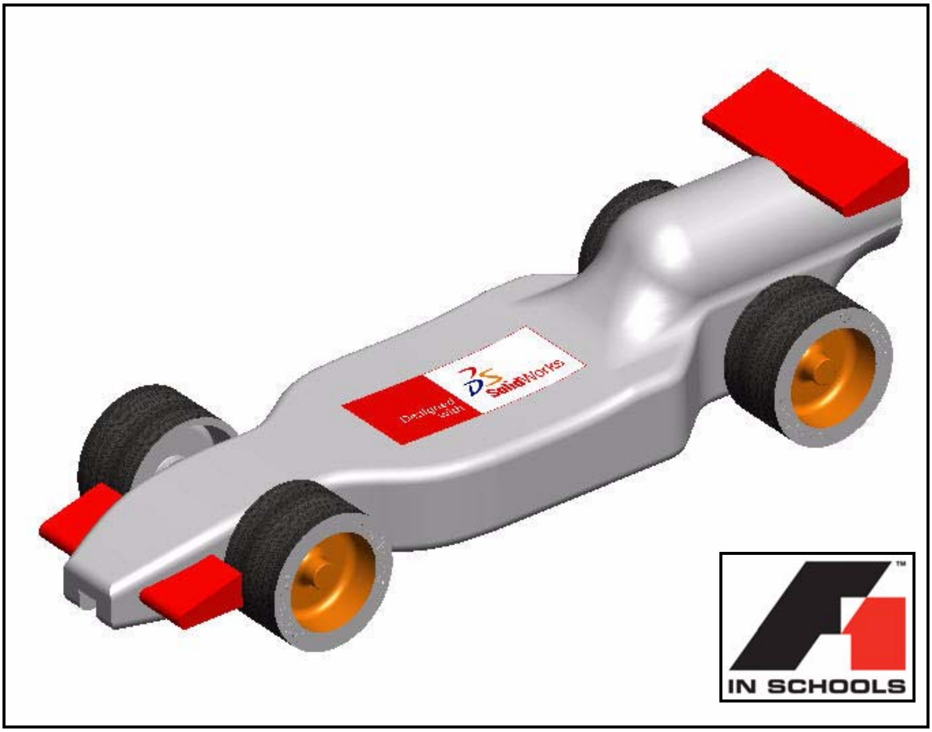
*Engineering Design
and Technology Series*

F1 in Schools™ Design Project with SolidWorks® Software



For Type-R Cars

Dassault Systèmes SolidWorks Corp.
300 Baker Avenue
Concord, MA 01742 USA
Phone: 1 800 693 9000

Outside the US: 1 978 371 5011
Fax: 1 978 371 7303
info@solidworks.com

Table of Contents

Lesson 1 Introduction

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

- Understand how to use this book for the *F1 in Schools™ Design Project* for R-Type cars
- Start a SolidWorks session
- Download the required files, folders, and models for this project
- Add the folder `Race Car Design Project files` to the SolidWorks Design Library in the Task Pane

Using This Book

The *F1 in Schools™ Design Project* helps you learn how to apply 2D and 3D SolidWorks modeling principles and techniques to create a `Race Car` assembly and drawing and apply the SolidWorks SimulationXpress and SolidWorks Flow Simulation analysis tools.

You will be learning by doing as you complete the lessons in this book:

- Ability to create a SolidWorks session
- Understanding of the SolidWorks user interface and toolbars
- Aptitude to open parts and create a 3D `Race Car` assembly
- Create a detailed multi-sheet, multi-view drawing of the `Race Car` assembly
- Apply the Measure and Mass tool
- Apply PhotoWorks
- Apply Analysis tools: SolidWorks SimulationXpress and SolidWorks Flow Simulation

What is SolidWorks Software?

SolidWorks is design automation software. In SolidWorks, you sketch ideas and experiment with different designs to create 2D and 3D sketches, 3D models, 3D assemblies, and 2D drawings using the easy to learn Windows® graphical user interface.

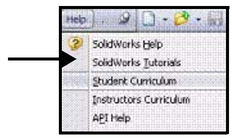
SolidWorks is used by students, designers, engineers and other professionals around the world to produce simple and complex parts, assemblies, and drawings.

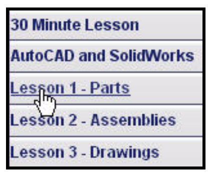
Prerequisites

Before you begin the *F1 in Schools™ Design Project* you should review and complete the following SolidWorks tutorials that are integrated in the SolidWorks software:

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

Click **Help**, **Student Curriculum** to access the Race Car Design Project folder. Click **Help**, **Instructors Curriculum** to access the Educator Resources.





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 5: Drawing Basics

Conventions Used in This Book

This manual uses the following typographical conventions:

Convention	Meaning
Bold Sans Serif	SolidWorks commands and options that you select, appear in this style. Example 1: **Extruded Boss/Base** means click the Extruded Boss/Base tool from the Features toolbar. Example 2: **View**, **Origins** means click View, Origns from the Menu bar menu.
`Typewriter`	Files and Folder names appear in this style. Example 1: `Race Car Design Project`. Example 2: `Sketch1`.
17 Do this step.	The steps in the lessons are numbered in sans serif bold.

Before You Begin

Copy the `Race Car Design Project` folder from the SolidWorks website onto your computer before you start this project.

1 Start a SolidWorks session.
Click **All Programs**, **SolidWorks**, **SolidWorks** from the Windows Start menu. The SolidWorks application is displayed.

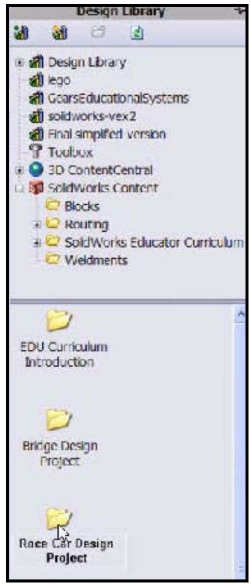
Note: If you created the SolidWorks icon on your desktop, click the icon to start a SolidWorks session.

2 Copy the Race Car Design Project folder.

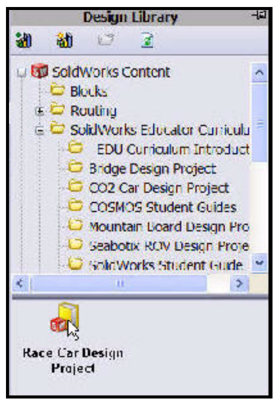
Click the **SolidWorks Resources** tab from the Task Pane.

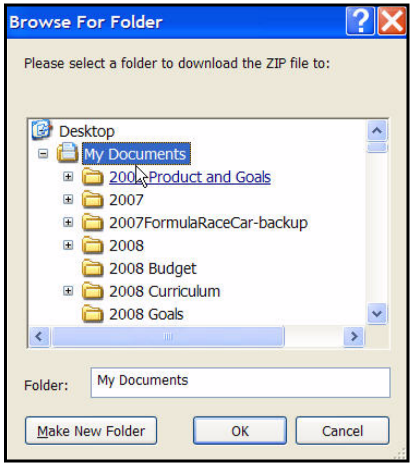
Click the `Student Curriculum` folder as illustrated.

Double-click the `Race Car Design Project` folder.



Ctrl-click the `Race Car Design Project` folder. A dialog box is displayed.





3 Download the zip file.

Press the **Ctrl** key.

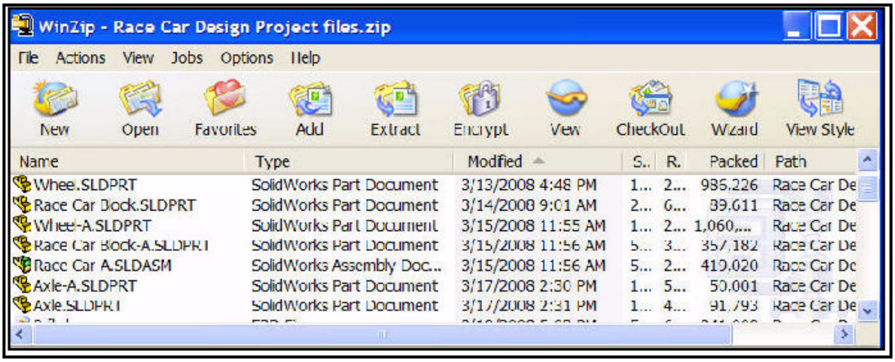
Click the **icon**. You will be prompted for a folder in which to save the Zip file.

Tip: Ask your teacher where you should save the zip file. Remember where you saved the downloaded zip file.

4 Open the zip file.

Browse to the folder where you saved the zip file.

Double-click the `Race Car Design Project files.zip` file.



5 Extract all files.

Browse to the location where you want to save the files. The system will automatically create a folder named `Race Car Design Project files` in whatever location you specify.

Note: For example, you might want to save it in the My Documents folder.

You now have a folder named `Race Car Design Project files` on your hard drive. The data in this folder is used in this book.

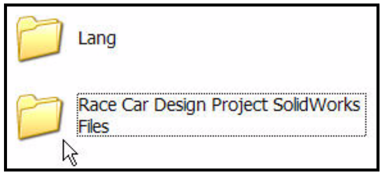
Add the Folder to the Design Library Path

The SolidWorks Design Library is a convenient way to access the partsused in the exercises. It is more efficient than clicking File, Open from the Menu bar menu and browsing for a file. Add the `Race Car Design Project files` folder to the search path of the Design Library.

The `Race Car Design Project files` folder contains two sub-folders:

- `Lang`
- `Race Car Design Project SolidWorks files`

1 Open the Task Pane.
Click the **Design Library** tab.



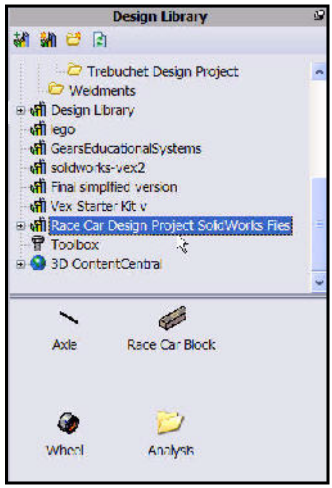
2 Add folder.
Click the **Add File Location** tab from the Design Library.

Browse to where you extracted the project files in step 4 page 5.

Click the `Race Car Design Project files` folder.

Click the `Race Car Design Project SolidWorks files` folder.

Click **OK**.



3 Results.
The contents of the `Race Car Design Project SolidWorks files` folder is now accessible through the SolidWorks Design Library.

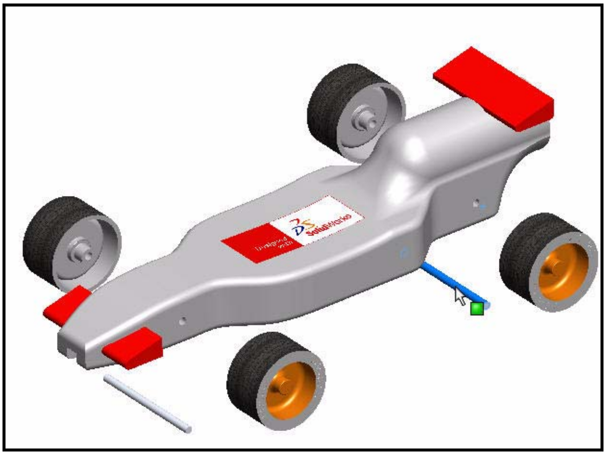
Note: Visit www.f1inschools.co.uk for updated design requirements and specifications along with free SolidWorks software.

Lesson 2
Designing the Race Car

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

- Describe important factors to the performance of a CO_2-powered `Race Car`
- Create the `Race Car` assembly from an existing model using the following Feature and Sketch tools: Extruded Boss/Base, Extruded Cut, Fillet, Line, Sketch Fillet, Smart Dimension, Mate, Explode, and Rotate Component
- Insert components into a new assembly
- Apply Standard mates between components in the `Race Car` assembly
- Create an Exploded configuration of the `Race Car` assembly
- Apply the `Mass Properties` tool
- Apply the `Measure` tool
- Open Parts from the `Race Car` assembly
- Confirm the required `Race Car` dimensions for Type-R with the Rules and Regulations of the F1 in Schools™ Design Project contest

Important Design Considerations



Within the framework of the *F1 in Schools™ Design Project* contest specifications, there are a few factors to keep in mind when it comes to building a winning car. These are:

- **Friction**
  Energy used to overcome friction is energy that isn't being used to accelerate your `Race Car`. Sources of friction include:
  - Wheels and axles: if the wheels do not spin freely, the `Race Car` will be slow.
  - Misaligned axles: if the axle holes are not drilled perpendicular to the centerline of the car, the car will have a tendency to turn to the left or right. This will cost you speed and the contest!
  - Misaligned screw eyes: if the screw eyes are not positioned and aligned properly, the guideline can drag on them, the car body, or the wheels. This can slow the car 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**
  There is a finite amount of thrust 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 car is one way to build a faster car. Keep in mind that the contest specifications stipulates a minimum mass of 55grams for the vehicle.
- **Aerodynamics**
  The air exerts a resistance, or drag, as the car tries to move through it. To minimize drag, your car should have a smooth, streamlined shape.

Note: Check in the back of this lesson for a summary of the required design requirements for your `Race Car` assembly. Visit www.f1inschools.co.uk for updated design requirements and specifications.

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 to harvest it.

Start SolidWorks and open an existing part

1. **Start the SolidWorks application.**
Click **All Programs, SolidWorks, SolidWorks** from the Start menu. The SolidWorks graphics area is displayed.

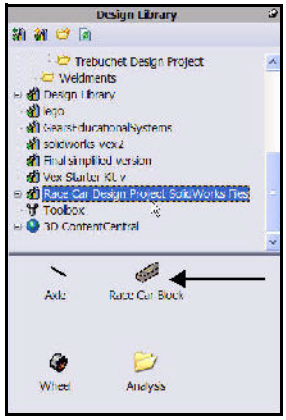
2. **Open the Design Library.**
Click the **Design Library** tab from the Task Pane.

3 Open the Race Car Block.

Click the **Race Car Design Project SolidWorks Files** folder located in the Design Library.

The contents of the folder is displayed in the lower portion of the Design Library window.

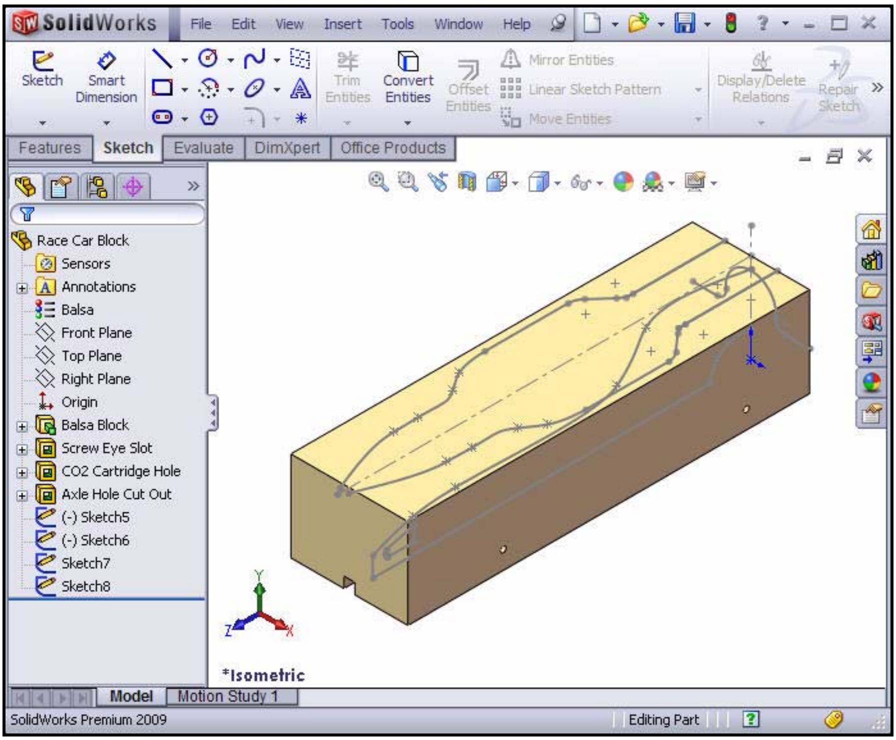
Drag and drop the part named **Race Car Block** into the SolidWorks graphics area. View the model and the FeatureManager design tree.



Note: This may take 1-5 seconds.

The FeatureManager design tree located on the left side of the SolidWorks window provides an outline view of the active model. This makes it easy to see how the model was constructed.

