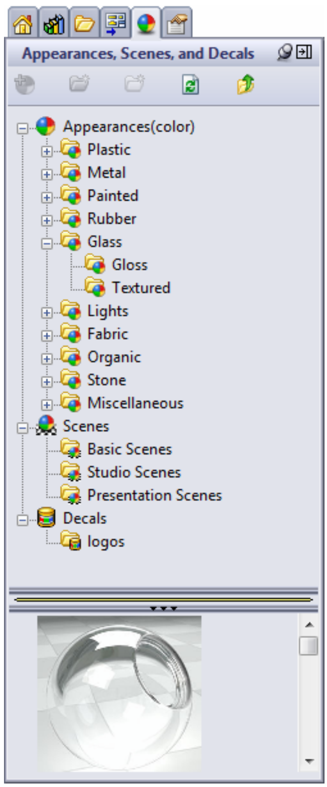
The DisplayManager provides an outline view of the appearances, scenery, decals, cameras and lighting associated with the active SolidWorks part or assembly.

The DisplayManager indicates which items of geometry are attached to which appearances and decals.

The DisplayManager also makes it easy to:

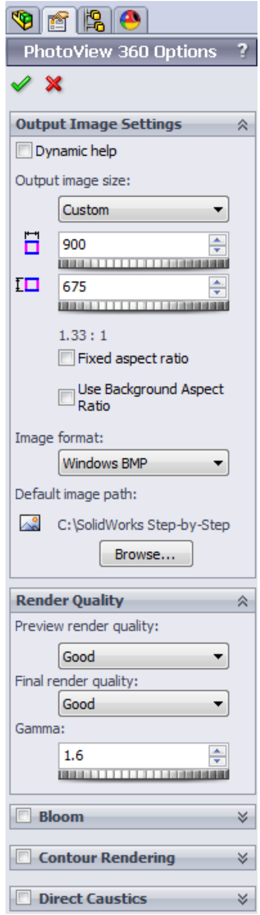
- Understand the way in which appearance and decal inheritance works.
- Select and edit appearances and decals associated with the model.
- Access the appearance, scene, lighting, camera and decal properties.
- Transfer appearances and decals between components, features, and faces.

Appearances, Scenes and Decals Tab



The **Appearances, Scenes and Decals** tab is located on the Task Pane and contains appearances, scenes, and decals. Appearances, scenes, and decals can be applied to the model by simply dragging the item from the bottom pane of the **Appearance, Scenes and Decals** tab and dropping it into the graphics area or on the model.

PhotoView 360 Options



PhotoView 360 has its own options dialog box. Options allow you to customize the PhotoView 360 software to reflect your preferences for default settings. Options are divided into **Output Image Settings**, **Render Quality**, **Bloom**, **Contour Rendering**, and **Direct Caustics**.

For a complete listing of all the settings available though the **PhotoView 360 Options** dialog refer to the **Help** menu.

Where to Find It

- Click **PhotoView 360, Options**.
- On the Render Tools toolbar, click **Options** .
- In the DisplayManager, click **PhotoView 360 Options** .

Make a Configuration for Rendering

It is good practice to make a configuration of the assembly specifically for the purposes of rendering. This way you can make changes to the assembly without effecting things like the drawing or the Flow Simulation analysis. Some rendering-specific changes you might make include:

- **Scenes/Lights –** You can use different scenes or turn lights on or off, change their position, color, intensity, and other optical characteristics. You probably don't want those changes to appear in all versions of the assembly; just the one you're using for rendering.
- **Props –** You can add other parts to the assembly to increase the realism of the rendering. You would not want these parts to appear in the drawing or to be included in the Flow Simulation analysis.

5 New configuration.

Switch to the ConfigurationManager. The active configuration should be Narrow Rear Wheels since that is what we used for the final analysis and for the drawings. If it is not, activate it.

Right-click the top-most icon in the tree and select **Add Configuration** from the shortcut menu. The new configuration will be a copy of the active one.

Name the new configuration Render.

Appearances

Appearances affect the way a surface reacts to light. They may be applied to assemblies, components, parts, bodies, features or faces. Appearances are of two general types: procedural and textures. Appearances are far more than just the color or pattern of colors you see on the screen because they also contain information about how the surface will reflect or refract light, transparency, mapping and more. All surfaces of a model have an appearance applied.

Appearances can be added when we are in the part or assembly, however where the appearance is attached can make a significant difference in what you see because of the appearance hierarchy.

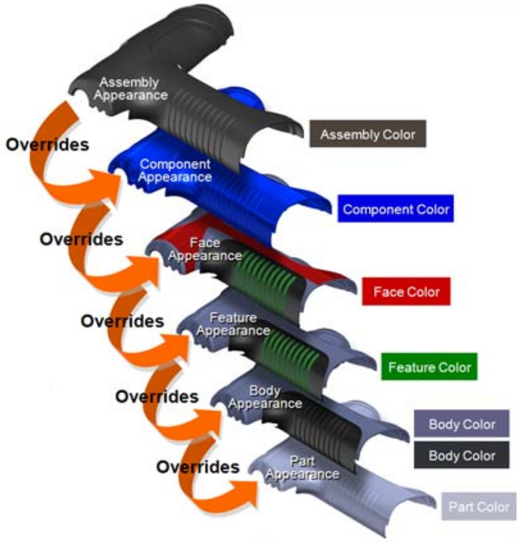
Appearances applied to the model are used for OpenGL, RealView and PhotoView 360 rendering. You do not have to do anything special to use the appearances for rendering.

Appearances vs. Materials

While materials have appearances associated with them, appearances can be applied that are completely different from the material. An example would be a painted piece of steel. The material applied would be steel, which would be used to calculate the weight of the part and stress calculated through Finite Element Analysis, while an appearance of paint would be used to show how the part will look with paint applied. Additionally, the surface finish of appearances may be different such as a brass appearance could have a surface finish of cast, rough, satin or polished.

For our CO2 car, the material balsa was added to the original part. Balsa applied a default appearance to the model that looks like a wood grain, however a different appearance was added to make the car look like a solid color to make it easier to understand the lessons. To get a better rendering, we are going to add another appearance to the car to make it look like it has a high gloss paint.

Hierarchy of Appearances



The place where appearances are applied affects the final result. The hierarchy in an assembly works opposite the way it does inside a part.

If you apply an appearance at the assembly level, it overrides all other appearances. If you apply an appearance to a part, it is overridden by appearances applied to elements of the part such as features, bodies and faces.

The hierarchy is:

1. Assembly
2. Component
3. Face
4. Feature
5. Body
6. Part

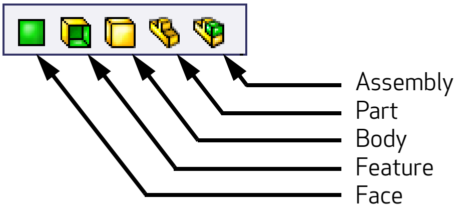
Applying Appearances

There are many methods to apply an appearance to a model. To apply an appearance:

- Drag the appearance from the Task Pane into the graphics area to apply the appearance to the entire part or assembly.
- Drag an appearance and drop it onto a body, feature or face in the graphics area. Select from the appearance target the entity to attach the appearance.
- Select the part, body, feature or face, then double-click the Appearance Selection area of the Task Pane.
- Select the part, body, feature, or face, then right-click the appearance in the Appearance Selection area of the Task Pane and click **Add Appearance to Selection(s)**.
- Select the part, feature or face, then right-click the appearance in the DisplayManager and click **Add Appearance**.

Note: If you press and hold the **Alt** key when dragging an appearance onto a part, body, feature, or face, the appearance's PropertyManager will open.

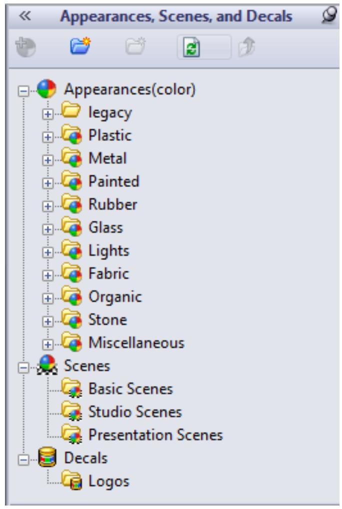
Appearance Target



The appearance target allows you to specify where the appearance will be attached. When you drag and drop an appearance on a component, the appearance target will appear. Select the appropriate icon to attach the appearance.

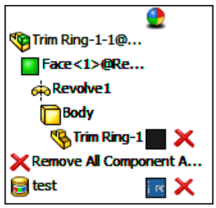
Note: The choices available in the appearance target will depend on the geometry available and whether you are in a part or assembly.

Task Pane



The **Appearances, Scenes, and Decals** tab on the Task Pane contains all the appearances, scenes, and decal setups available. You can also put you own custom folders in this area.