The FeatureManager design tree and the graphics area are dynamically linked. You can select features, sketches, drawing views, and construction geometry in either pane.

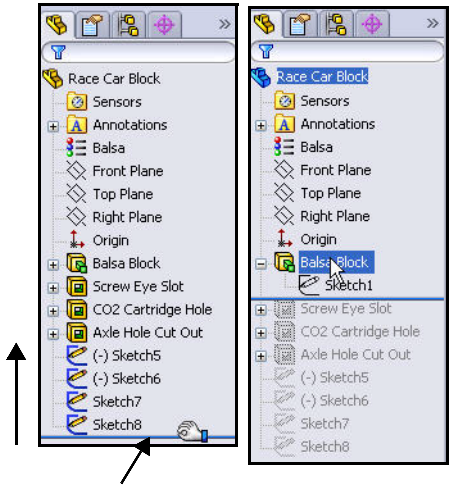


4 Review the created features and sketches in the model.

Drag the **rollback bar** upwards to a position before the `Balsa Block` feature.

The `Balsa Block` feature is displayed.

Double-click the `Balsa Block` feature in the FeatureManager. The feature is displayed in blue in the graphics area and Sketch1 is displayed. View the dimensions. If needed, press the z key to fit the model to the graphics area.



Note: The `Balsa Block` is 223mm x 50mm x 65mm. If you plan to use a fixture to machine your car, you must make sure that your design is no longer that 210mm. Most fixtures have a nose plate that holds the front of the balsa block and if your design is too long, it can/will end up breaking the endmill or possible damaging the fixture.

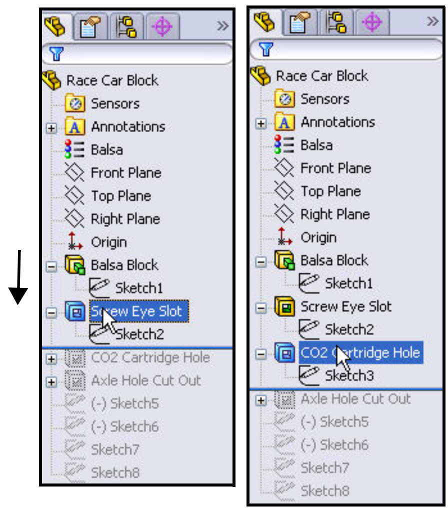
Drag the **rollback bar** downwards to a position before the `Screw Eye Slot` feature.

View the features in the graphics area.

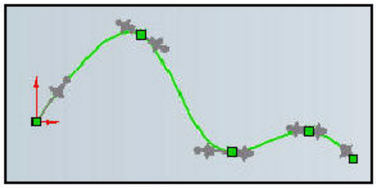
Double-click the `Screw Eye Slot` feature in the FeatureManager. The feature is displayed in blue and `Sketch2` is displayed.

Drag the **rollback bar** downwards to a position before the `CO2 Cartidge Hole` feature. View the features in the graphics area.

Double-click the `CO2 Cartidge Hole` feature in the FeatureManager. The feature is displayed in blue and `Sketch3` is displayed.



Drag the **rollback bar** downwards to a position before the `Axle Hole Cut Out` feature. View the features in the graphics area.



Double-click the `Axle Hole Cut Out` feature in the FeatureManager. The feature is displayed in blue and Sketch4 is displayed.

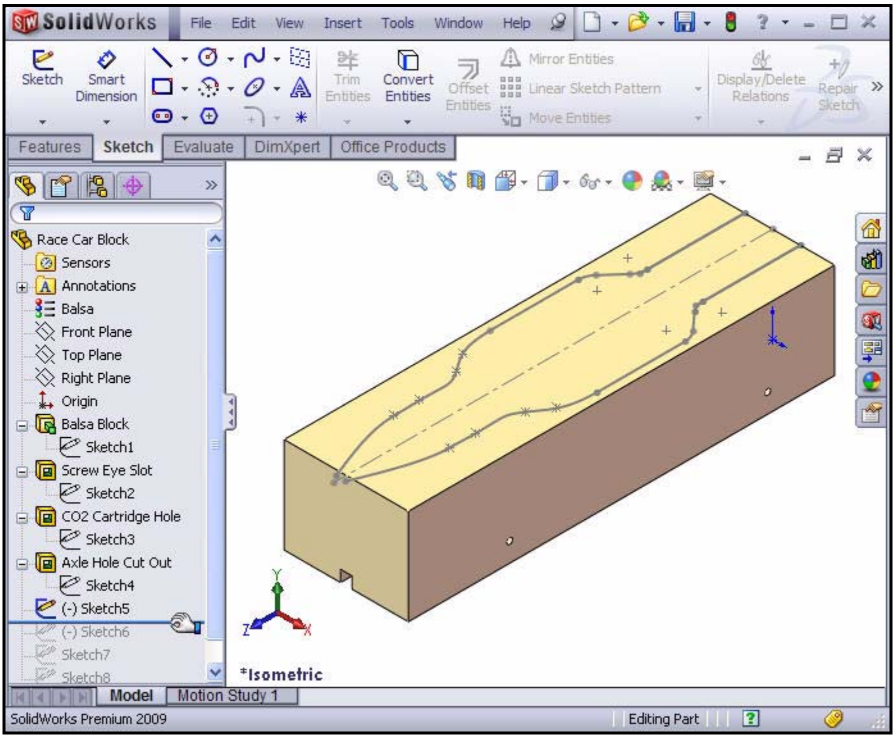
Drag the **rollback bar** downwards to a position before `(-)Sketch5`.

Click `(-)Sketch5` from the FeatureManager. View `(-)Sketch5` in the graphics area.

`(-)Sketch5` is the sketch of a Spline. Splines are used to sketch curves that have continuously changing shape. Splines are defined by a series of points between which the SolidWorks software uses equations to interpolate the curve geometry.

Splines are very useful for modeling free-form shapes, "body of the Race Car" that are smooth.

Note: `(-)Sketch5` is not fully defined, because a spline is free-form and will vary by the designer.

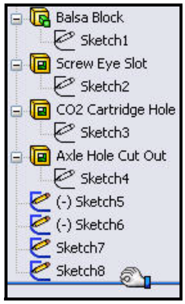


Drag the **rollback bar** downwards to a position below `Sketch8`.

Click `Sketch8` from the FeatureManager.

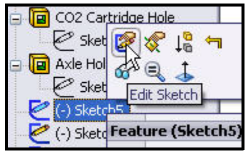
View `Sketch8` in the graphics area.

Click **inside** the graphics area.

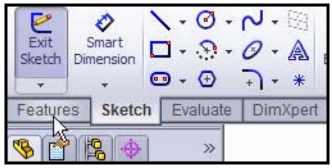


Extruded Cut Feature

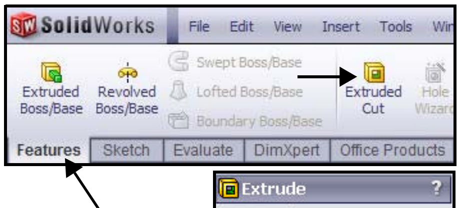
An Extruded cut feature removes material from a part or an assembly. Remove material for the `Race Car Body`.



1 Create the first Extruded Cut Feature. Right-click `(-)Sketch5` from the FeatureManager.



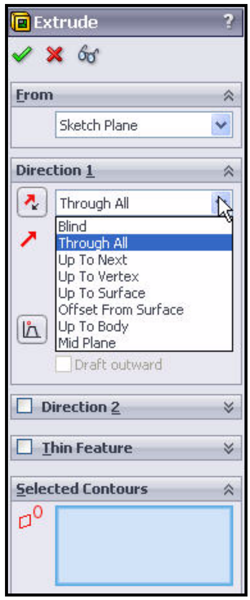
Click **Edit Sketch** from the Context toolbar. The Sketch toolbar is displayed in the CommandManager.



Click the **Features** tab from the CommandManager. The Features toolbar is displayed.

Click the **Extruded Cut** tool from the Features toolbar. The `Extrude` PropertyManager is displayed.

Select **Through All** for the `End Condition` in Direction 1.

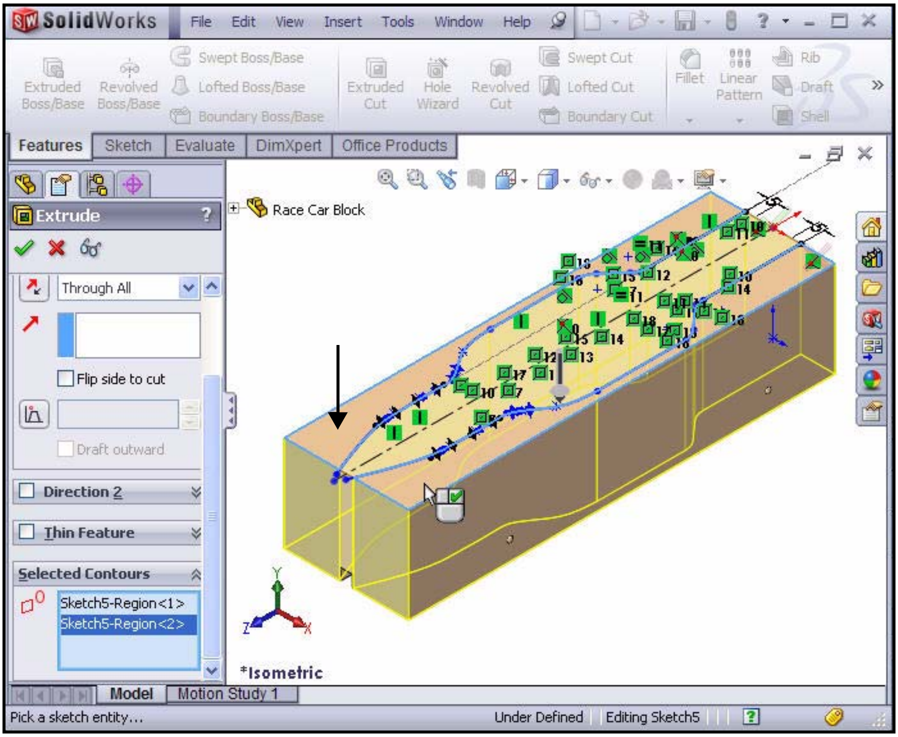


Click the **two surfaces** as illustrated in the graphic area. `Sketch5-Region<1>` and `Sketch5-Region<2>` are displayed in the `Selected Contours` dialog box.

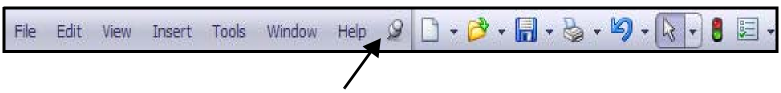
Click **OK** from the `Extrude` PropertyManager.

`Extrude1` is displayed in the FeatureManager.

Click **inside** the graphics area. View the results.

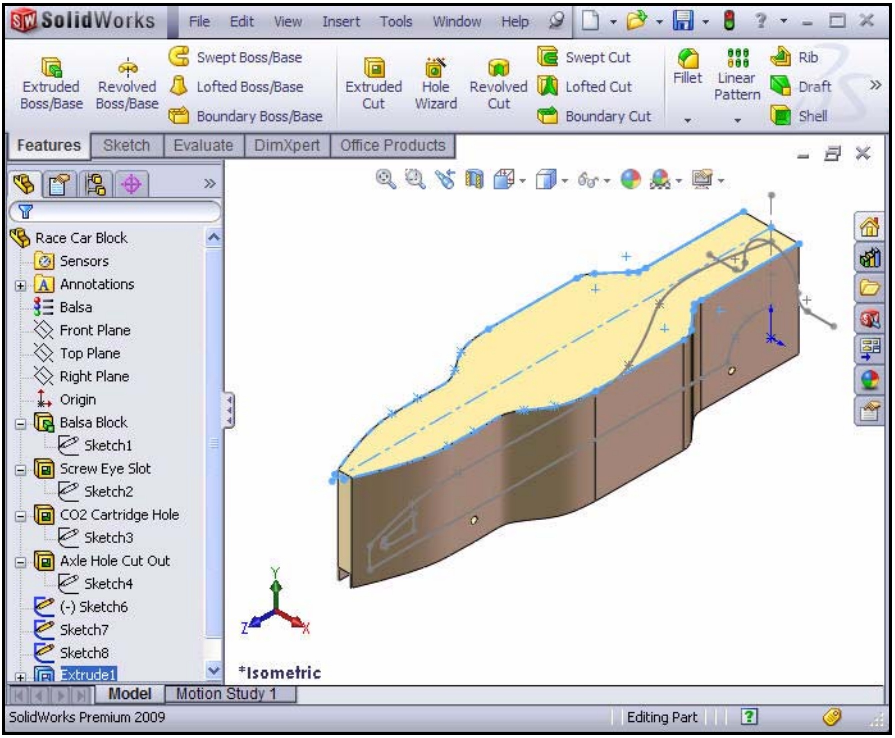


Note: **Pin** the Menu bar toolbar and the Menu bar menu to obtain access to both menus in this book.



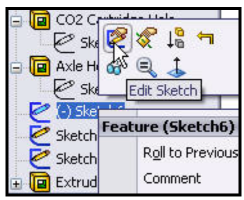
2 Save the model.

Click **Save** from the Menu bar toolbar.



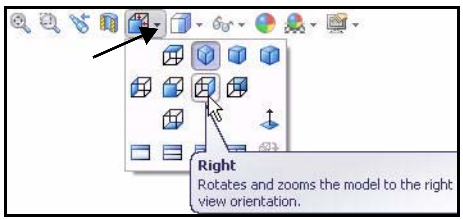
3 Create the second Extruded Cut Feature.

Right-click `(-)Sketch6` from the FeatureManager.



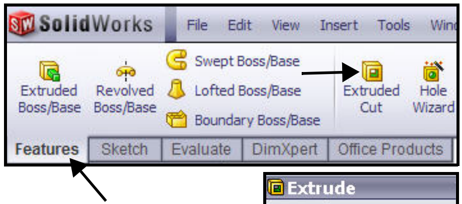
Click **Edit Sketch** from the Context toolbar. The Sketch toolbar is displayed in the CommandManager.

Click **Right** view from the Heads-up View toolbar. The Right view is displayed.

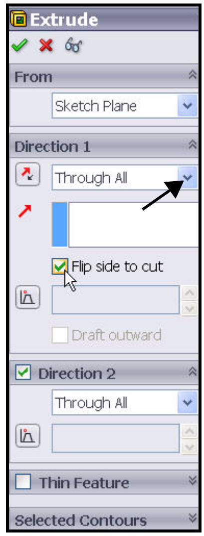


Press the **z** key to Zoom out. Press the **Z** key to Zoom in. Press the **f** key to fit the model to Graphics area.

Click the **Features** tab from the CommandManager. The Features toolbar is displayed.



Click the **Extruded Cut** tool. The `Extrude` PropertyManager is displayed.

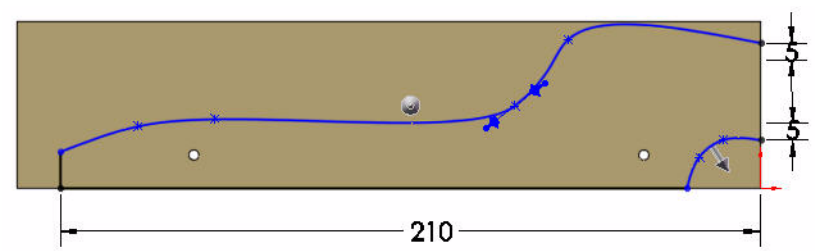


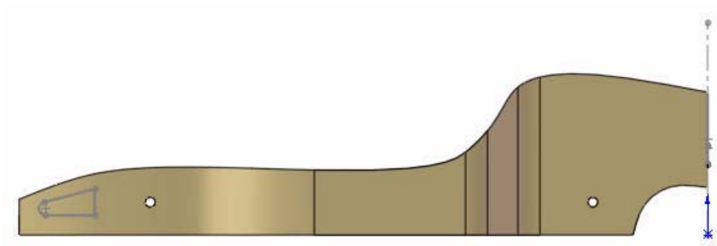
Note: `Through All` is selected for `End Condition` in Direction 1 and Direction 2.

Check the **Flip side to cut** box. View the direction of the extrude.

Click **OK** from the `Extrude` PropertyManager. `Extrude2` is displayed.

Click **Save** from the Menu bar toolbar.



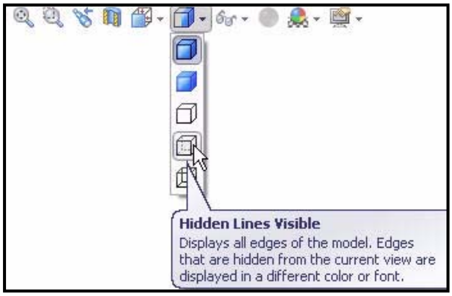


4 Create the third Extruded Cut Feature.

Create the hole for the CO_2 cartridge.

Right-click `Sketch7` from the FeatureManager.

Click **Edit Sketch** from the Context toolbar. The Sketch toolbar is displayed in the CommandManager.

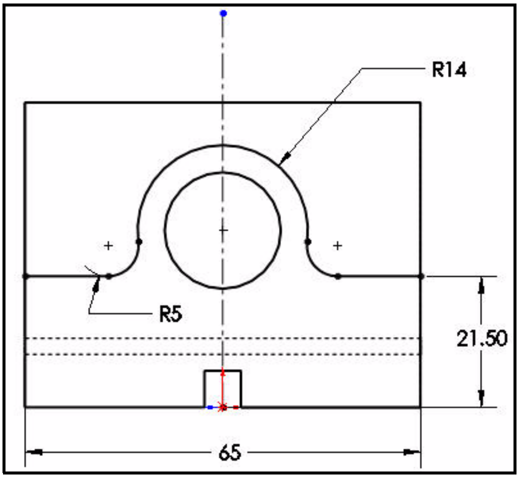


Click **Back** view from the Heads-up View toolbar.

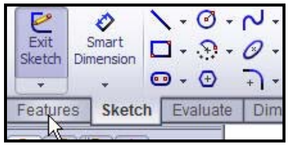
Click **Hidden Lines Visible** from the Heads-up View toolbar.

View the dimensions of the sketch.

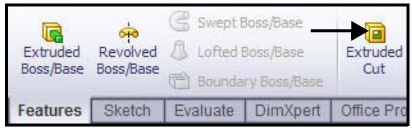
Note: `Sketch7` is the sketch for the CO_2 cartridge hole.



Click the **Features** tab from the CommandManager. The Features toolbar is displayed.

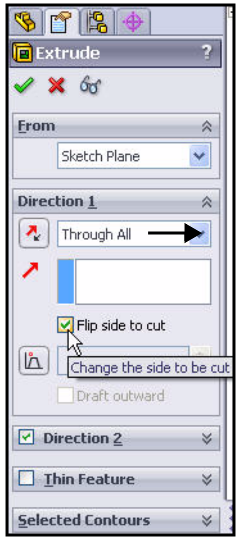


Click the **Extruded Cut** tool. The `Extrude` PropertyManager is displayed.

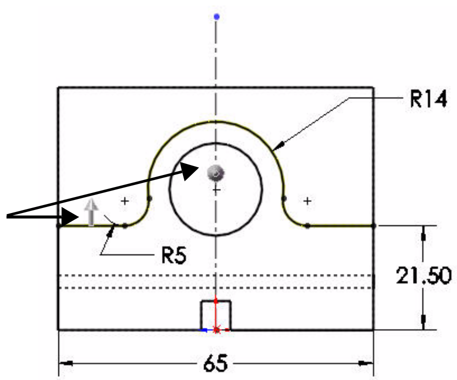


Click **Through All** for `End Condition` in Direction 1 and Direction 2.

Check the **Flip side to cut** box.



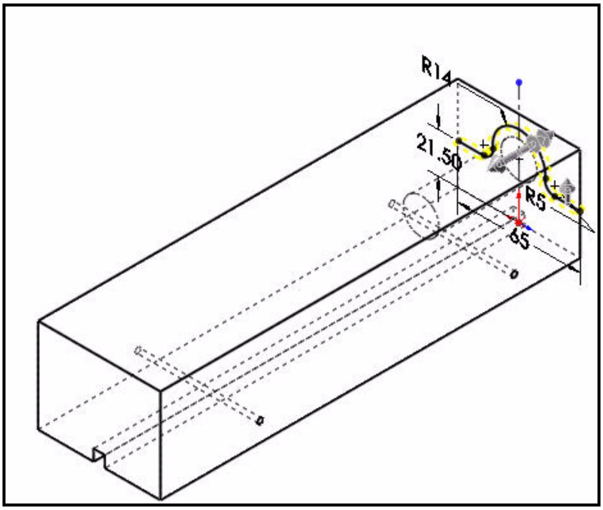
Note: View the direction of the Extrude feature arrows.



Click **Isometric** view from the Heads-up View toolbar. View the Extruded Cut feature.

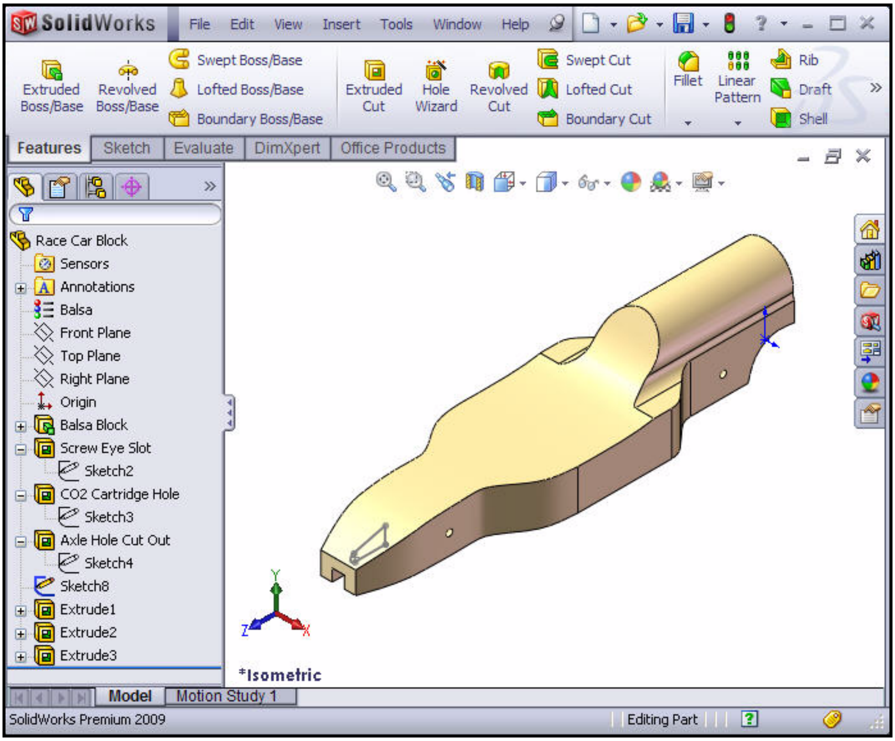
Click **OK** from the `Extrude` PropertyManager. `Extrude3` is displayed.

Click **Shaded With Edges** from the Heads-up View toolbar.



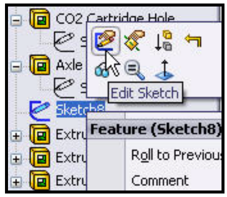
5 Save the model.

Click **Save**.



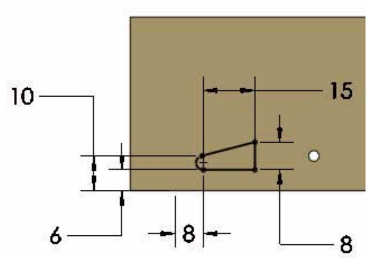
Create the Front Wing

1 Create a MidPlane Extruded Boss Feature.
Right-click `Sketch8` from the FeatureManager. `Sketch8` is the Sketch for the front wing of the car.



Click **Edit Sketch** from the Context toolbar. The Sketch toolbar is displayed in the CommandManager.

Click **Right** view from the Heads-up View toolbar.



Click the **z** key to fit the model in the graphics area.

View the sketch dimensions.

2 Create an Extruded Boss Feature.
An Extruded Boss feature adds material to the model.

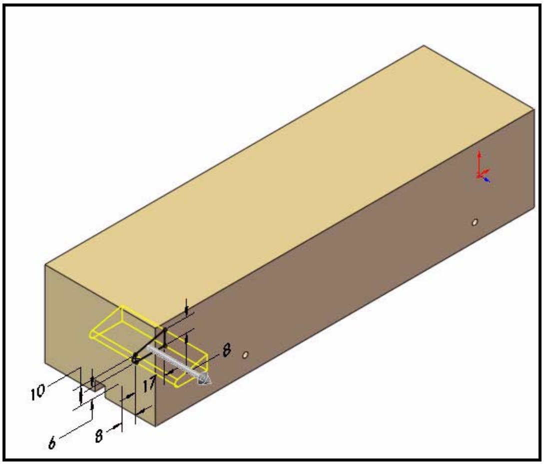
Click the **Features** tab from the CommandManager. The Features toolbar is displayed.

Click **Extruded Boss/Base** from the Features toolbar. The **Extrude** PropertyManager is displayed.

Select **Mid Plane** for `End Condition` in Direction 1.

Enter **50.00**mm for `Depth`.

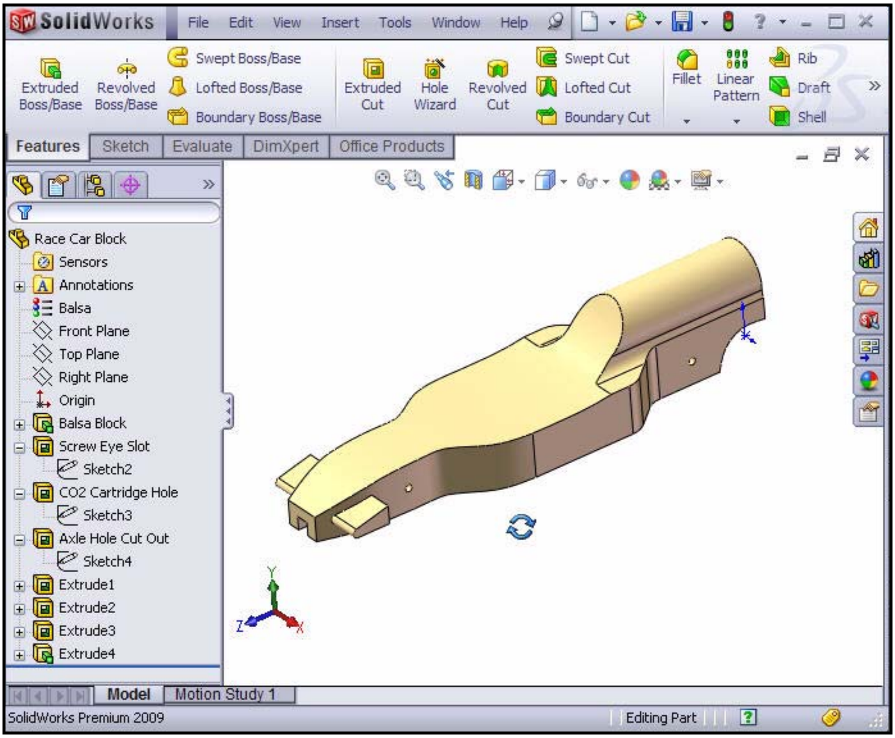
Click **Isometric** view from the Heads-up View toolbar. View the Extruded Boss feature.



Click **OK** from the Extrude PropertyManager. Extrude4 is displayed.

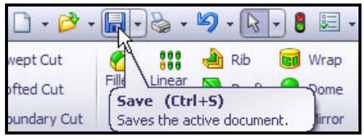
Click **inside** the graphics area.

Note: Use your **middle mouse button** to rotate the model in the graphics area. View the created features.



3 Save the model.

Click **Save** from the Menu bar toolbar.

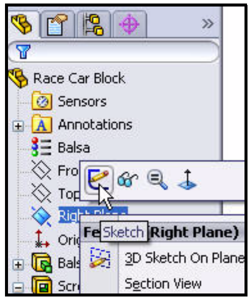


Create the Rear Wing

1 Create a Sketch.

Click **Hidden Lines Removed** from the Heads-up View toolbar.

Right-click **Right Plane** from the FeatureManager.



Click **Sketch** from the Context toolbar. The Sketch toolbar is displayed. Right Plane is your Sketch plane.

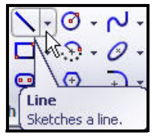
Click **Right** view from the Heads-up View toolbar.

Press the **z** key to fit the model to the graphics area.

Click the **Zoom to Area** tool from the Heads-up View toolbar.

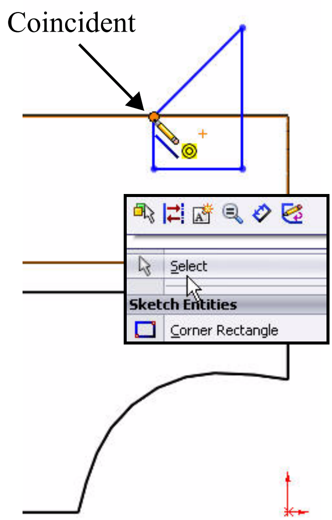
Zoom in on the back of the car as illustrated.

Click the **Zoom to Area** tool from the Heads-up View toolbar to deactivate.



Click the **Line** tool from the Sketch toolbar. The `Insert Line` PropertyManager is displayed.

Sketch **four lines** as illustrated. The first point is Coincident with the top horizontal edge of the car.

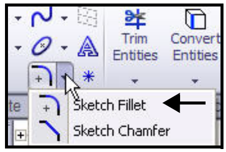


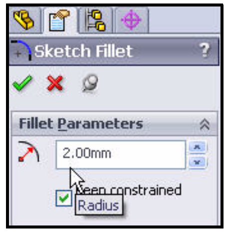
2 Deselect the Line Sketch tool.
Right-click **Select** in the graphics area.

3 Apply the Sketch Fillet tool.

Click the **Sketch Fillet** tool from the Sketch toolbar. The `Sketch Fillet` PropertyManager is displayed.

Enter **2**mm for `Radius`.

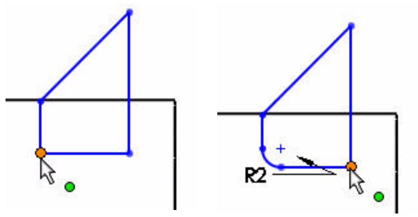




Click the **left endpoint** of the horizontal line.

Click the **right endpoint** of the horizontal line.

Click **OK** from the `Sketch Fillet` PropertyManager.



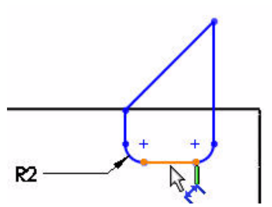
4 Dimension the Rear Wing.

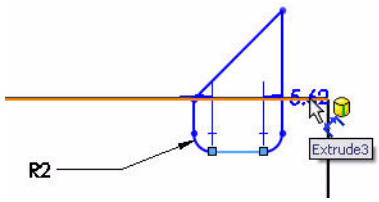
Click the **Smart Dimension** tool from the Sketch toolbar. The Smart Dimension icon is displayed on the mouse pointer.

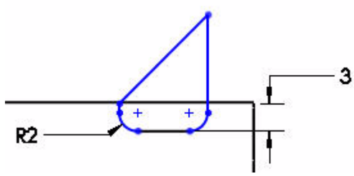
Click the **two** illustrated edges.

Click a **position** to the right.

Enter the **3**mm dimension.



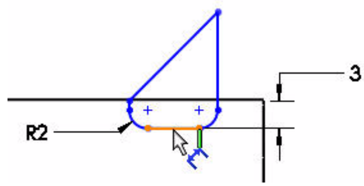


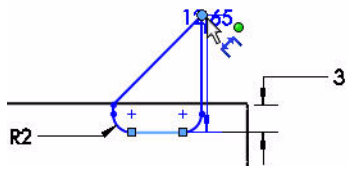


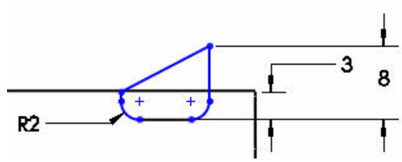
Click the illustrated **edge** and **point**.

Click a **position** to the right.

Enter the **8**mm dimension.



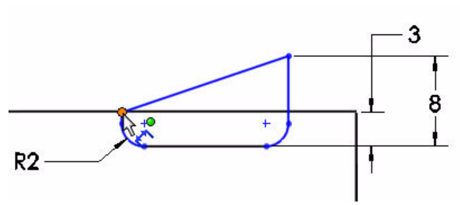


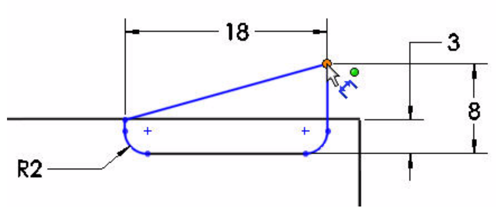


Click the illustrated **two points**.