Appearance Callouts



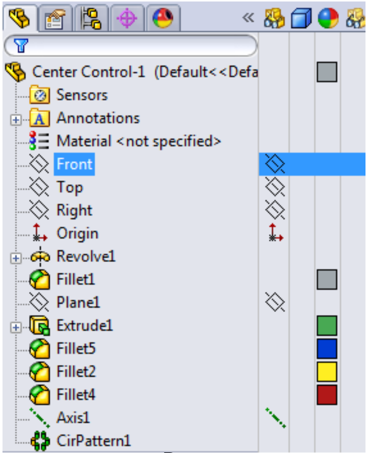
Appearance callouts show the appearances applied to the selected entity in hierarchal order. This can be very useful when trying to determine which appearance is being shown.

Where to Find It

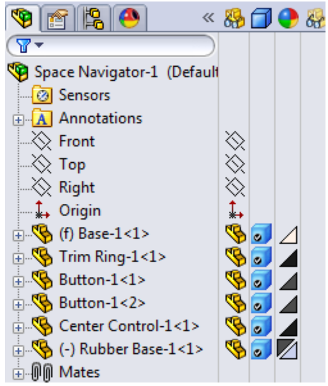
- Right-click or click a body, feature or face and select **Appearances** .

DisplayPane

The Display Pane is used to display and access visual properties of features in a part, and parts or components in an assembly.



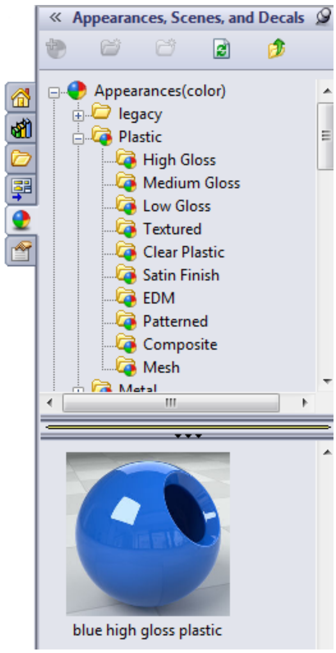
Part



Assembly

1 Apply appearance to the car body.

In the FeatureManager design tree of the assembly, select the `CO2 Car Blank` component. Be sure to select the part, not the assembly.



Tip: You do not want to select it in the graphics window because you would actually be applying appearance only to the face you clicked.

Select the **Appearance, Scenes, and Decals** tab on the Task Pane.

Click the plus sign next to `Appearances` and then `Plastic` to expand the folders. Select the folder `High Gloss`.

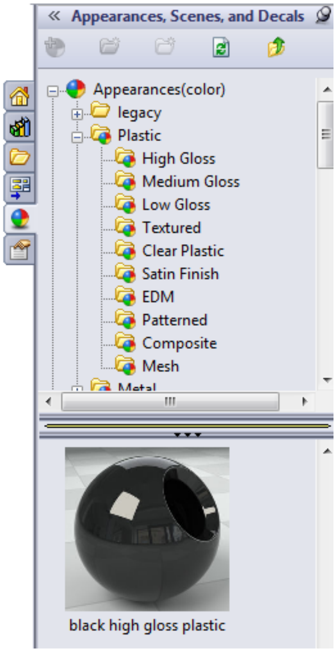
In the lower pane, locate the appearance `blue high gloss plastic`. With the `CO2 Car Blank` part still selected in the FeatureManager design tree, right-click the `blue high gloss plastic` appearance and click **Add Appearance to selection(s)**.

2 Apply appearance to the wheels.

In the FeatureManager design tree select all four wheel components.

Right-click the appearance `black high gloss plastic`, also found in the `Appearances, Plastic, High Gloss` folder, and click **Add Appearance to selection(s).**

We now have appearances on the car and all the wheels.



Scenes

SolidWorks scenes are made up of the things we see in the rendering that are not the model. They can be thought of as a virtual sphere around the model. Scenes are composed of environments, backgrounds, floors and lights. SolidWorks has a number of predefined scenes to make initial renderings quick and easy.

Applying Scenes

You can access pre-defined scenes from two sources:

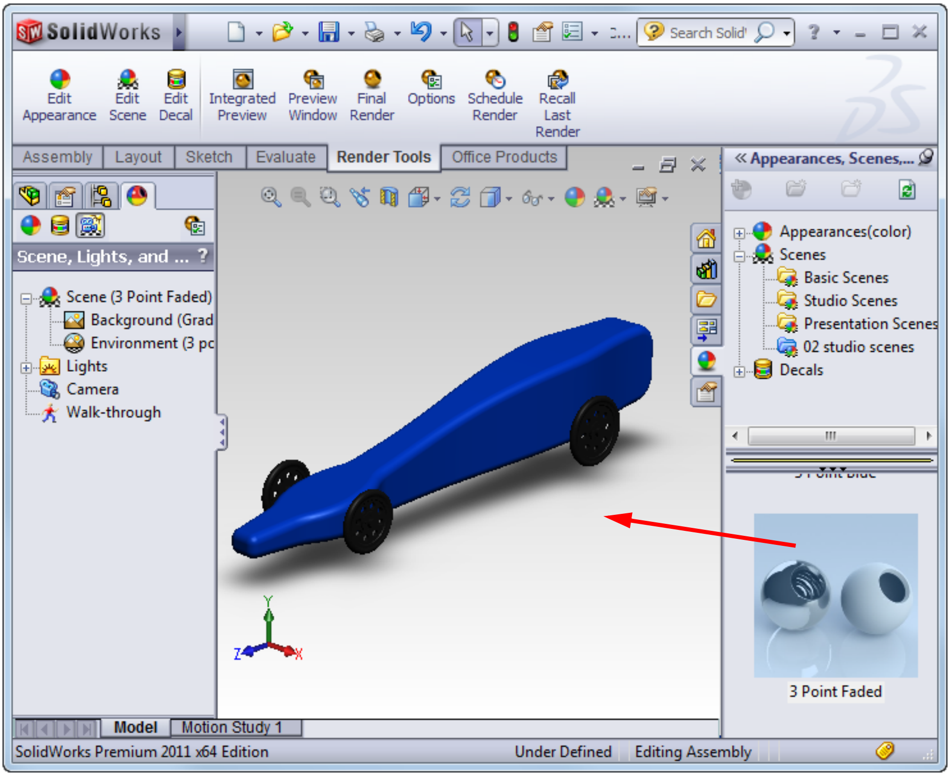
- The **Appearances, Scene, and Decals** tab in the Task Pane which lists all the pre-defined scenes that are available to apply to the model. The can be applied by dragging the scene into the graphics area.
- The **Apply Scenes** button on the Heads-Up toolbar. Click the scene and it is applied to the model.

1 Apply a Scene.

Select the **Appearances, Scenes and Decals** tab of the Task Pane.

Expand the `Scenes` folder and then select `Basic Scenes`.

Drag the scene `3 Point Faded` into the graphics area.

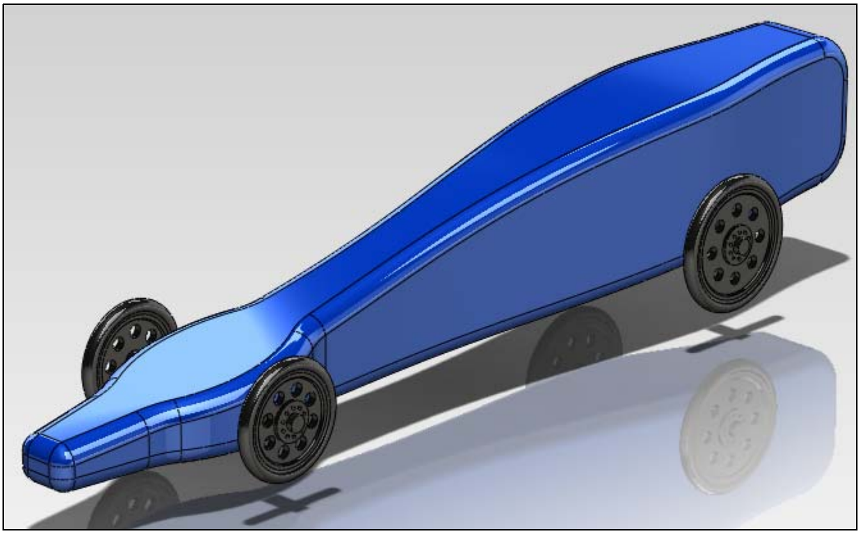


2 Turn on shadows.

If you do not have a shadow under the model, click **View, Display, Shadows In Shaded Mode** from the menu.

3 Change to RealView.
Click **View, Display, RealView Graphics** from the menu.

Examine the model, there is a shadow and reflection under the model, but the model looks like it is suspended in midair. This is not exactly what we want as we want it to appear that the car looks like it is resting on a shiny surface.



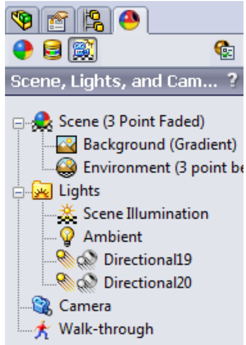
Note: In most cases, the floor of the scene will align properly without any additional steps, however for training purposes this model is aligned differently.