Click a **position** above the model.

Enter the **18**mm dimension.



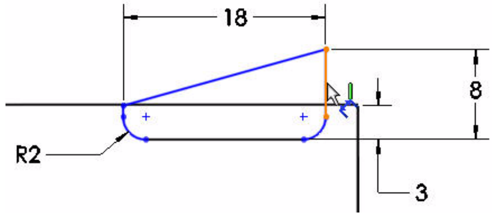


Click the illustrated **two edges**.

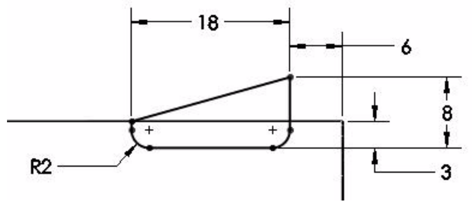
Enter the **6**mm dimension.

Click a **position** above and to the right.

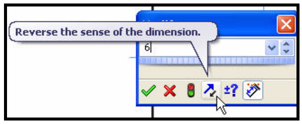
`Sketch9` is fully defined and is displayed in black.



Note: If needed, click the `Reverse the sense of dimension` icon in the Modify dialog box.



Click **OK** from the Dimension PropertyManager.



5 Create an Extruded Boss Feature.

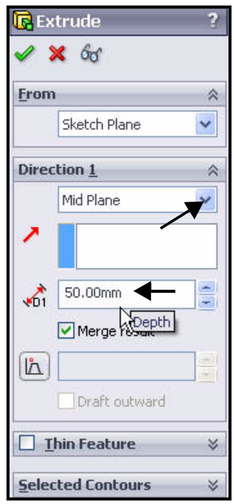
Click the **Features** tab from the CommandManager. The Features toolbar is displayed.

Click the **Extruded Boss/Base** tool. The `Extrude` PropertyManager is displayed.

Click **Isometric** view from the Heads-up View toolbar.

Select **Mid Plane** for the `End Condition` from the drop-down menu.

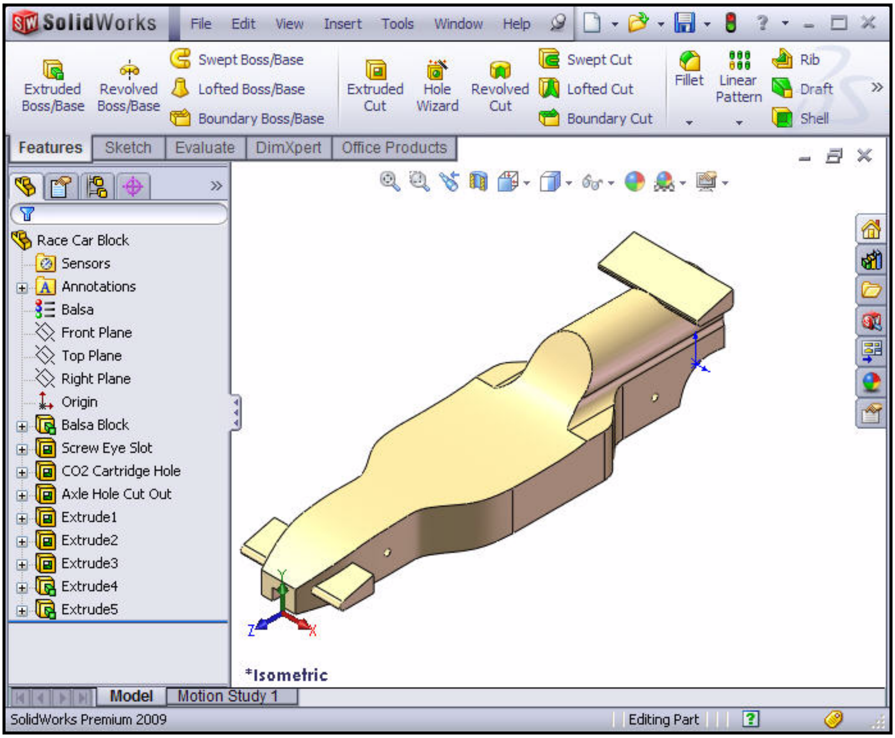
Enter **50**mm for `Depth`.



Click **OK** from the `Extrude` PropertyManager. `Extrude5` is displayed.

Click **Shaded With Edges** from the Heads-up View toolbar.

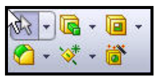
Click **inside** the graphics area. View the results.



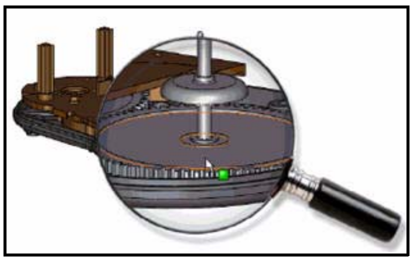
6 Save the model.

Click **Save** from the Menu bar toolbar.

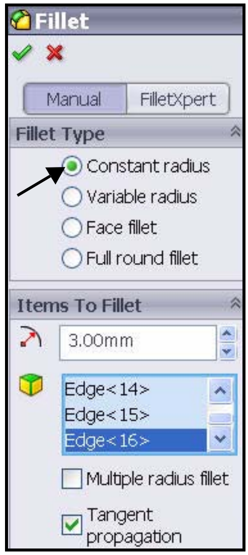
Note: Press the **s** key to view the previous commands in the graphics area.



Note: Press the **g** key to activate the Magnifying glass tool. Use the Magnifying glass tool to inspect a model and make selections without changing the overall view of the model.



Insert Fillets



1 Insert a Fillet Feature.

Fillets creates a rounded internal or external face on the part. You can fillet all edges of a face, selected sets of faces, selected edges, or edge loops.

Click **Hidden Lines Removed** from the Heads-up View toolbar.

Click the **Fillet** tool from the Features toolbar. The `Fillet` PropertyManager is displayed.

Click the **Manual** tab in the `Fillet` PropertyManager. Check the `Constant radius` Fillet Type box.

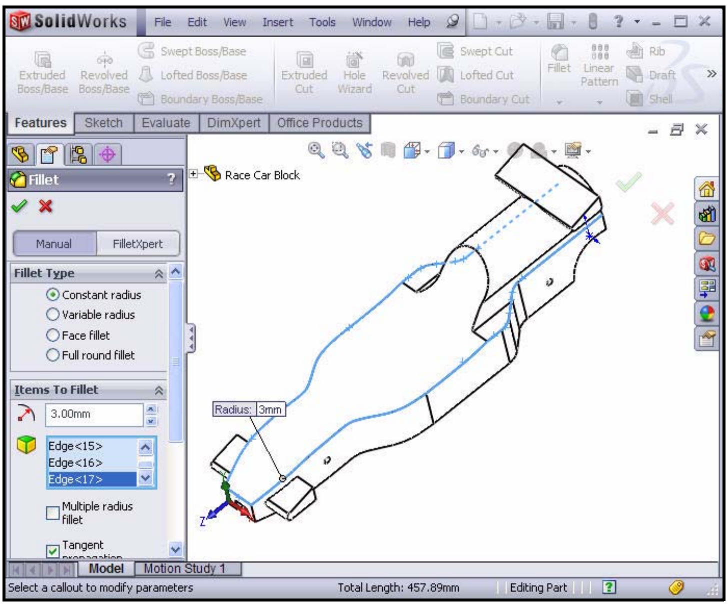
Enter **3**mm for `Radius`.

Click the **8 edges** on the top right of the car. The selected edges are displayed in the `Items To Fillet` box.

Rotate the car with the middle mouse button to view the left side of the car.

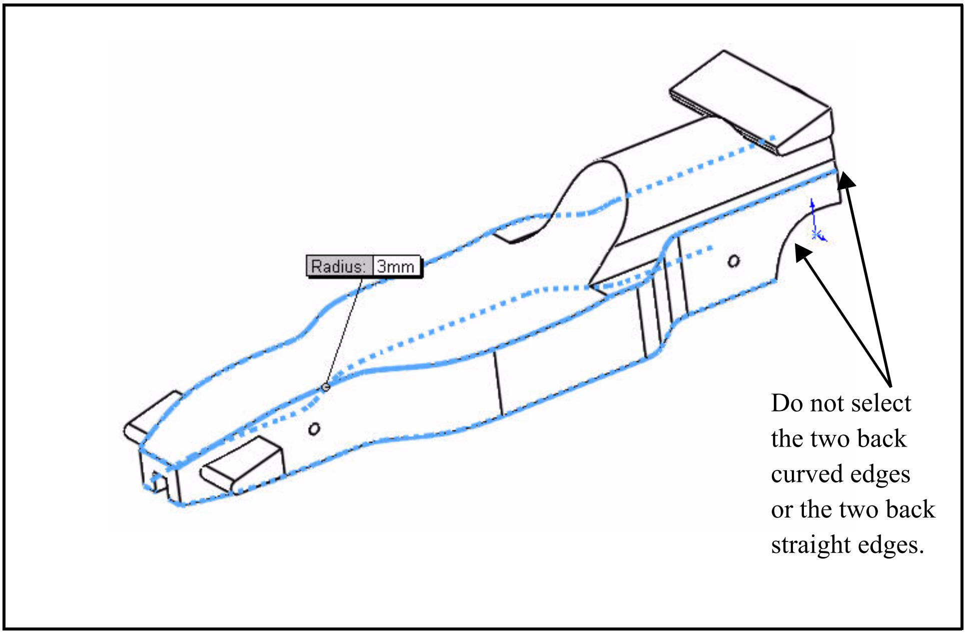
Click the **8 edges** on the top left of the car.

Click the **top front edge** of the car. The selected edges are displayed in the `Items To Fillet` box.



Rotate the car to view the bottom with the middle mouse button.

Click the **bottom edges** of the car. Do not select the two back curved edges or the two back straight edges as illustrated. The selected edges are displayed in the `Items To Fillet` box.



Click **OK** from the `Fillet` PropertyManager. View the `Fillet1` feature in the FeatureManager.

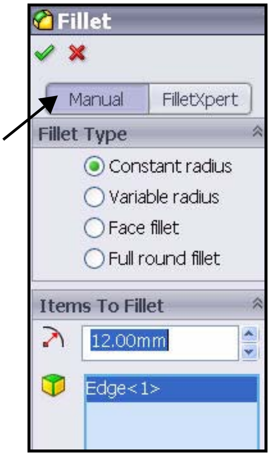
Click **Isometric** view from the Heads-up View toolbar.

2 Insert a second Fillet Feature. Fillet the Cockpit Area.

Click the **Fillet** tool from the Features toolbar. The `Fillet` PropertyManager is displayed.

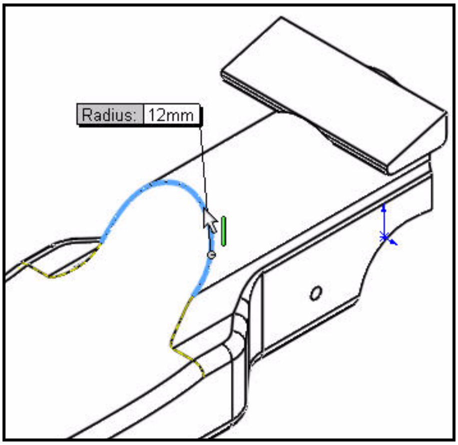
Click the **Manual** tab in the `Fillet` PropertyManager. `Constant radius` Fillet type is selected by default.

Enter **12**mm for `Radius`.



Click the illustrated **back edge**. Edge1 is displayed in the `Items To Fillet` box.

Click **OK** from the `Fillet` PropertyManager. View the `Fillet2` feature in the FeatureManager.



3 Save the model.

Click **Save** from the Menu bar toolbar.

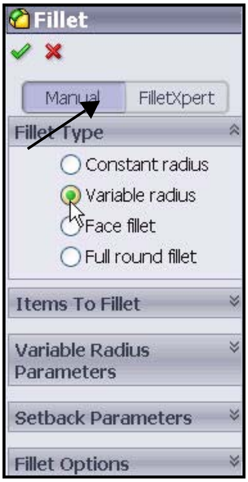
4 Create a Variable Fillet.

Rotate the model using the middle mouse button to view the back curved edges with the middle mouse button.

Click the **Fillet** tool from the Features toolbar. The `Fillet` PropertyManager is displayed.

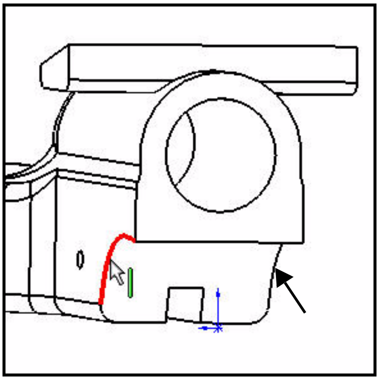
Click the **Manual** tab in the `Fillet` PropertyManager. `Constant radius Fillet Type` is selected by default.

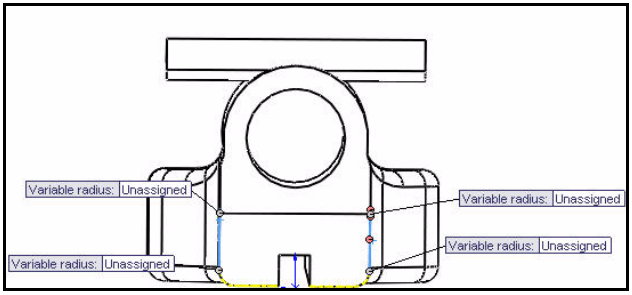
Check the **Variable radius** box for `Fillet Type`.



Click the **two curved edges**.

Click and drag the **Variable radius** boxes off the model.





Click **inside** the top left `Unassigned` box.

Enter **15**mm.

Click **inside** the topright `Unassigned` box.

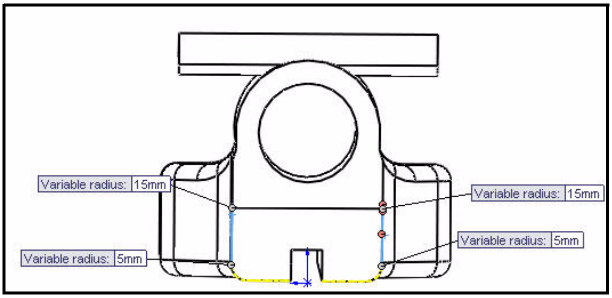
Enter **15**mm.

Click **inside** the bottom left `Unassigned` box.

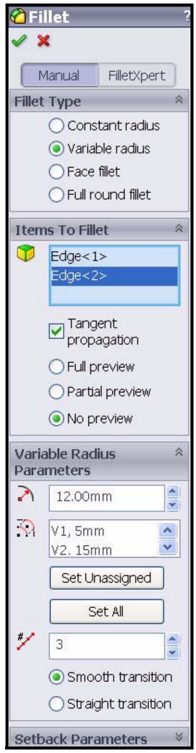
Enter **5**mm.

Click **inside** the bottom right `Unassigned` box.

Enter **5**mm.



Click **OK** from the `Fillet` PropertyManager. View the `VarFillet1` feature in the FeatureManager.



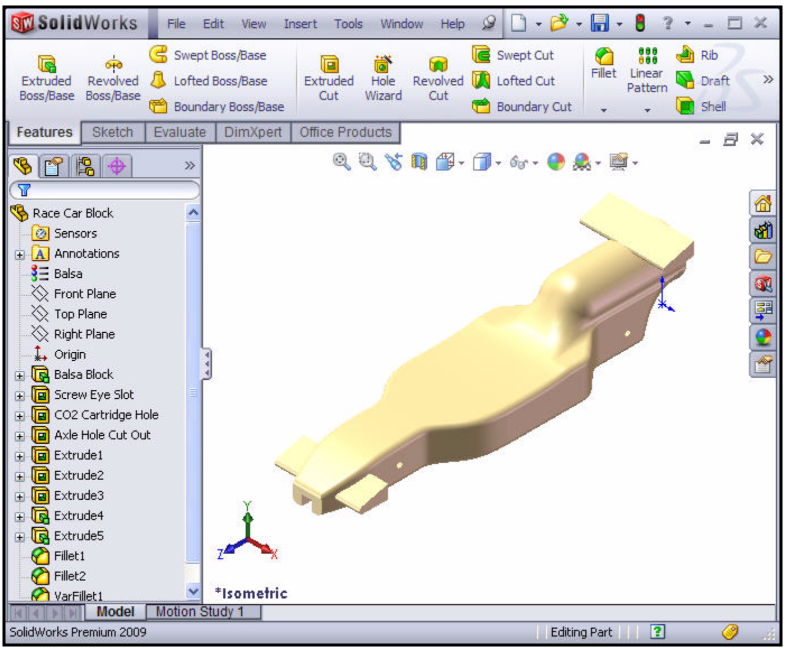
Click **Isometric** view from the Heads-up View toolbar.

Click **Shaded** from the Heads-up View toolbar.

5 Save the model.

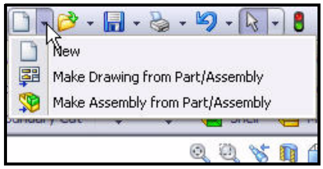
Click **Save** from the Menu bar toolbar.

View the model.



Create an Assembly

Create an assembly with the `Race Car Block`. Insert the Wheels and Axles.



1 Create an assembly.
Click the **Make Assembly from Part/Assembly** tool from the Menu bar toolbar.

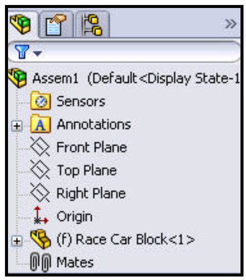
Click **OK** to accept the default Assembly template. The `Begin Assembly` PropertyManager is displayed.

The `Race Car Block` part file is listed in the `Open documents` box.

2 Locate the Component.
Click **OK** from the `Begin Assembly` PropertyManager. The `(f) Race Car Block` is displayed in the assembly FeatureManager design tree as fixed.

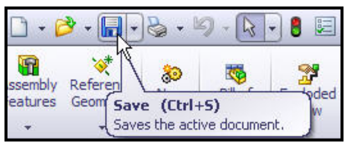
3 Deactivate the Planes.
If needed, click **View**, un-check **Planes** from the Menu bar menu.

Note: The initial component added to the assembly is fixed by default. A fixed component cannot be moved unless you float it.



4 Set Isometric view with Hidden Lines Removed.
Click **Isometric** from the Heads-up View toolbar.

Click **Hidden Lines Removed** from the Heads-up View toolbar.



5 Save the assembly.
Click **Save** from the Menu bar menu.

Save the assembly under the name `Race Car` in the downloaded folder.

Note: If needed, click **View**, un-check **All Annotations.**

6 Insert the Axles.

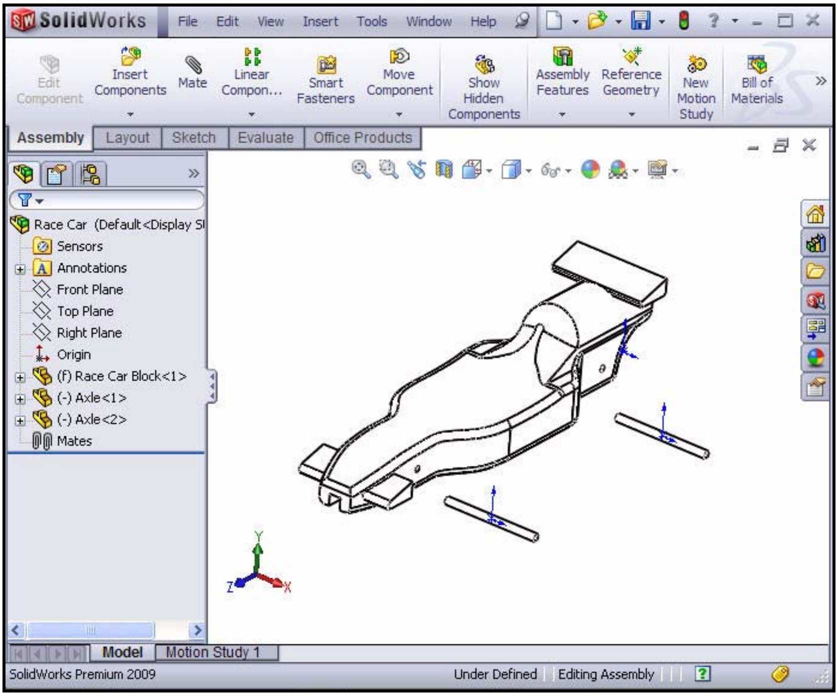
Click and drag the **Axle** part from the Design Library window.

Click a **position** near the rear of the car. The Insert Components PropertyManager is displayed. A second Axle is displayed on the mouse pointer.

Drag the **second Axle** to the front of the car. Click a **position**.

Click **Cancel** from the `Insert Components` PropertyManager. View the FeatureManager.

`Axle <1>` and `Axle <2>` are displayed.



7 Insert the first Wheel.

Click and drag the **Wheel** part from the Design Library window.

Click a **position** near the right rear of the car. The Insert Components PropertyManager is displayed. A second Wheel is displayed on the mouse pointer.

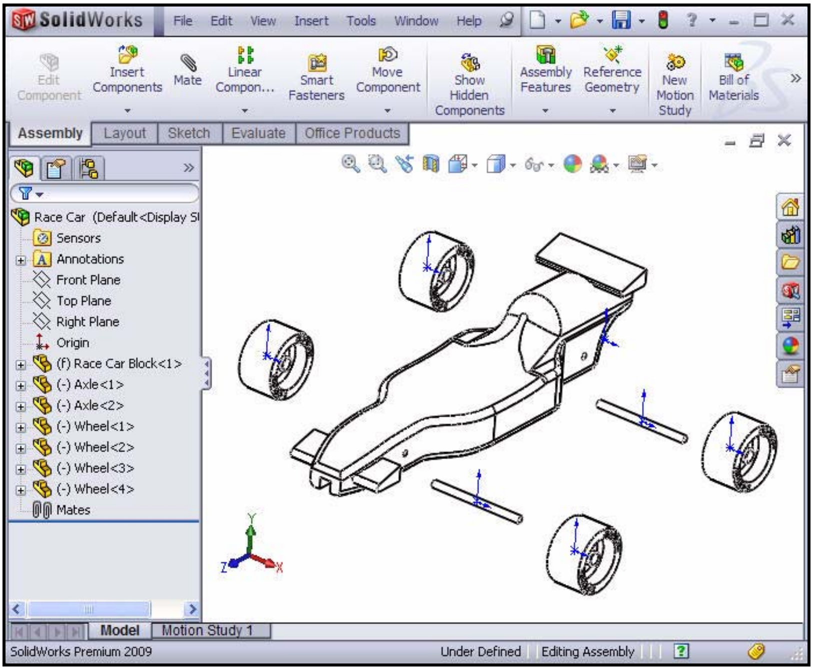
8 Insert the other three Wheels.

Insert the **second Wheel** near the right front of the car; `Wheel<2>`.

Insert the **third Wheel** near the left rear of the car; `Wheel<3>`.

Insert the **fourth Wheel** near the left front of the car; `Wheel<4>`.

Click **Cancel** from the `Insert Components` PropertyManager. View the updated FeatureManager.



9 Deactivate the Origins.

Click **View**, un-check **Origins** from the Menu bar menu.

10 Save the model.

Click **Save** from the Menu bar toolbar.

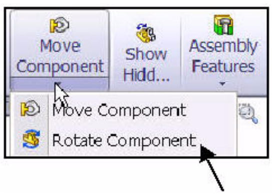
Click **Yes** to rebuild the model.

11 Apply the Rotate Component tool.

Rotate the two Wheels located on the left side of the model.

Click the **Assembly** tab from the CommandManager.

Click `Wheel<3>` from the FeatureManager. This is the rear left wheel.



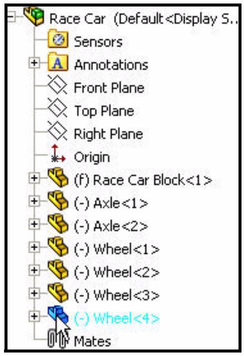
Click the **Rotate Component** tool from the Assembly toolbar. The `Rotate Component` PropertyManager is displayed.

Rotate `Wheel<3>` as illustrated.

Click `Wheel<4>` from the fly-out FeatureManager. This is the front left wheel.

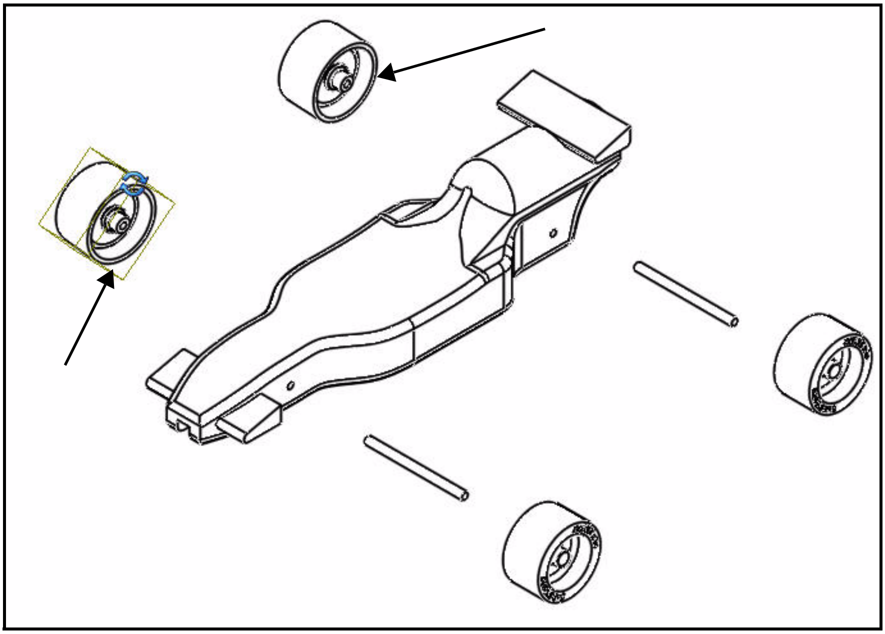
Rotate `Wheel<4>` as illustrated.

Click **OK** from the `Rotate Component` PropertyManager.



12 Rebuild the model.

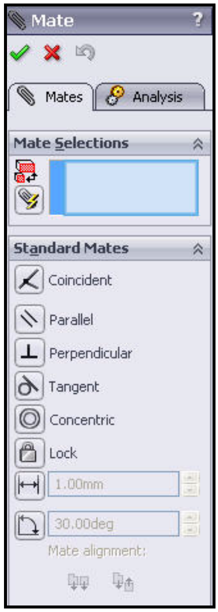
Click **Rebuild** from the Menu bar.



Insert Mates

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; **Standard Mates**, **Advanced Mates** and **Mechanical Mates**.

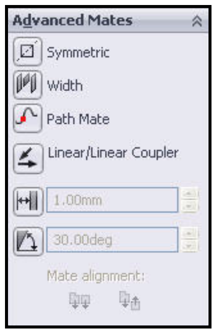


Standard Mates

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

Advanced Mates

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



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

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

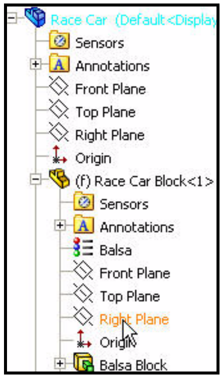
Note: In this section, position the model to view the correct sketch entity. Apply the **Zoom to Area** tool in the Heads-up View toolbar, the middle mouse button, and the **f** and **z** keys.

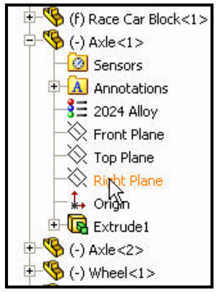
1 Mate the Axles to the body.

Create a Coincident mate between the rear axle and the body.

Click the **Mate** tool from the Assembly toolbar. The `Mate` PropertyManager is displayed.

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

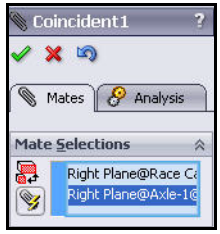




Expand the fly-out `Race Car` FeatureManager in the graphics area.

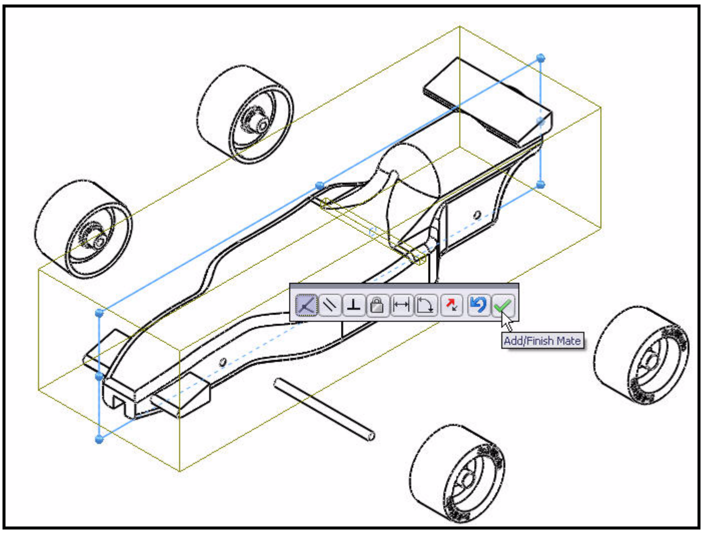
Click the `Race Car Block/ Right Plane` in the fly-out FeatureManager.

Click the `Race Car Axle<1>/ Right Plane` in the fly-out FeatureManager. Coincident mate is selected by default.



The selected planes are displayed in the `Mate Selections` box.

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



2 Insert a Concentric mate.

Create a Concentric mate between the rear axle and the body.

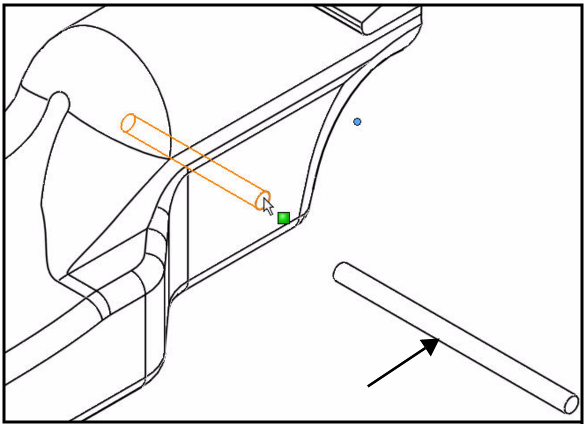
Drag `Axle<1>` as illustrated.

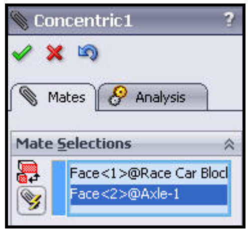
Click the **inside cylindrical face** of the rear hole.

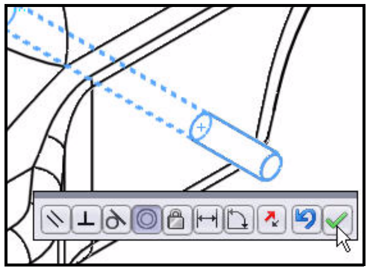
Click the **cylindrical outside face** of `Axle<1>`.

Concentric mate is selected by default.

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







Note: In this section, position the model to view the correct sketch entity. Apply the **Zoom to Area** tool, the middle mouse button, and the **f** and **z** keys.

3 Insert a Coincident Mate.

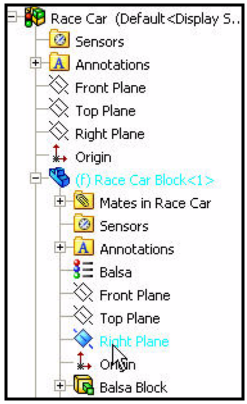
Create a Coincident mate between the front axle and the body.

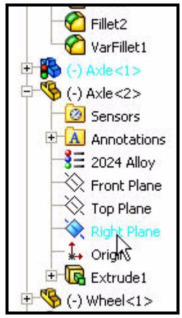
Click the `Race Car Block/Right Plane` in the fly-out FeatureManager.

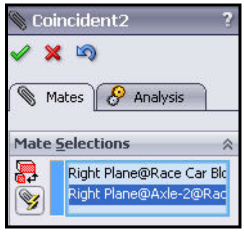
Click the `Race Car Axle<2>/Right Plane` in the fly-out FeatureManager.