4 Editing the scene.

Select the DisplayManager tab in the FeatureManager design tree, then select **View Scene, Lights and Cameras** .

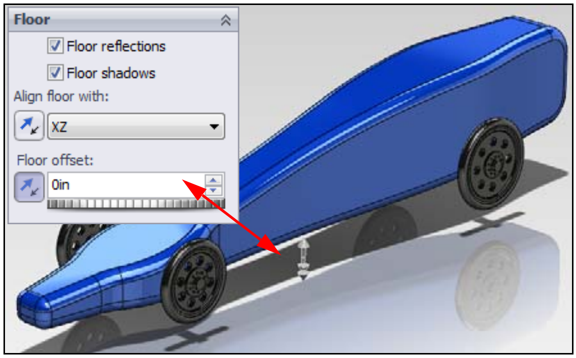
Right-click **Scene** and click **Edit Scene**.

Select the **Basic** tab.

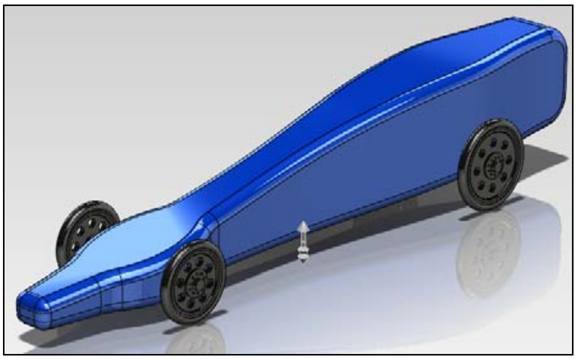


5 Adjust the floor.

The floor is just a planar surface under the model. If needed, we could add an appearance directly to this floor surface. The floor also has the ability to show shadows and reflections.



The floor offset from the model can be adjusted either through the **Edit Scene** PropertyManager or by manually dragging handles in the graphics area.



Using either the drag handle or the **Floor offset** setting in the PropertyManager, move the floor until it just touches the wheels.

Click **OK** to accept the changes.

Rendering

Rendering is the process of calculating the appearance, scene, lighting and decal information of the model by tracing the light rays through the scene. Rendering applies all options set within PhotoView 360.

Review Renders

As rendering can take a considerable amount of time for large images with many refined options, the model rendering can be done in a preview window or right in the graphics area. The choice of which preview option to use is a personal choice as both work equally well. In the following examples, the Preview Window will be used for consistency.

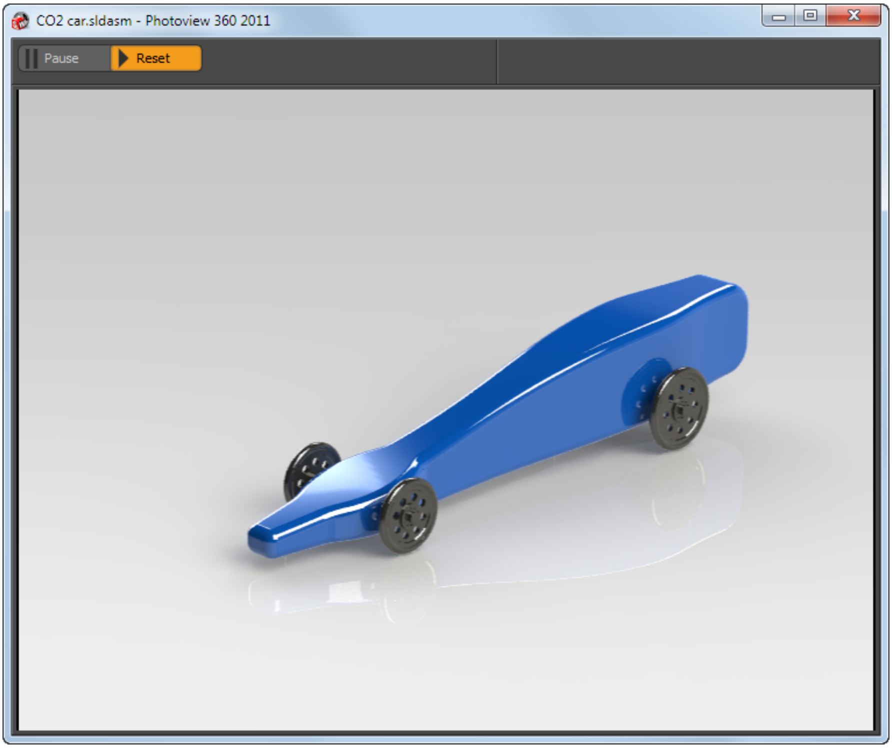
Preview renders are done by a progressive rendering technique where the entire screen or window is redrawn and then refined until the selected quality is achieved.

Final Render

Once you are satisfied with the view, appearances, scenes, etc. as viewed in the Preview Window, the model must actually be rendered. This final rendering is done in the Final Render window. Once created, the image can then be saved to any number of image file types.

6 **Preview Render.**
Click **Preview Render** on the Render Tools toolbar, or click **PhotoView 360, Preview Window**.

The model is rendered to the Preview Window and we can see how the model will look if we do a full rendering. Note that while we had a distinct shadow under the model from the directional light created with the scene in RealView, the shadow is not currently visible in the rendered preview even though the model is well lit. We can also see the reflections of the wheels in the shiny sides of the car.



Now that we have a first look at the rendered model, we will make some additions that will add decals to the side of the car and also add the shadow under the car back into the picture.

The Preview Window may be left open. If it is in your way, minimize it.

Decals

Decals are artwork that are applied to the model. They are in some ways like texture appearances in that they are applied to the surface of the part, feature or face.

Decals can have parts of the image masked out. Masking enables the appearance of the underlying part to show through the decal image.

Decals can be made from a variety of image files including but not limited to:

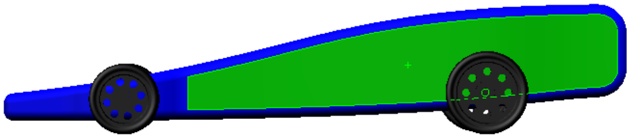
- Windows bitmap (*.bmp)
- Tagged Image File (*.tif)
- Joint Photographic Expert Group (*.jpg)

The size and position of a decal is controlled with the Decal PropertyManager. The Decal PropertyManager, like the other PropertyManagers we've seen, has a series of tabs to control different aspects of applying the decal.

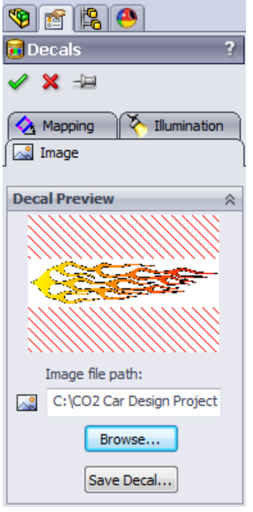
Decals are normally applied to the part rather than in an assembly.

1 **Change to a right-side view.**
This view will make it easier to position the decal.

Select the large face on the side of the car body.



2 **Apply the Decal.**
Click **Edit Decal**  on the Render Tools toolbar, or click **PhotoView 360, Edit Decal**.



The Decal PropertyManager opens.

3 **Select a decal image.**
Under **Image file path**, click **Browse...**.

Select Colored Flame.bmp from the CO2 Car Design Project\Images folder.

Click **Open**.

4 **Save as a decal file.**
PhotoView 360 stores decal files with the extension *.p2d.

Click **Save Decal...**

Normally decals would be saved in the ...\SolidWorks\data\graphics\Decals folder. However, you might not have write permission for the disc drive and/or folder where SolidWorks is installed.

Therefore, save the decal in the CO2 Car Design Project\Images folder.

Name the decal file Colored Flame and click **Save**.

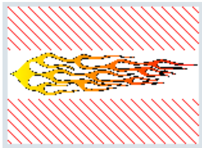
5 New folder.

There was only one folder open in the **Decal Editor**, and it is not the folder where the new decal is located. SolidWorks asks,

The folder where you have chosen to save this decal is not currently visible in the Decals folder of the PhotoView 360 Items in the Task Pane. Do you wish to make the folder visible?

Click **Yes**. This will add the Images folder to the Task Pane.

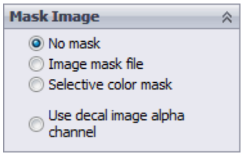
Masks



The preview shows the graphic of the flame on a white background. The actual image file is long and thin. The red crosshatch indicates the portion of the image window that is not part of the decal.

Masks are used to selectively remove part of the decal image file. All image files are rectangular. Without masks, we could only apply the entire rectangular image.

Masking Techniques



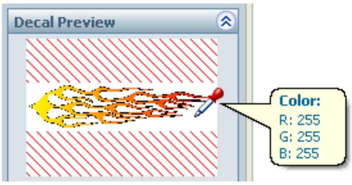
No mask will apply the entire rectangular decal.

Image mask uses an existing gray scale image to selectively mask out parts of the decal.