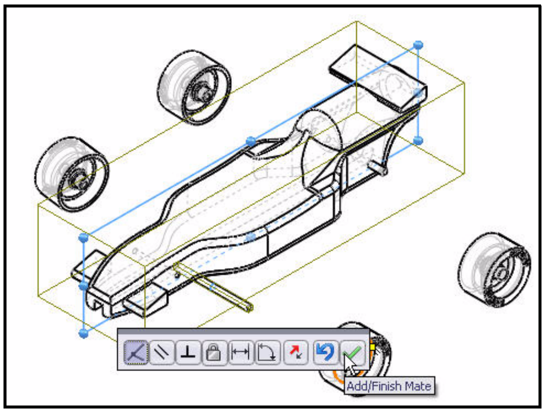
Coincident mate is selected by default.

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









4 Insert a Concentric Mate.

Create a Concentric mate between the front axle and the body.

Drag `Axle<2>` as illustrated.

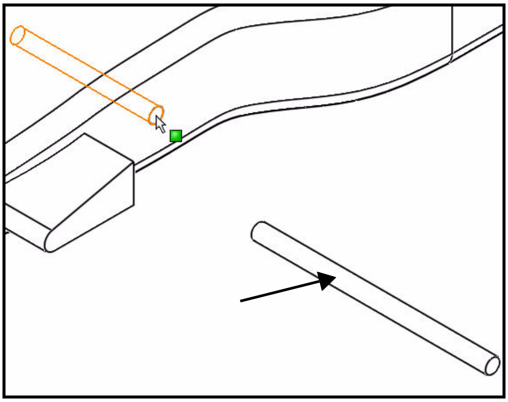
Click the **inside cylindrical face** of the front hole.

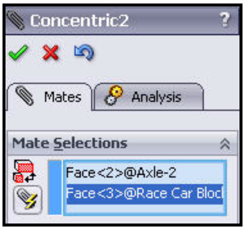
Click the **cylindrical outside face** of `Axle<2>`.

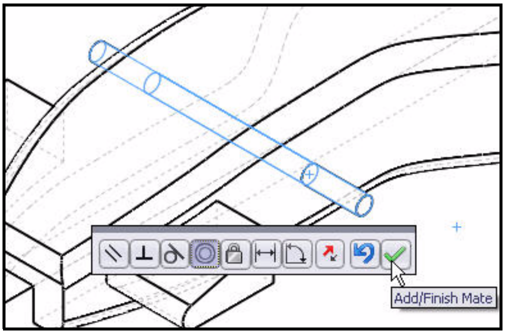
Concentric mate is selected by default.

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

In the next section mate the Wheels with the Axles.



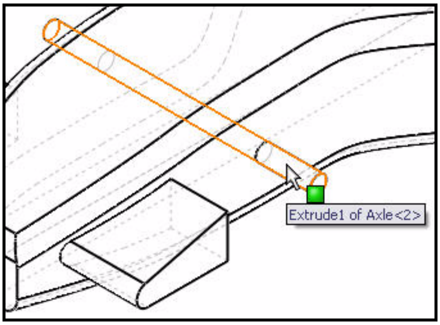




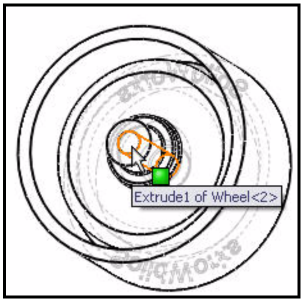
1 Mate the Wheels to the Axles.

Create a Concentric mate between the front axle and the front right wheel.

Click the **cylindrical outside face** of `Axle<2>`.



Click the **cylindrical inside face** of the front right `Wheel<2>`.



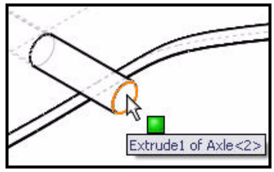
Concentric is selected by default.

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

Note: Position the model to view the correct sketch entity.

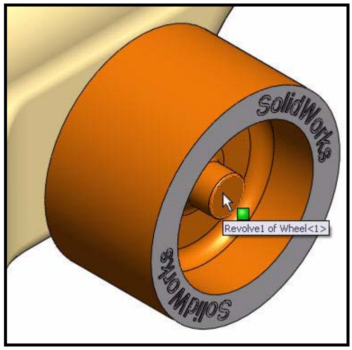
2 Create A Distance Mate.

Create a Distance mate between the outside end face of the front right `Axle<2>` and the outside face of the front right `Wheel<2>`.



Click the **outside end face** of the front right `Axle<2>`.

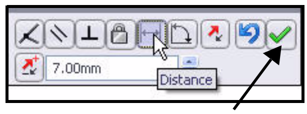
Click **Shaded** from the Heads-up View toolbar.



Click the **outside face** of the right front `Wheel<2>` as illustrated.

Click the **Distance Mate** tool.

Enter **7**mm.



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

3 Mate the three remaining Wheels to the Front and Rear Axles.

Repeat the **above procedures** to create Concentric mates between the Axles and the Wheels.

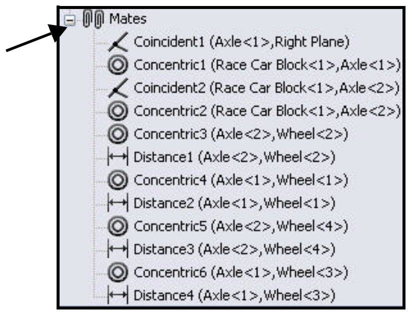
Create Distance mates between the outside end face of the Axles and the outside face of the Wheels.

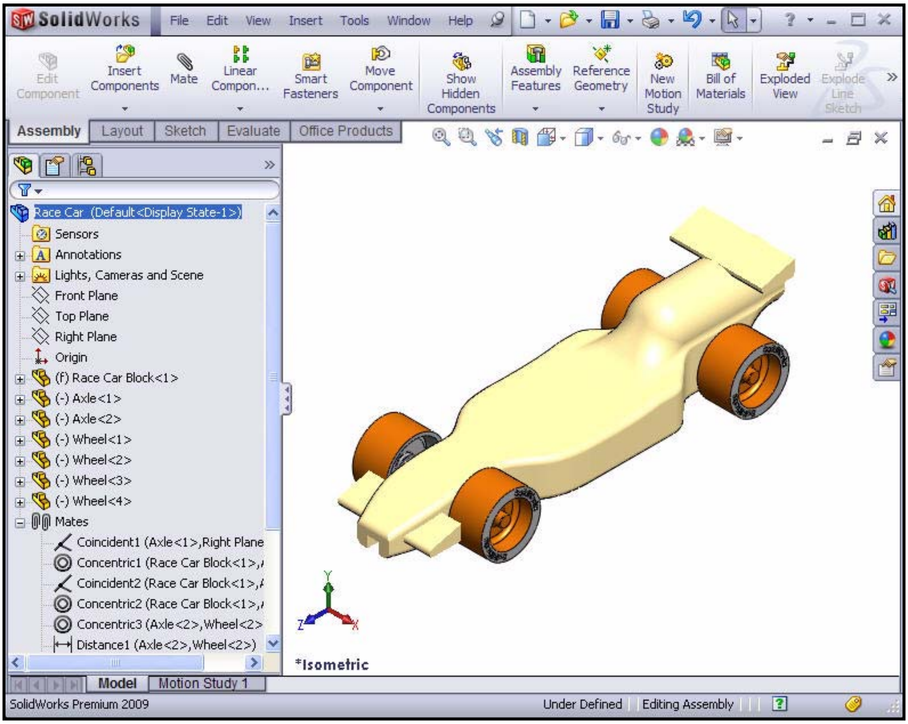
Click **OK** from the `Mate` PropertyManager.

4 View the Created Mates.

Expand the `Mates` folder from the FeatureManager.

View the created mates.

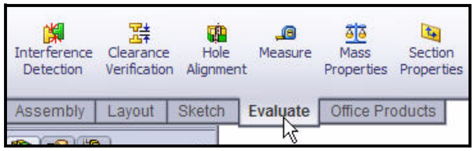


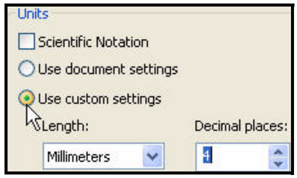


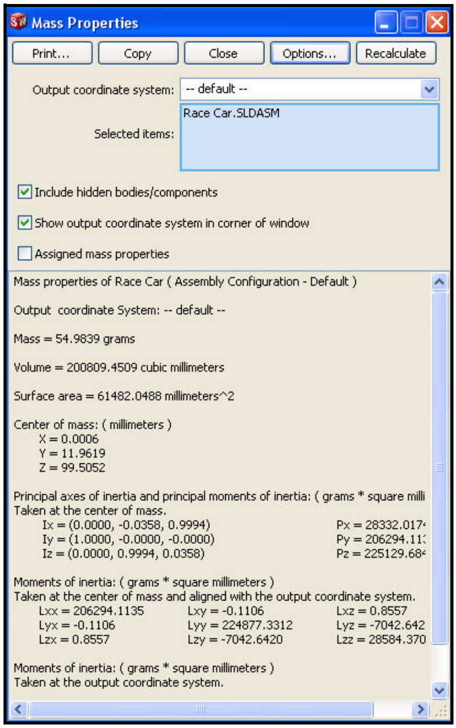
5 Save the model.

Click **Save** from the Menu bar toolbar.

Calculate the Weight of the Race Car







When you are finished and ready to race, your car has to weigh no less than 55 grams. This does not include the CO_2 cartridge. Check the weight of the model. Apply the `Mass Properties` tool.

1 Click the **Evaluate** tab from the CommandManager.

Click **Mass Properties** from the Evaluate toolbar. The `Mass Properties` dialog box is displayed.

Click the **Options** button.

Check the **Use custom setting** box.

Select **4** for `Decimal place`.

Click **OK**.

The Mass = 54.9839 grams.

Note: The mass can be different if you did not fillet all of the edges or too many.

There will be eye hooks, paint, decals, and sanding. Use this mass as an estimate and make sure to weigh the completed car before racing. A list of critical dimensional rule requirements are provided at the end of this lesson.

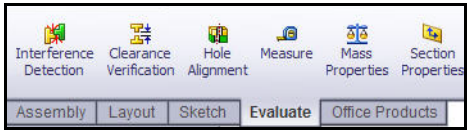
Note: The mass of the Axle part using 2024 Alloy is .9896 grams. If the Axle part was changed to AISI 304, the total mass increase of the `Race Car` would be approximately 3.67 grams. This is explored in the Analysis section of the book.

Close the Mass Properties dialog box.

2 Save the model.

Click **Save** from the Menu bar toolbar.

Calculate the Overall Length of the Race Car



When you are finished and ready to race, your car can't exceed 210mm in length and the Wheels have a minimum of 26mm and a maximum of 34mm. Apply the `Measure` tool to obtain these measurements of the `Race Car` assembly.

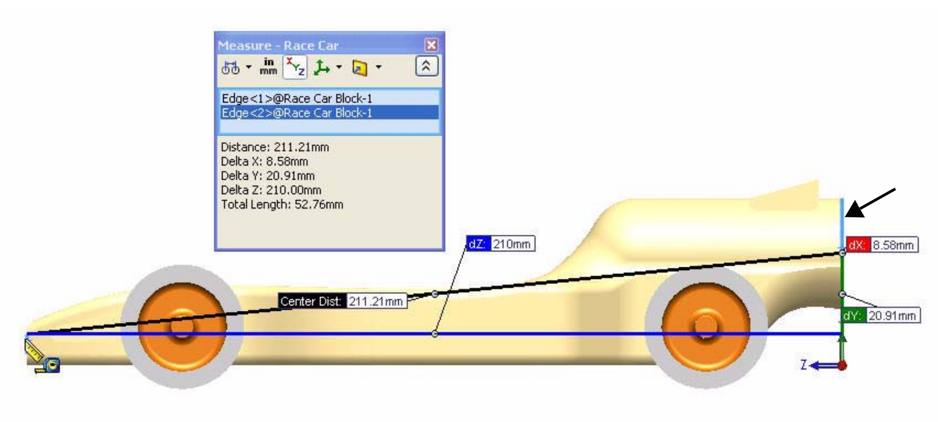
1 **Measure the Overall Length of the Car.**

Click **Right view**  from the Heads-up View toolbar.

Click the **Measure**  tool from the Evaluate toolbar. The `Measure - Race Car` dialog box is displayed.

Click the **front edge** of the `Race Car`. **Zoom in** if needed to select the edge.

Click the **back edge** of the `Race Car`. Note: Select an edge, not a point or face. View the results.

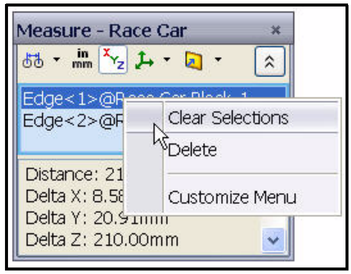


Note: The `Balsa Block` is 223mm x 50mm x 65mm. If you plan to use a fixture to machine your car, you must make sure that your design is no longer that 210mm. Most fixtures have a nose plate that holds the front of the `Balsa Block` and if your design is too long, it can/will end up breaking the endmill or possible damaging the fixture.

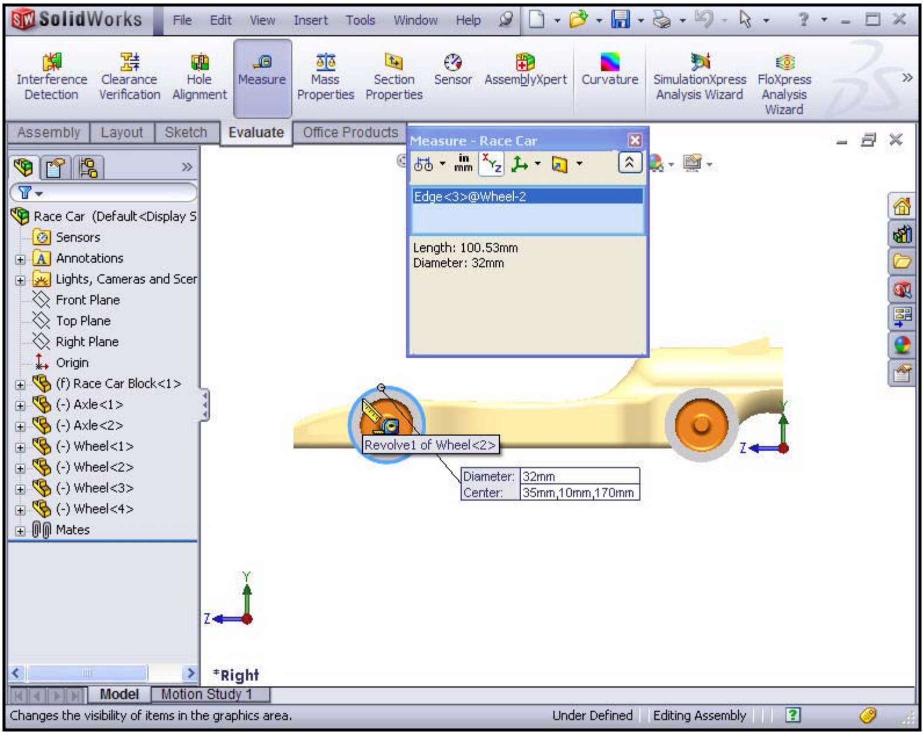
2 Measure the Wheel<2> diameter.
Right-click inside of the `Selection` box.

Click **Clear Selections**.

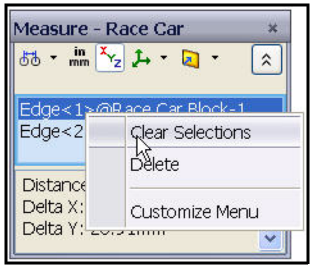
Click the **diameter** of the front `Wheel<2>`. The diameter of `Wheel<2>` is 32mm.



Note: Remember your wheels need to be between 26mm and 34mm in diameter.



3 Measure the center distance between the two Wheel hubs.
Right-click inside of the `Selection` box.

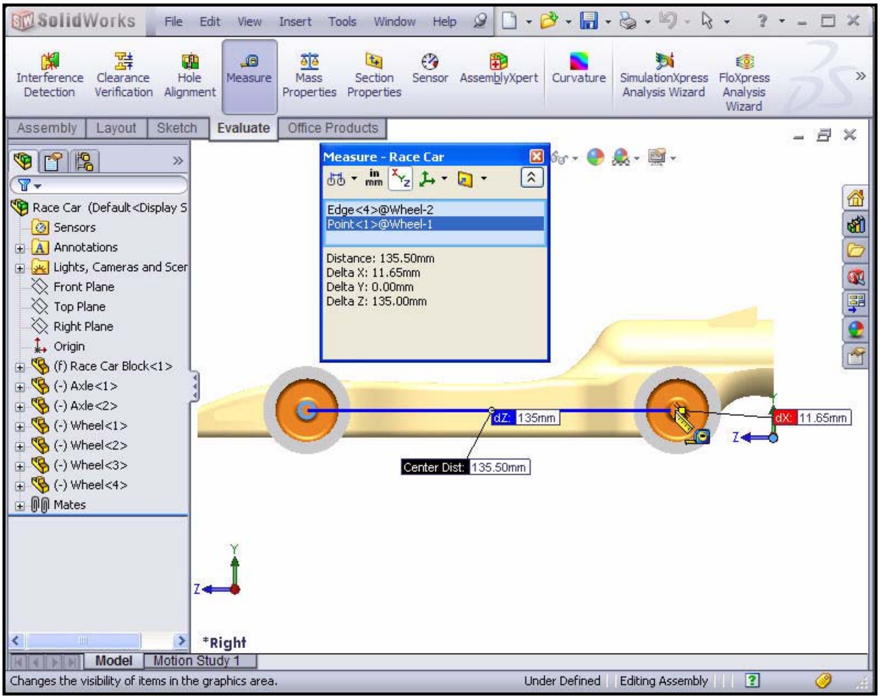


Click **Clear Selections**.

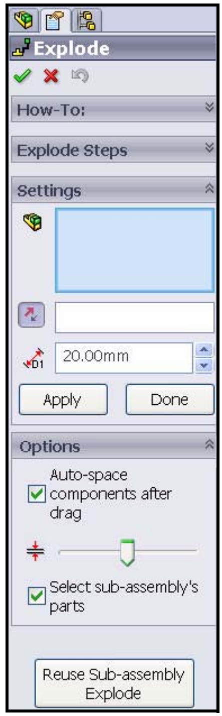
Click the **front hub face** of front `Wheel<1>`.

Click the **front hub face** of back `Wheel<2>`. The center distance between the two Wheel hubs is 135mm.

Close the `Measure - Race Car` dialog box.



Create an Exploded view

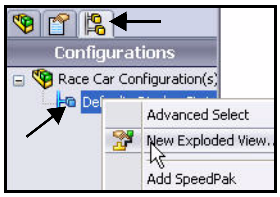


For manufacturing purposes, it is often useful to separate the components of an assembly to visually analyze their relationships. Exploding the view of an assembly allows you to look at it with the components separated.

An exploded view consists of one or more explode steps. An exploded view is stored with the assembly configuration with which it is created. Each configuration can have one exploded view.

The `Explode` PropertyManager is displayed when you create or edit an exploded view of an assembly.

Note: While an assembly is exploded, you cannot add mates to the assembly.



1 **Create an Exploded view Configuration.**

Click **Isometric** from the Heads-up View toolbar.

Click the **ConfigurationManager** tab.

Right-click **Default** from the ConfigurationManager.

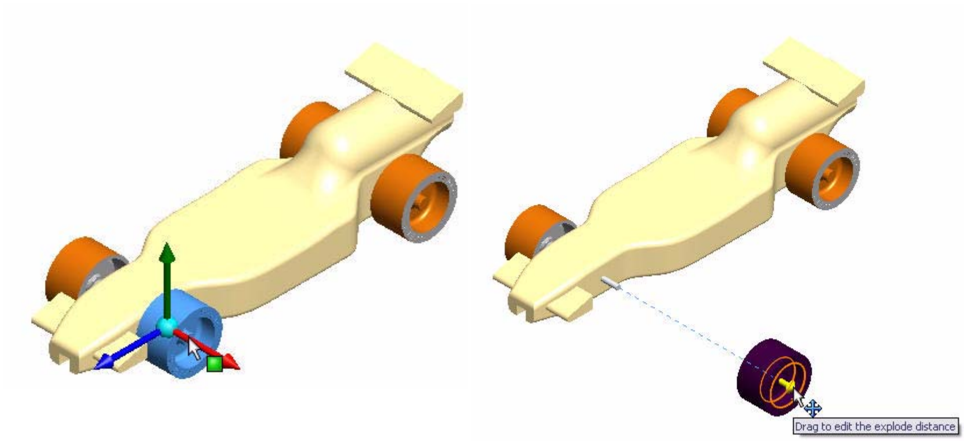
Click the **New Exploded View** tool. The `Explode` PropertyManager is displayed.

Click the **right front** `Wheel<2>` of the model in the graphics area. A Triad is displayed.

Click and drag the **red/yellow Triad** arrow to the right.

Note: Drag the wheel far enough to the right to leave room for `Axle<2>`.

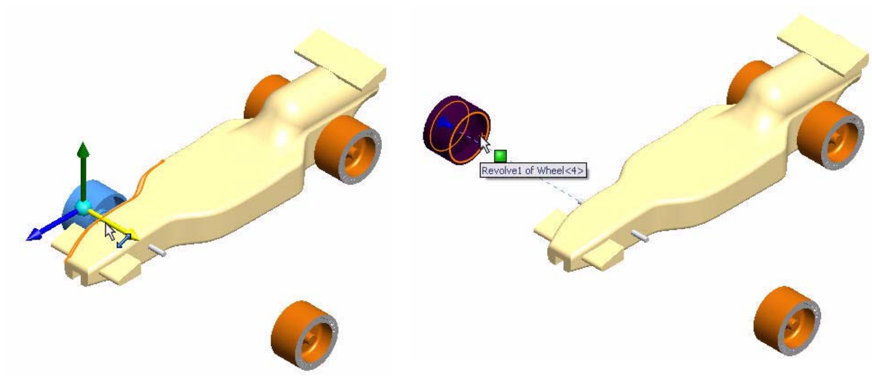
Click the **Done** button from the `Settings` box.



2 Create Explode Step2.
Click the **left front** `Wheel<4>` of the model. A Triad is displayed.

Click and drag the **red/yellow Triad** arrow to the left.

Click the **Done** button from the `Settings` box.



3 Create Explode Step3.
Click the **right back** `Wheel<1>` of the model. A Triad is displayed.

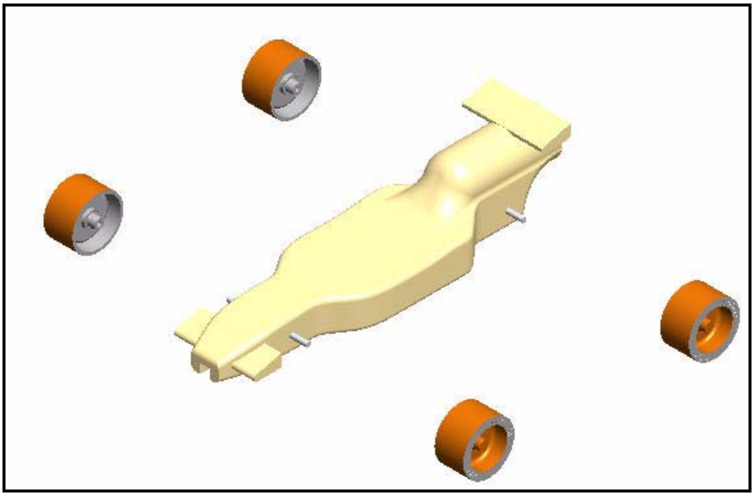
Click and drag the **red/yellow Triad** arrow to the right. Drag the wheel far enough to the right to leave room for `Axle<1>`.

Click the **Done** button from the `Settings` box.

4 Create Explode Step4.
Click the **left back** `Wheel<3>` of the model. A Triad is displayed.

Click and drag the **red/yellow Triad** arrow to the left.

Click the **Done** button from the `Settings` box. View the results.



5 Create Explode Step5.
Click the **front** `Axle<2>` of the model. A Triad is displayed.

Click and drag the **red/yellow Triad** arrow to the right.

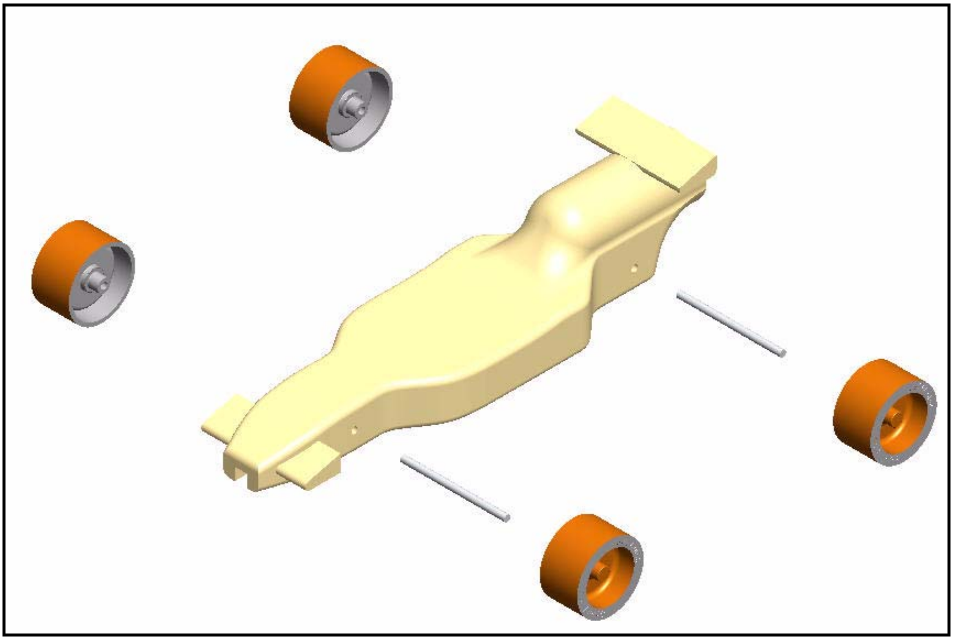
Click the **Done** button from the `Settings` box.

6 Create Explode Step6.
Click the **right back** `Axle<1>` of the model. A Triad is displayed.

Click and drag the **red/yellow Triad** arrow to the right.

Click the **Done** button from the `Settings` box. View the model.

Expand each **Explode Step** in the `Explode Steps` box. View the results.



7 Return to the ConfigurationManager.

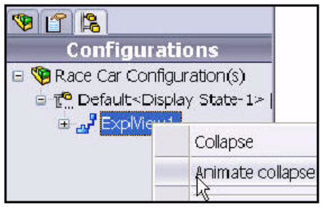
Click **OK** from the `Explode` PropertyManager.

8 Animate the assembly.

Expand the **Default** configuration. `ExpView1` is displayed.

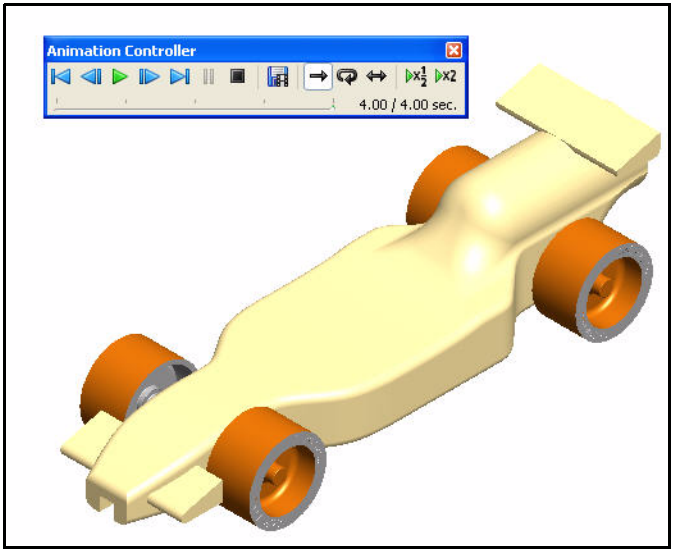
Right-click **ExplView1.**

Click **Animate collapse**. View the results.



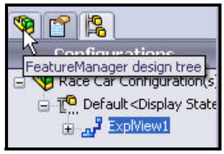
Click the **Play** button from the `Animation Controller` dialog box. View the Animation of the `Race Car`.

Close the `Animation Controller` dialog box.



9 Return to the FeatureManager.

Click the **FeatureManager** tab.



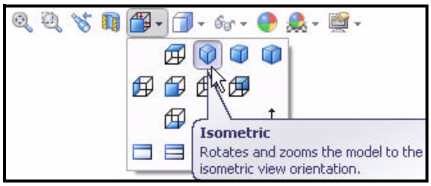
10 Save the model.

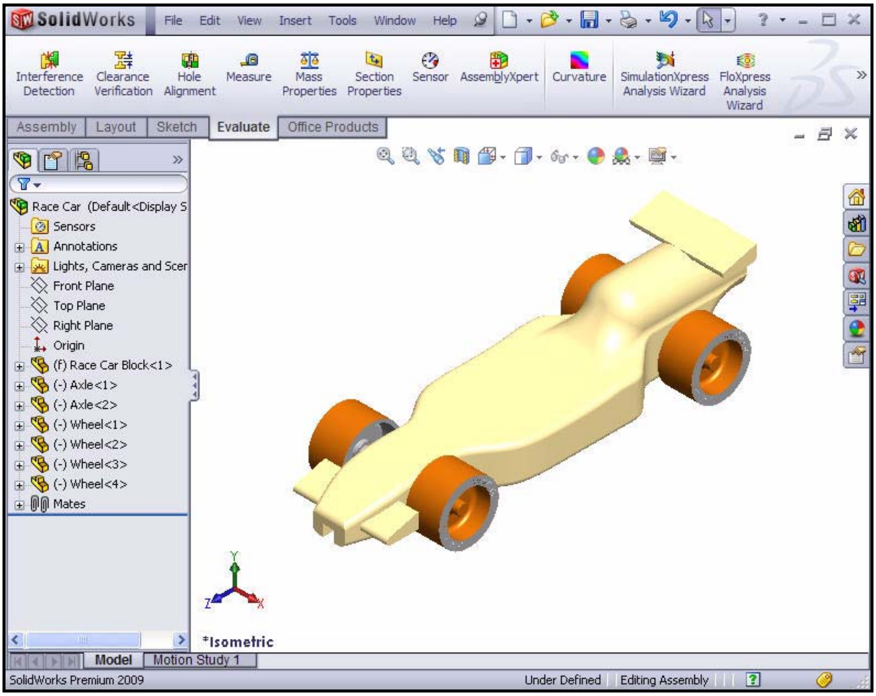
Click **Isometric** from the Heads-up View toolbar.

Click **Save** from Menu bar.

You are finished with the assembly.

In the next section, open individual parts from the assembly and apply the Measure tool.

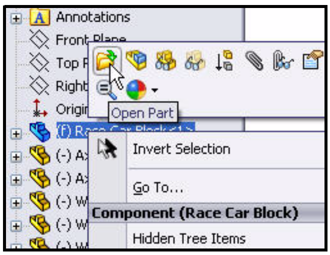




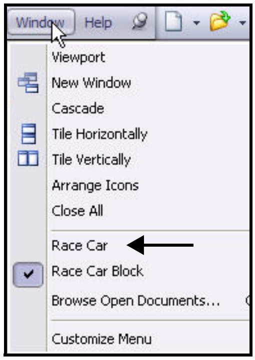
1 Open the Race Car Block Part from the Assembly.

Right-click `(f) Race Car Block<1>` from the FeatureManager.

Click **Open Part** from the Context toolbar. The `Race Car Block` FeatureManager is displayed.

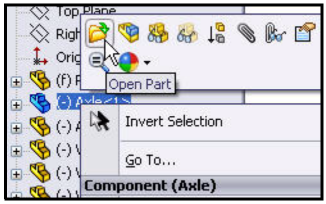


2 Return to the Race Car Assembly.
Click `Window, Race Car` from the Menu bar menu. The `Race Car` assembly is displayed.



3 Open the Axle Part from the Assembly.
Right-click `Axle<1>` from the FeatureManager.

Click **Open Part** from the Context toolbar. The `Axle` FeatureManager is displayed.



4 Apply the Measure tool to the Axle.
Measure the overall length.

Click **Front** view from Heads-up View toolbar.

Press the **f** key to fit the model to the graphics area.

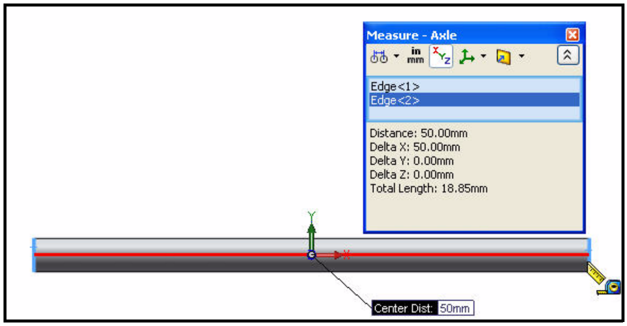
Click the **Measure** tool from the Evaluate toolbar. The `Measure - Axle` dialog box is displayed.