Selective color mask is used to manually choose which colors in the decal will be masked out. This is a quick way to create the mask if the decal is a simple word or logo, or is against a single background color.

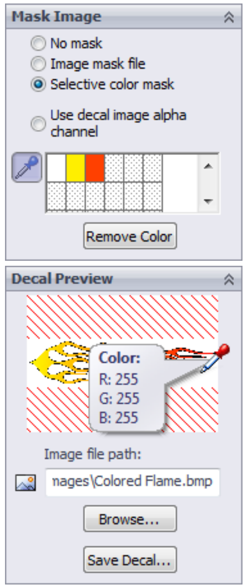
Use decal image alpha channel may be used when the image file used for the decal has alpha channel information. Only certain file types that can be used as decals can have an alpha channel.

Selecting a Color



Individual colors from the preview may be selected for masking. With the **Pick color** tool, the desired mask color is selected from the decal image preview.

That means, any pixels in the decal which have the selected color will be treated as transparent.



The cells in the **Selected Colors** grid display the chosen mask colors.

To deselect a color, you would pick the color in the **Selected Colors** grid, and click **Remove Color**. This unmasks the selected color in the decal. That is, any pixels in the decal which have that color will be opaque.

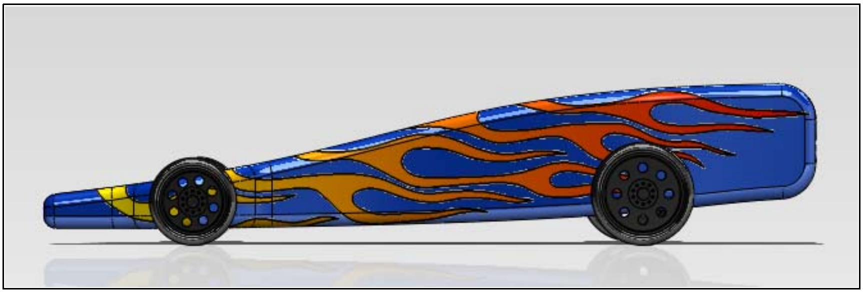
6 Create a mask.

We only want to apply the flame itself as the decal, not the white background. The mask will filter out the white background.

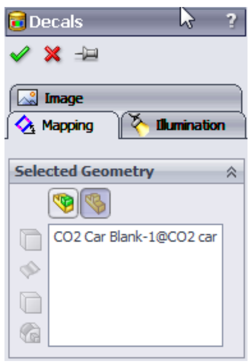
Click the **Pick color** button.

Select the white background from the decal image preview above. It should be the only color selected.

The model view updates to show the new decal representation.



Don’t click **OK** yet, we still have some work to do.



7 Mapping.

Select the **Mapping** tab. The decal is not positioned or scaled very well for the model. We will make some adjustments to this now.

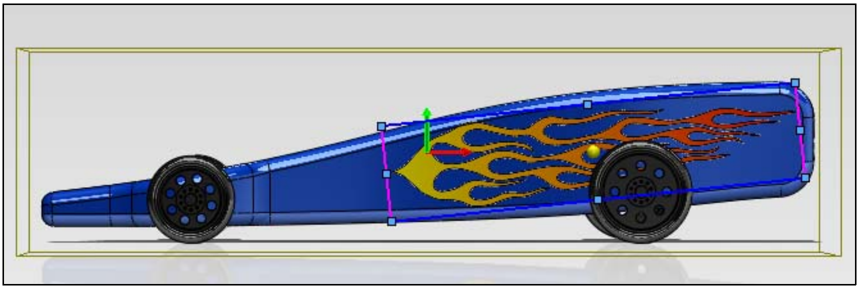
If not already set, change **Mapping type** to **Projection**, and **Projection Direction** to **Current View**.

Set the model in the graphics view in the **Right** view orientation, and click **Update to Current**. This sets the decal on this (right side) planar orientation.

Use the graphics view decal frame to move, resize and rotate the decal appropriately (approximate placement).

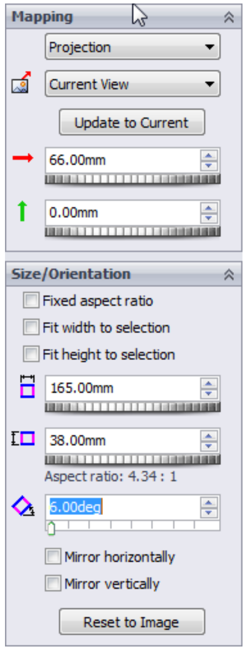
Dragging edges or anywhere inside the frame moves the image, dragging corners resizes, and dragging the center ball rotates.

Clear the **Fixed aspect ratio** check box to stretch the decal so it is longer without making it higher.



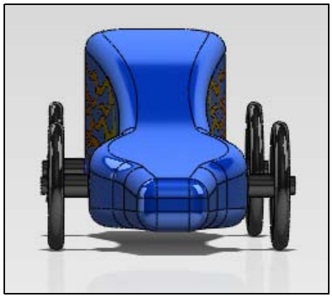
In the **Mapping** and **Size/Orientation** areas, these values can be set more precisely if desired, as follows:

- **Horizontal location: 66.00 mm**
- **Vertical location: 0.00 mm**
- **Fixed aspect ratio: cleared**
- **Fit Width to Selection: cleared**
- **Fit Height to Selection: cleared**
- **Width: 165.00 mm**
- **Height: 38.00 mm**
- **Rotation: 6.00 deg**



Do not click **OK** yet. We still have more work to do.

What You Can't See Doesn't Matter



One curious effect of projection mapping is that the texture, in this case the decal, goes all the way through the object it is applied to. That means it shows up on the other side of the car.

With rendering, we are only concerned about how the picture looks. Remember, this isn't real. When you are applying textures and appearances, you only have to be concerned about what the "camera" sees. If something is hidden from view, don't worry about it.

In this case, we can use projection mapping to our advantage because it can give us the appearance of having the decal on both sides of the car. But that only matters if the view *shows* both sides of the car.

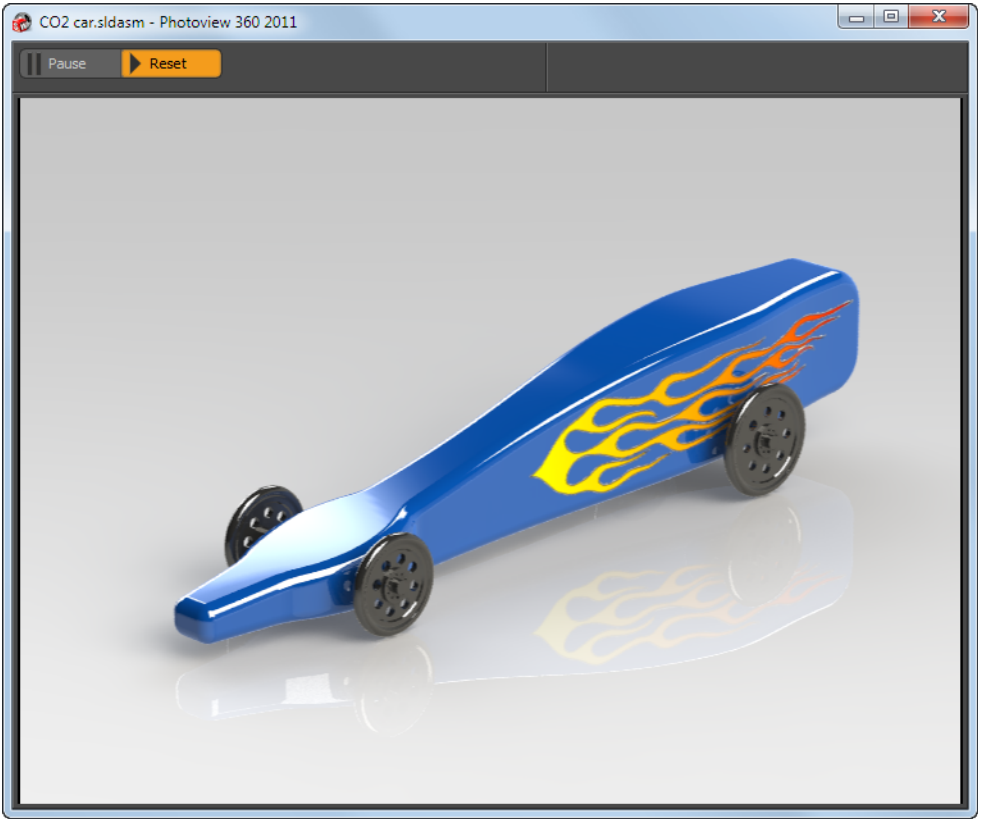
8 Reset the view.
Press the **Spacebar** to bring up the **View Orientation** palette.

Double-click `Render01` to change back to the view we saved for rendering.

9 Preview Render.

Examine the preview render in the Preview Window

The flames add some interest but they look a little dull. We can also see that the rear wheel is not reflected in the decal the same way it is from the blue surface.



What Makes an Image Look Realistic?

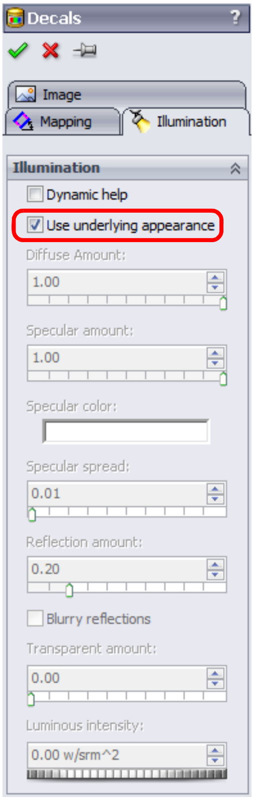
The human eye and brain are very keen at observing and very hard to fool. That is why special effects companies spend millions of dollars trying to develop ever more sophisticated ways of generating computer graphics for special effects shots in movies. It would be very difficult to make a rendering so realistic that people would mistake it for an actual photograph. But by understanding how we interpret certain aspects of an image, we can make improvements.

Reflections

Highly reflective surfaces are visually more interesting when there are details in the environment for them to reflect. In this case, the glossy paint reflects the floor and the wheels. There are also specular highlights from the environment. These are visual cues that enhance the realism of the image.

However, they can also work against us. Look closely at the highlights on the blue appearance. The highlight is only on the blue paint and not on the decal. Also, the reflection of the floor is not continuous across the decal. This is because the illumination appearance of the decal is different from the blue body appearance.

10 Illumination.
Click on the **Illumination** tab.



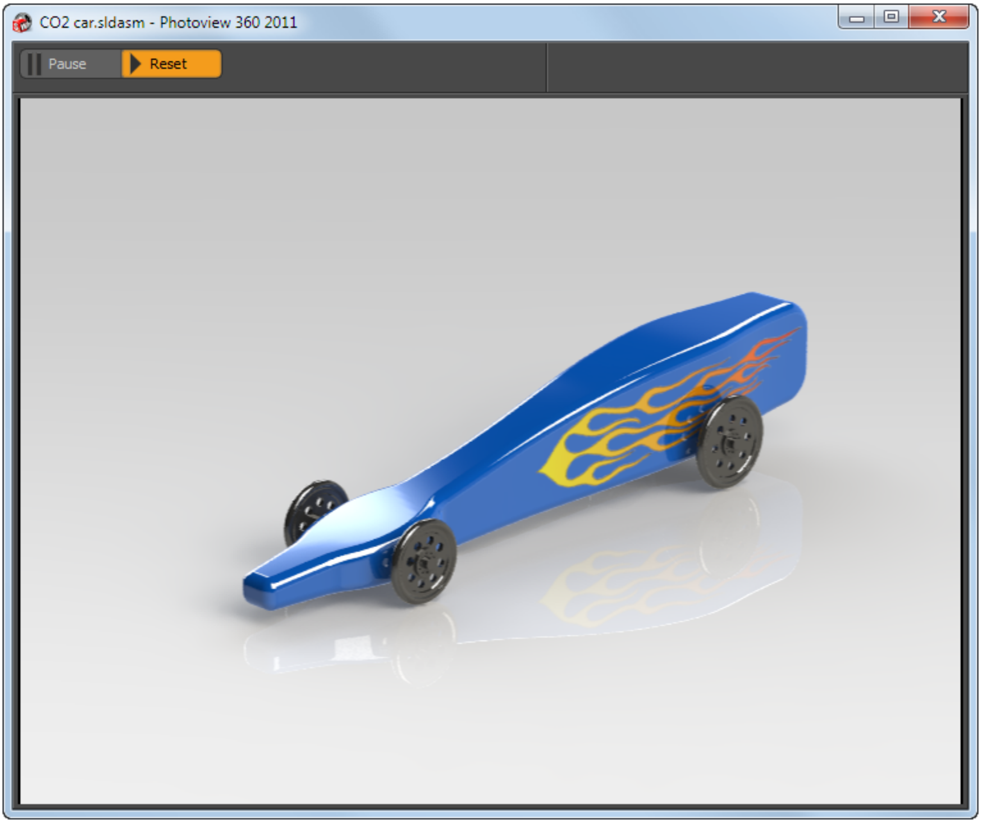
When the decal is applied, it has its own set of illumination properties that can be adjusted separate from the properties of the underlying surface. As this decal is applied to a very shiny surface, we would like the decal to be shiny as well. Rather than manually duplicate the properties of the underlying surface, we can select one option and it will keep the decal's properties the same as those of the surface it is applied to.

In the Decal PropertyManager, select **Use underlying appearance**.

Click **OK**.

11 Preview.

Examine the Preview Window. Check the reflections of the rear wheel and you can see that the wheel is now reflected exactly the same from the decal and the blue surface. This gives the impression that both the blue color and decal are under a shiny clear coat.

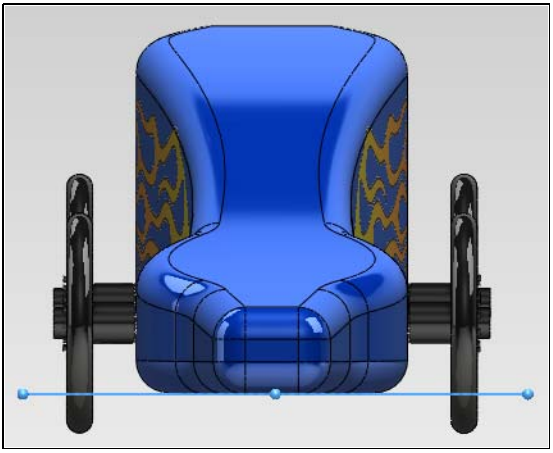


Props

Photographers and artists include props in their work to provide visual interest and to give the subject a sense of context. You can do the same thing with a rendering.

We Need a Real Floor

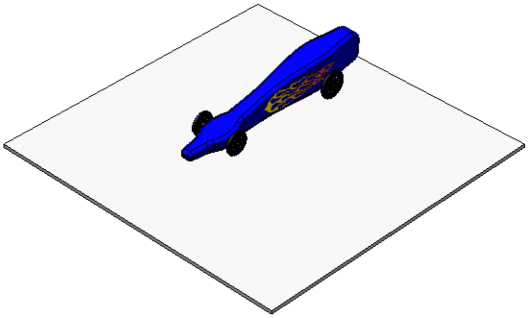
We want to add a couple of CO_2 cartridges as props. We want to have them sitting on the floor next to the car. However, the “floor” we see in the rendering is a virtual floor. It doesn’t exist as geometry in the assembly.



Since the origin of the car body is mated to the origin of the assembly, the assembly’s Top plane cannot serve as a substitute floor because it is located above the bottom of the wheels.

The solution is to add a floor for the purposes of adding and mating the CO_2 cartridges.

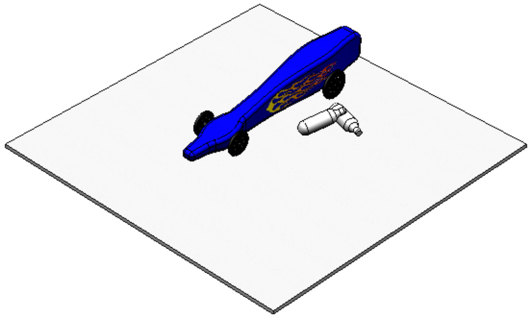
12 Add a floor.
Click **Insert Components** .



Click **Browse** and select the Floor.sldprt file from the CO2 Car Design Project folder.

Add a **Parallel** mate between the Top reference plane of the Floor and the Top reference plane of the assembly. This will keep the Floor from tilting.

Add a **Tangent** mate between the uppermost face of the Floor and the bottom of the wheels. This locates the Floor vertically.



13 Add two instances of the CO2 Cartridge.
Use a combination of mates, **Rotate Component**, **Move Component**, and **Physical Dynamics**, to position the cartridges.

Strive for an interesting arrangement that is not too "mechanical".

Tip: When you get one of the cartridges positioned the way you want it, right-click it and select **Fix** from the shortcut menu. This will lock it in place so it doesn't move when you are positioning the second cartridge.

14 Hide the Floor.
We do not want to include the Floor component in the rendering.

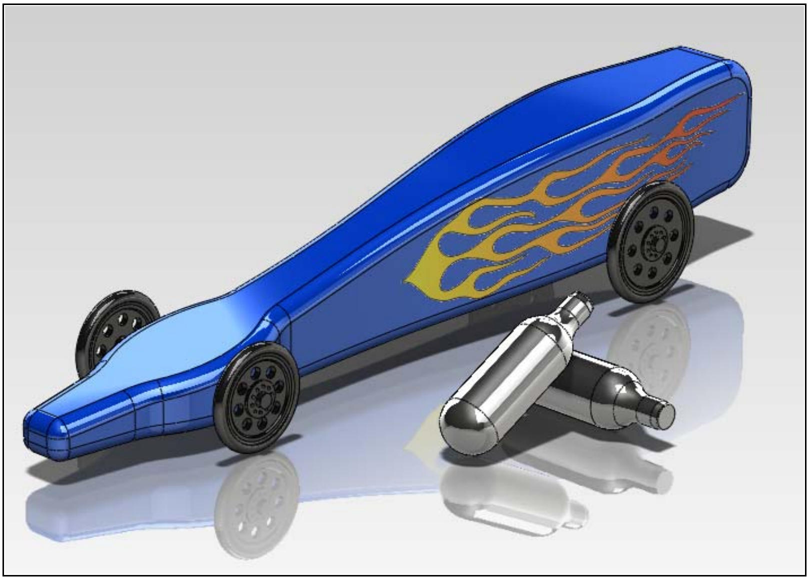
Right-click the Floor and press **Hide**. We could have also deleted the Floor, however by just hiding it, it can be used again easily if we decide to add additional props.

15 Fix the cartridges.
If you didn't fix the cartridges in step **13**, do so now.

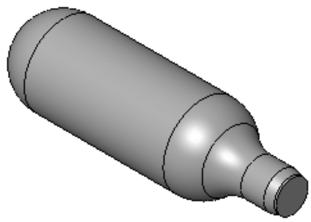
Press **Ctrl**. Select the two instances of the CO2 Cartridge. Right-click and select **Fix** from the shortcut menu.

This will prevent them from accidently being moved.

16 Change view.
Reset the view to the saved state `Render01`.



17 Open the `CO2 Cartridge` part.
Right-click one of the `CO2 Cartridge` components and select **Open Part** from the shortcut menu.



18 Add appearance.
Select the **Appearances, Scenes and Decals** tab in the Task Pane, then expand the Metal and then Brass folders.

Drag the **polished brass** appearance into the graphics area, this will apply the appearance to the entire part.

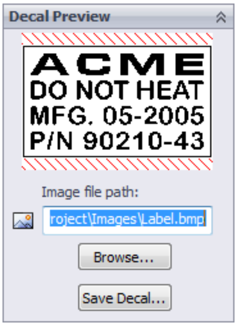
19 Add a decal.
Select the cylindrical face of the cartridge.

Click **Edit Decal** on the Render Tools toolbar.

20 Create a decal from an existing file.

Browse... to `Label.bmp` from the `CO2 Car Design Project\Images` folder.

Click **Open**.



21 Save the decal file.

Save the decal file (**Save Decal...**) to the `CO2 Car Design Project\Images` folder.

Name the decal file `Label`.

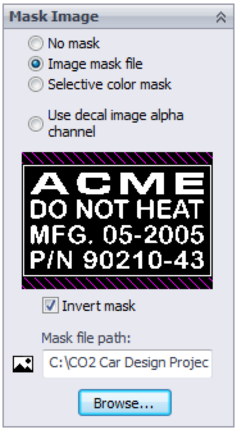
Click **Save**.

22 Create a mask.

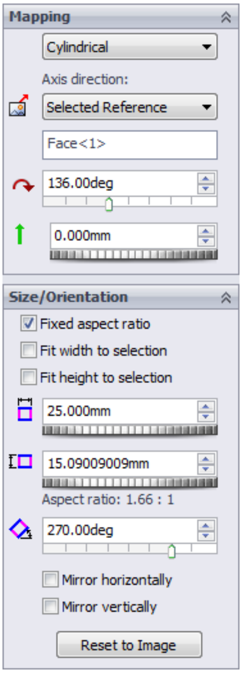
We need to create a mask to filter out the white background so that only the black letters show when the decal is rendered. As the image is already black and white, we can use the same image as a mask by just reversing the colors.

Select **Image Mask File**. Click **Browse** and select the same `Label.BMP` image as the decal file.

Select **Invert mask**, this will reverse the black and white of the image so that in the mask, the letters are white (transparent) and the background is black (opaque).



23 Adjust the mapping.
Click the **Mapping** tab.



For **Mapping type**, **Cylindrical** should already be selected because we applied the decal to a cylindrical face.

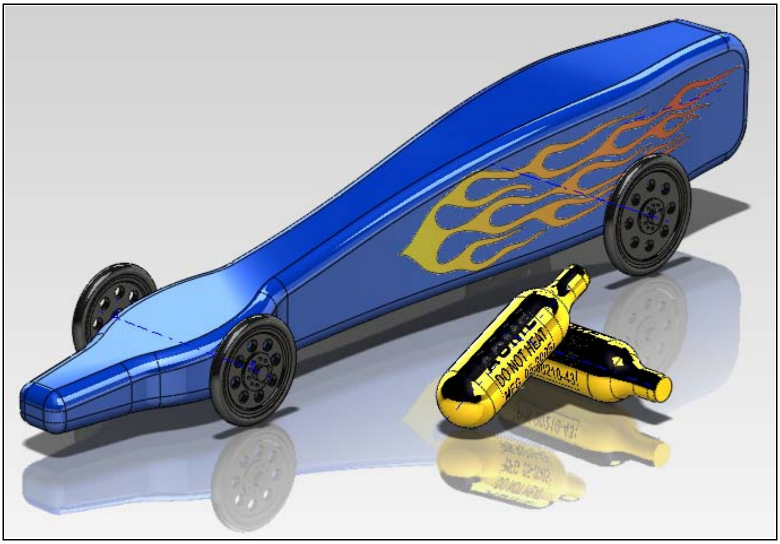
Because we are applying the decal at the part level instead of in the assembly, the **Axis direction** should already be set to **Selected Reference**. `Face<1>` represents the cylindrical face of the cartridge you selected in step **19**.

It is not critical that your settings match what is in the book. The important thing is to look at the graphics view preview and adjust the settings to achieve the look you want.

When you are satisfied, click **OK**.

24 Switch back to the assembly document.
Click **Window, CO2 Car Baseline.sldasm** to switch back to the assembly.

When you add a decal at the part level it appears on the face of both cartridges.



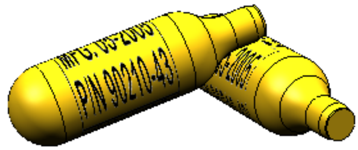
25 Fine tune the position of the label.

We can't see the label as well as we'd like on one of the cartridges. We have two options:

- Rotate the cartridge around its centerline axis so the label is more visible.
- Edit the mapping of the decal to position it better. The problem with this approach is that the decal will move on both of the cartridges because it was added at the part level.

We will rotate the cartridge, not the decal.

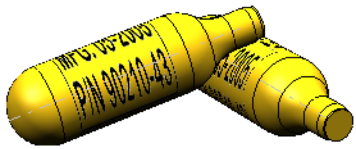
26 Hide the car.



Press **Ctrl** and select the car body and all four wheels.

Click **Hide/Show Components** on the Assembly toolbar.

27 Temporary axis.



Click **View, Temporary Axes**.

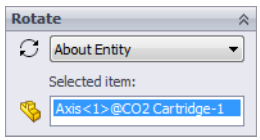
This make the temporary axes that are associated with all cylindrical features visible. We will rotate the cartridge around this axis.

28 Float the cartridge.

In step **15** on page 157 we fixed the cartridges so they wouldn't move. Before we can rotate the cartridge, we have to float (unfix) it.

Right-click the cartridge you want to rotate and select **Float** from the shortcut menu.

29 Rotate component.



Select the cartridge you want to rotate.

Click **Rotate Component** on the Assembly toolbar. Select **About Entity** and then select the temporary axis.

Rotate the cartridge until the decal is visible the way you want it.

Click **OK**.

30 Fix the cartridge.

Right-click the cartridge and select **Fix** from the shortcut menu.

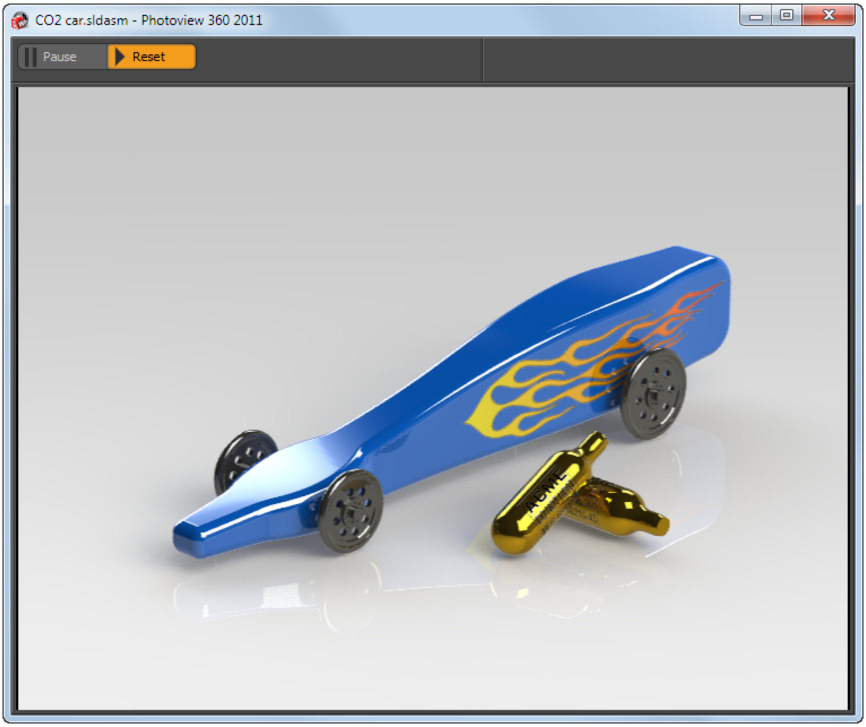
31 Turn off the temporary axes.

Click **View, Temporary Axes** to toggle off the display of the temporary axes.

32 Show the car.
In the FeatureManager design tree, select the car body and the four wheels.

Click **Hide/Show Components** on the Assembly toolbar to toggle on the display of the components.

33 Preview Render.
The props make the rendering more interesting and put the car in context – it is powered by a CO_2 cartridge.



Shadows

As our model is now pictured, we are using a studio set up. The model is shown against a seamless background of a single color as compared to an image of the model placed in a scene where it might be on a table top or on the race track.

When rendered in PhotoView 360, any discrete lights in the scene are turned off as the default. As all the light comes from large light sources in the scene, the shadows are very soft and not very well defined. To make the scene more interesting, we can turn on one of the directional lights in PhotoView 360 and have it cast a shadow to give a more realistic effect.

Lighting

Proper lighting can greatly enhance the quality of a rendering. The same principles used by photographers work well in PhotoView 360.

Lights are created and positioned in SolidWorks. PhotoView 360 has separate options to control the brightness of the lights separate from that used in SolidWorks as well as controls to show shadows and their quality.

Types of Lights

There are four types of lights in SolidWorks: Ambient, Directional, Point, and Spot. In this exercise, we will only deal with one type, the directional light:

Directional light comes from a source that is infinitely far away from the model. It is a collimated light source consisting of parallel rays arriving from a single direction, like the sun. The central ray of a directional light points toward the center of the model.

Currently the `Lights` folder in the DisplayManager contains one `Ambient` light and two `Directional` lights.

You can turn the ambient light on or off, but you cannot delete it or add additional ambient lights. The ambient light is not used in rendering.

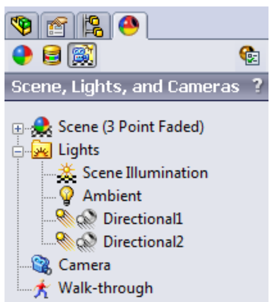
You can turn the directional lights on or off, or delete them. You can also add additional directional light sources.

The maximum number of light sources in any document is nine (the ambient light and eight others in any combination).

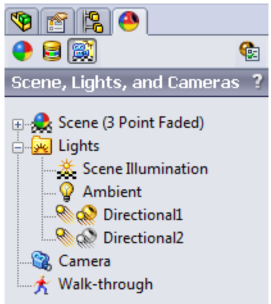
1 **Turn on a directional light.**
In the DisplayManager, select **View Scene, Lights and Cameras** .

Expand the `Lights` folder.

Notice that next each directional light that there are two icons. The left icon indicates the light's status in SolidWorks and the right icon the status in PhotoView 360. Like other icons in SolidWorks, a colored icon indicates on and a gray icon indicates off.



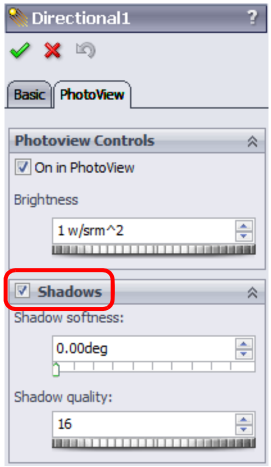
Right-click the light `Directional1` and click **On in PhotoView**.



The right icon is now colored , indicating the light is on in PhotoView 360.

2 Turn on shadows.

In the DisplayManager, double-click the light `Directional1`. This will open the PropertyManager for the light. The PropertyManager has two tabs. The **Basic** tab controls the lights visual properties for OpenGL and RealView. Also on this tab are controls to position the light.



Select the **PhotoView** tab. This tab has a control for the brightness of this light in PhotoView 360 and is completely independent of the **Brightness** control on the **Basic** tab.

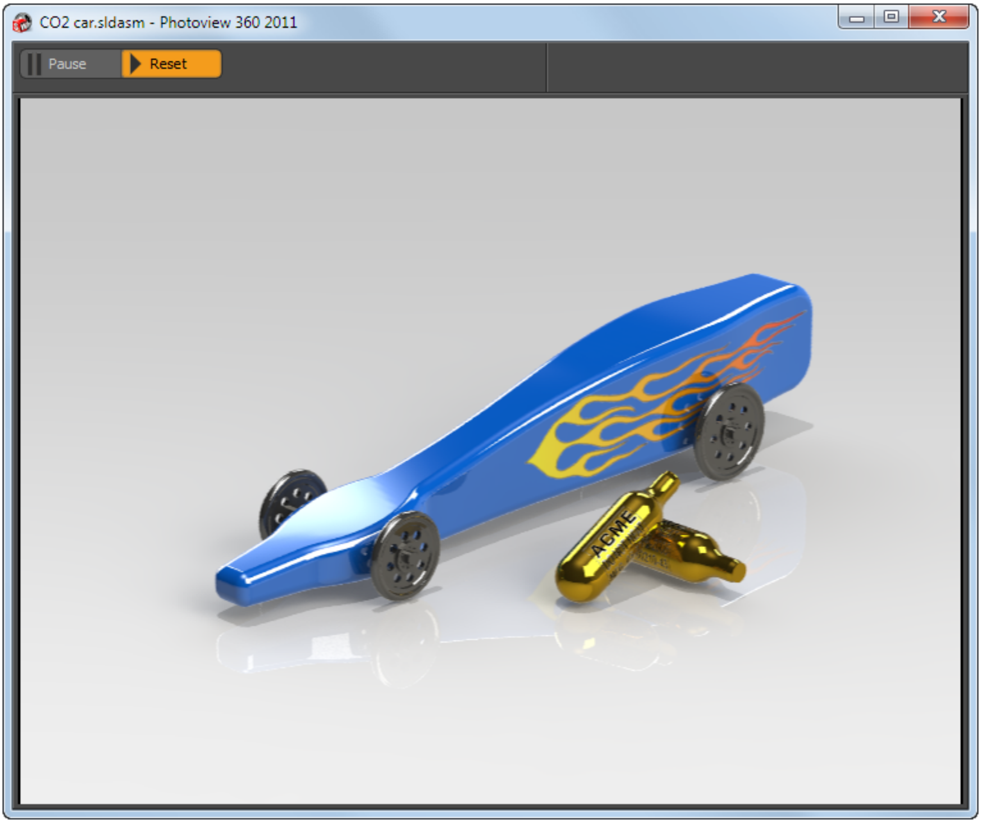
Select **Shadows**. This will turn on the shadows for just this one light when the image is rendered.

Click **OK**.

3 Preview Render.

We can now see a very hard shadow under the car and the cartridges from the directional light.

Are we done? There is no firm answer to that question as rendering is very subjective and you could continue to adjust and tweak the settings to get something else. For now however, we have a good setup for the final rendering.

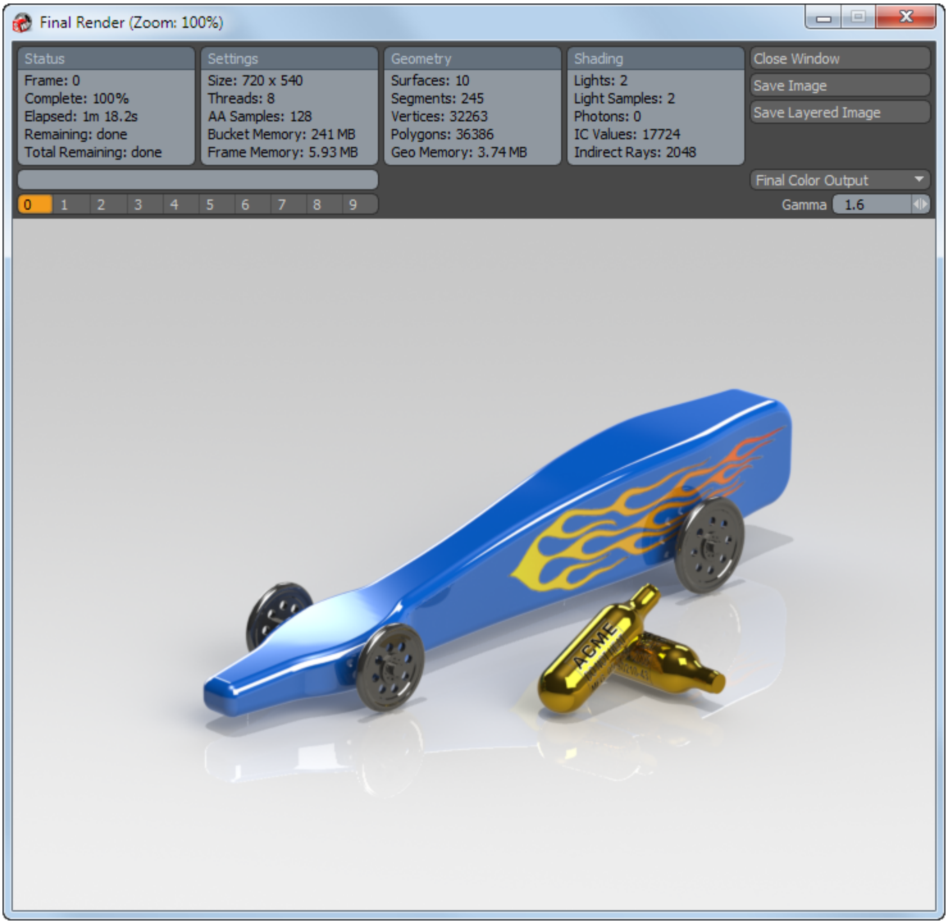


4 Final render.

Now that we have everything setup the way we want, we need to turn the image into a form that is usable in other programs.

Click **Final Render** on the Render Tools toolbar.

The Final Render window will open and rendering will begin. There will be four irradiance passes where you will see the a partial image appear in four stages as PhotoView 360 calculates all the lighting, reflection and refractions in the scene. After the irradiance passes, the scene will be rendered in little squares which are referred to as buckets. The number of buckets will be based on the number of computer CPU cores and threads that are available in your computer.



Output Options

When the image is first rendered, it is only visible on the computer screen in the Final Render window. PhotoView 360 keeps the last ten rendered images available for recall. To make these images usable in other programs, they must be saved as an image file.

Image files can be used for many purposes, including web pages, training manuals, sales brochures, and PowerPoint® presentations.

Rendered image files can be further manipulated with other software to add lettering, effects or make adjustments beyond the capabilities of the PhotoView 360 software. This is known as the post-production phase.

File Types

Rendered images can be saved to the following file types:

- Flexible Precision Image Format (*.flx)
- TARGA (*.tga)
- Windows Bitmap (*.bmp)
- Radiance High Dynamic Range HDR (*.hdr)
- JPEG (*.jpg)
- Portable Network Graphic PNG (*.png)
- SGI RGB (*.SGI)
- TIFF (*.tif)
- Open EXR (*.exr)

Methods to Increase Rendering Quality

The quality of the image file can vary depending on the options chosen in both SolidWorks and PhotoView 360. Generally speaking, rendering quality and rendering time are directly proportional. Some choices to improve image quality are listed below.

Note: Not all of these options were covered during this introduction to rendering. For more information, ask your teacher about getting a copy of *Photorealistic Rendering using SolidWorks and PhotoView 360*. It is available from your school's value-added SolidWorks reseller.

- Increase SolidWorks image quality.
  PhotoView 360 uses the tessellated data of the shaded SolidWorks models when importing those models for rendering. Increasing shaded image quality reduces jagged edges on curved surfaces.

- Increase the PhotoView 360 rendering quality.
  PhotoView 360 has four levels of render quality: **Good**, **Better**, **Best** and **Maximum**. Each increase essentially doubles the rendering time of the previous quality setting. Except to achieve very specific rendering goals, Best is usually a sufficiently high enough quality for most purposes.
- Increase the number of pixels rendered.
  Set the number of pixels rendered to a larger number in the PhotoView 360 options.
- Increase shadow quality.
  Increasing shadow quality improves the edges of shadows.

How Many Pixels to Render

For the highest quality output with the most efficient file size, we need to determine the correct size to render the image. As a general rule, do not scale up bitmap images. This causes loss of definition. Images may be scaled down, but the original file will be larger than necessary.

Dpi Versus Ppi

Dots per inch (dpi) and pixels per inch (ppi) are sometimes used interchangeably, but they are actually different. Dots per inch are the number of dots printed per linear inch. Pixels per inch measures the resolution of an image projected on a display. As it takes multiple ink dots to create most colors, the pixels per inch that are rendered should be less than the dots per inch the printer can create.

Calculating Correct Number of Pixels

Question: How do you calculate the number of pixels to render for the final output?

Answer: Work backwards from the output.

For general reference, web images use a resolution of 96 dpi. Newspapers use resolutions from 125 dpi to 170 dpi. High-quality brochures and magazines use resolutions from 200 dpi to 400 dpi. For books, the range is generally from 175 dpi to 350 dpi. PowerPoint presentations are normally 96 ppi.

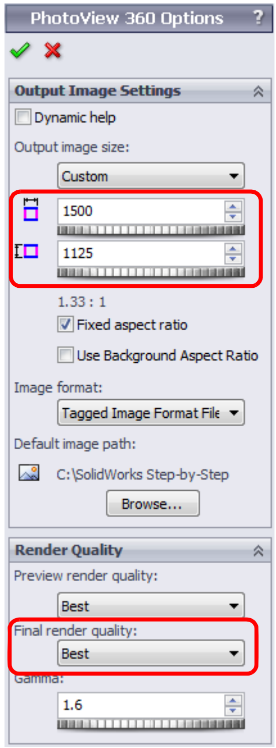
If the output will be to a printer, and you want to make the image look like a photograph, you may need a printer capable of 600 or 1200 dots per inch and a rendered image of 300 pixels per inch.

Multiply the size of the final printed image by the quality desired. The correct number of pixels can be entered directly in the PhotoView 360 options.

Example #1

Suppose we want to include a rendering of the CO_2 Car in a Microsoft Word report and would like a high quality image of 300 ppi. We want the image to be 5 inches wide and 3.75 inches high.

Multiplying the size of the desired image times the ppi gives 1500 by 1125 pixels.



1 **Set size and quality.**

Click **Options** on the Render Tools toolbar.

Enter a width of **1500** and a height of **1125**.

For image format, choose **Tagged Image Format File TIF** from the list. TIFF files provide good print quality, but result in a large file size but with excellent definition.

Select **Best** for the **Final Render Quality**.

Click **OK**.

2 **Render.**

We have to render the image again because we changed the settings.

Click **Final Render** on the Render Tools toolbar.

3 **Save the image.**

Now that the image is rendered, it must be saved to a separate file to be usable in Microsoft Word.

Click **Save Image** in the upper right corner of the Final Render window.

Name the file `CO2 Car.tif` and save it to the `CO2 Car Design Project` folder. As we selected Tagged Image Format File TIF in the PhotoView 360 options, that is the file type selected. If we needed the file to be saved to other formats, we could just change the file type here.

Click **Save**.

4 **Examine the file.**

Locate the file `CO2 Car.tif` and open it in any image viewer to check your work.

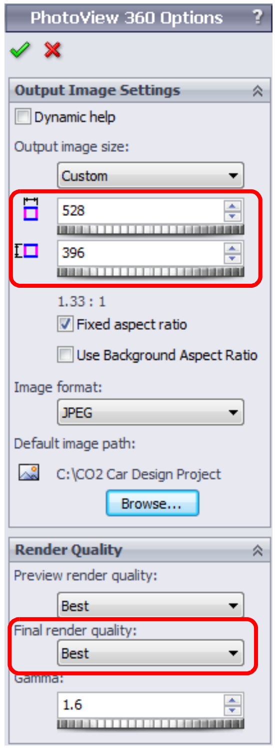
Example #2

Suppose we want to incorporate our rendering into a PowerPoint presentation. PowerPoint presentations generally use images that are 96 ppi. We want the image to be 5.5 inches wide.

To maintain the same aspect ratio, calculate the correct height: $\frac{5}{3.75} = \frac{5.5}{NewHeight}$

Solving, we get $3.75 \times 5.5 = 5 \times NewHeight$ or $20.625 = 5 \times NewHeight = 4.125$

Multiplying the size of the desired image times 96 dpi gives 528 by 396 pixels.



5 Set size and quality.

Click **Options** on the Render Tools toolbar.

Enter a width of **528** and a height of **396**.

For image format, choose **JPEG** from the list. JPEG files are compressed and much smaller than the higher quality types. They are one of the preferred file types for use on the web or cases where a lot of images are required.

Select **Best** for the **Final Render Quality**.

Click **OK**.

6 Render.

Click **Final Render** on the Render Tools toolbar.

7 Save the image.

Now that the image is rendered, it must be saved to a separate file to be usable in Microsoft PowerPoint.

Click **Save Image** in the upper right corner of the Final Render window.

Name the file CO2 Car.jpg and save it to the CO2 Car Design Project folder.

Click **Save**.

8 Examine the file.

Locate the file CO2 Car.jpg and open it in any image viewer to check your work.

Notice that there is a large difference in file size between the TIF and JPEG images. Part of the difference is the size of the images as the TIF image has more than eight times the number of pixels. The remainder of the difference is from the file types.

9 Save and close.

Save and close all open files.