Click the **left edge** of `Axle<1>`.

Zoom in if needed to select the edge.

Click the **right edge** of `Axle<1>`.

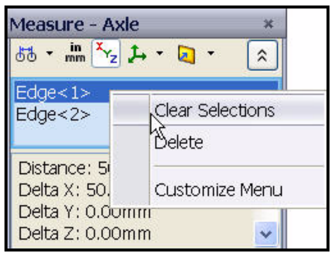
View the results.



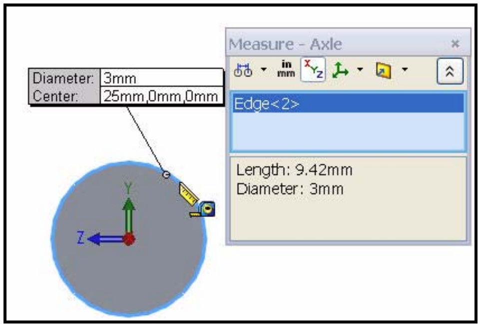
5 Measure the diameter of the Axle.
Right-click inside of the `Selection` box as illustrated.

Click **Clear Selections**.

Click **Right** view from the Heads-up View toolbar.



Click the **circumference** of the `Axle<1>`. The diameter is 3mm.

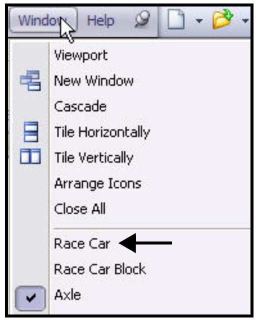


Close the `Measure - Axle` dialog box.

Click **Isometric** view from the Heads-up View toolbar.

6 Return to the Race Car assembly.

Click `Window, Race Car` from the Menu bar menu.



The `Race Car` assembly is displayed.

1 Explore various Scenes and View Settings.

Click the drop-down arrow from the **Apply scene** tool in the Heads-up View toolbar.

View your options.

Click **Backdrop - Studio Room**.

View the results in the graphics area.

Click **Plain White**.

View the results in the graphics area.

Click **Warm Kitchen**.

Click the drop-down arrow from the **View settings** tool in the Heads-up View toolbar.

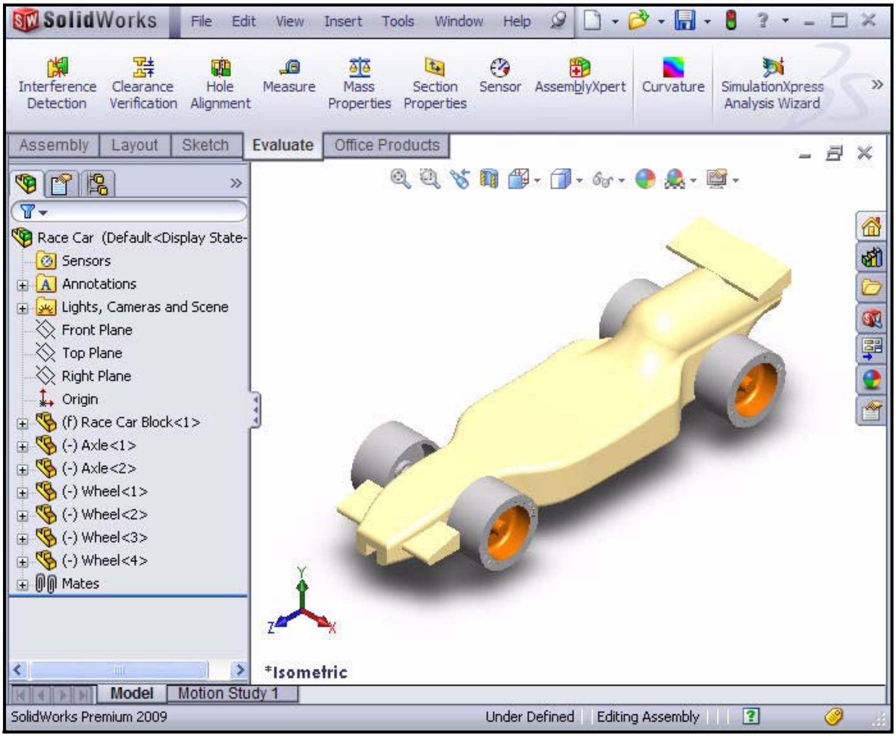
Click the **Shadows In Shaded Mode** icon.

Rotate the model with your middle mouse button. View the results.

2 Save the model.

Click **Isometric** from the Heads-up View toolbar.

Click **Shaded** from the Heads-up View toolbar.

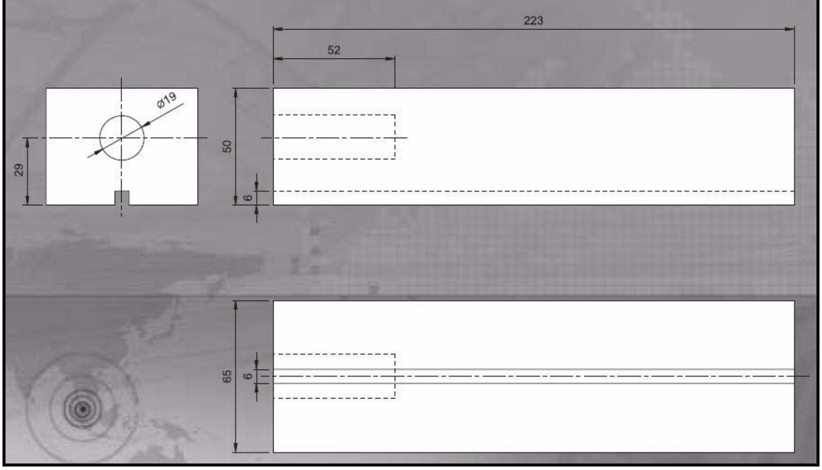


Click **Save** from the Menu bar. You are finished with the assembly. Review below for some of the dimensional rule requirements for the `CO2 Cartridge Race Car` assembly. In the next lesson, you will create an `Race Car` assembly drawing with dimensions.

Race Car Dimensional Requirements

Below are some of the dimensional requirements (Type-R) for the `Race Car Block` and the CO2 cartridge hole. Review the dimensional requirements. Apply the Measure tool to confirm that you meet the design requirements!

Below: Third angle orthographic projection of the (R-TYPE) blank balsa wood block, used for construction of the F1 racing cars. All dimensions listed are in mm (millimetres). Note - the drawing is **NOT TO SCALE**



Body dimensions copied from the 2008 - 2009 Rules and Regulations folder from the F1inschools.co.uk site.

Body Dimensions

No.	*Structure*	*Min.*	*Max.*
3a.	Full body length *	170	210
3b.	Body height above the track* (excluding eyelets) including side pods and wings	3	10
3c.	Body width at side pods*	50	65
3d.	Total body width, including wheels *	60	85

(all dimensions stated in millimetres, mm.)

No.	*Structure*	*Min. Weight*
3e.	Body weight without the CO_2 cartridge	55.0

(all weight values stated in grams, g.)

3f. No part of the body should be less than 3mm thick - this excludes air foils / wings

3g. Maximum body height (including aerofoils) 60

* Additional Notes

3a. measured between front and rear extemeties of body.
3b. measured from track surface to the car body.
3c. measured from side-to-side of the car body - the side pods are the part of the car that flanks the sides of the cockpit area of the car. The outside face of the side pods when viewed from the side the pods must present a surface measuring not less than 30X15 mm - a sticker of 30X15mm will be applied to both side pods and must be 100% visible when viewed from the side. Side pods can be convex, concave or flat but capable of taking the F1 in Schools promotional logo decal.
3d. measured between outside edges of the wheels or body, whichever is widest.

Wheel dimensions copied from the 2008 - 2009 Rules and Regulations folder from the F1inschools.co.uk site.

Wheel Dimensions

4a. All F1 cars must have 4 wheels, two at the front, two at the rear and all wheels must be cylindrical.

4b. All wheels must fit the following criteria:

No.	*Structure*	*Min.*	*Max.*
4c.	Front wheel diameter *	26	34
4d.	Front wheel width * (at surface contact point)	15	19
4e.	Rear wheel diameter *	26	34
4f.	Rear wheel width * (at surface contact point)	15	19

(all dimensions stated in millimetres, mm.)

4g. All 4 wheels must touch the racing surface at the same time and all wheels should roll easily.

4h. Wheel dimensions must be consistent with the whole diameter/circumference of the wheel.

4i. A school/college/organised youth group may manufacture their own wheels, as long as they fit within the set specification.

* Additional Notes

4c. & 4e. measured to the extreme outer edges of each wheel.
4d. & 4f. measured between the extreme edges (including any protrusions).

Wheel to Body dimensional design requirements copied from the 2008 - 2009 Rules and Regulations folder from the F1inschools.co.uk site.

Wheel to Body Dimensions

The wheels are not allowed to be inside the car body and 100% of the wheel should be visible from the plan, side and views.

No.	*Structure*	
5a.	Front wheel visible (from the plan/side view)	Yes / No
5b.	Rear wheel visible (from the plan/side view)	Yes / No

Power Plant dimensional design requirements copied from the 2008 - 2009 Rules and Regulations folder from the F1inschools.co.uk site.

Power Plant

The event organisers will provide all CO_2 cartridges for the regional finals, national finals and World Championship.

No.	*Structure*	*Min.*	*Max.*
6a.	CO_2 cartridge chamber diameter	19.1	19.9
6b.	Lowest point of chamber to the track surface *	22.5	30
6c.	Depth of hole	50	60
6d.	Wall thickness around cartridge *	3.1	_

6e. No paint is allowed inside the chamber (please seal off or protect the chamber while painting).

* Additional Notes

6b. measured from track surface to lowest surface part of the CO_2 chamber.
6d. clear space surrounding the CO_2 cartridge below 3 mm the car will not be allowed to race and loose marks accordingly.

Car Body and Wings dimensional design requirements copied from the 2008 - 2009 Rules and Regulations folder from the F1inschools.co.uk site

Car Body and Wings

8a. The car body including side pods AND rear wing, must be machined from a single piece of balsa wood. Aerofoils at the front may be machined as part of the car body or from a seperate material - non-metallic.

8b. The design of the completed R-TYPE car should resemble an actual F1 car and shall include the following features:
An aerofoil on the front nose of the car, an aerofoil on the rear of the car and side pods on both sides of the car

No.	*Structure*	*Min.*	*Max.*
8c.	Rear/Front Wing width (where the wing is split by the body of the car, the width is calculated as a sum of both parts.)	40	65
8d.	Rear/Front wing depth	15	25
8e.	Front wing thickness	1	12
8f.	Rear wing thickness	3	12

* Additional Notes

The whole of the front aerofoil when viewed from the side must be in front of the centre line of the front axle.
The whole of the rear aerofoil when viewed from the side must be behind the centre line of the rear axle.
A driver cockpit/driver is an opptional feature.
Designs will be tested and examined for any implants or voids hidden within the car body.
8e/8f. The minimum depth of both front and rear wings is to be measured at the narrowest point on each wing.

Lesson 3
Create an Assembly Drawing

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

- Create a B-size Sheet drawing of the `Race Car` assembly
- Apply the View Palette in the Task Pane
- Insert an Isometric view with a Bill of Materials
- Modify the View scale
- Modify the Sheet scale
- Add a drawing Sheet
- Edit the drawing Title block
- Insert a Front, Top and Right view
- Insert Dimensions into drawing views
- Create an Exploded Isometric view

Drawings

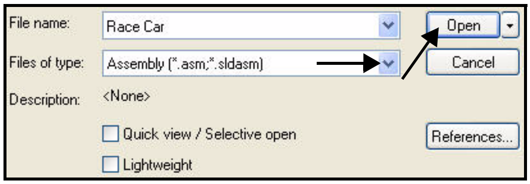
SolidWorks enables you to easily create drawings of parts and assemblies. These drawings are fully associative with the parts and assemblies they reference. If you change a dimension on the finished drawing, that change propagates back to the model. Likewise, if you change the model, the drawing updates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

Create an Assembly Drawing

1 **Open the Race Car Assembly.**
Click **File, Open**, or click **Open** from the Menu bar toolbar.

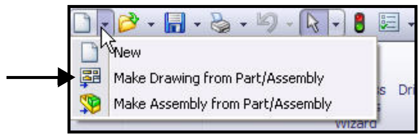


Browse to the `Race Car` assembly folder.

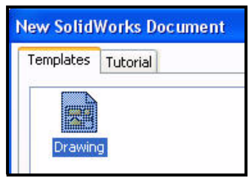
Open the `Race Car` assembly.

The `Race Car` assembly FeatureManager is displayed.

2 **Create an Assembly Drawing.**
Click the **Make Drawing from Part/Assembly** tool from the Menu bar toolbar.

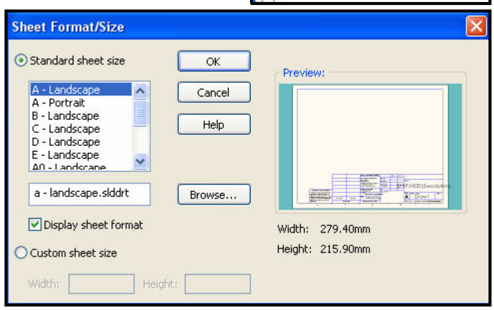


Accept the standard Drawing Template.



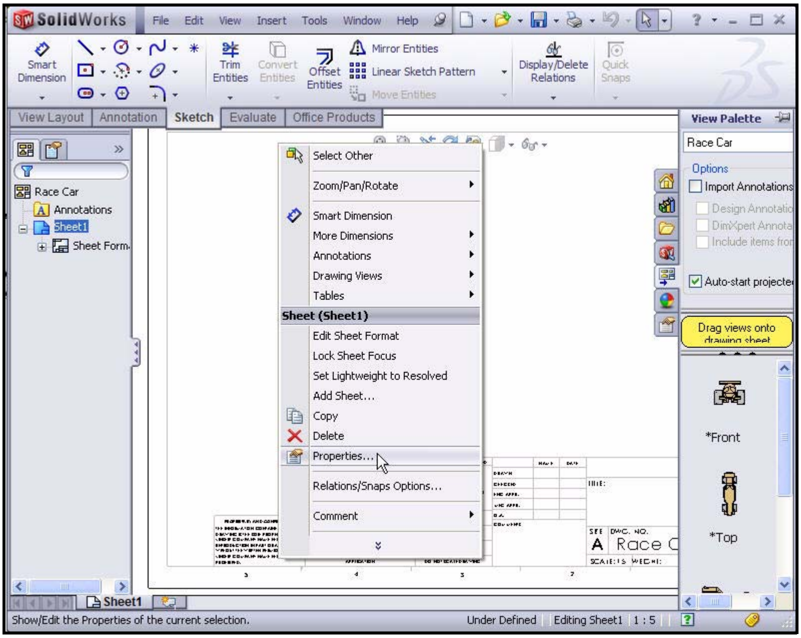
Click **OK** from the `New SolidWorks Documents` dialog box.

Click **OK** from the `Sheet Format/Size` dialog box.



Right-click inside the drawing sheet.

Click **Properties**. The `Sheet Properties` dialog box is displayed.



3 Select the sheet size.

Click **B - Landscape** for `Sheet Format/Size`.

The default name of the Sheet is `Sheet1`.

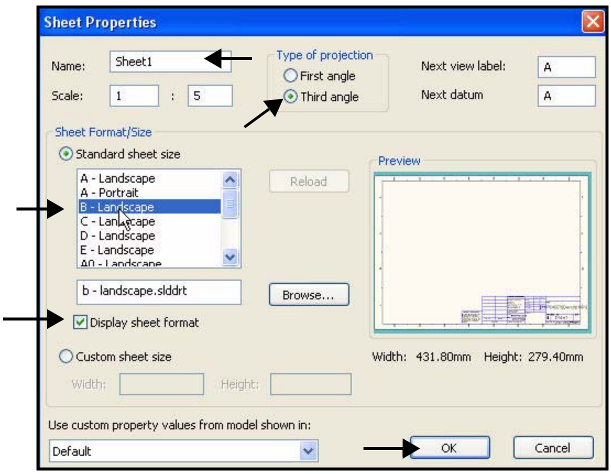
The `Type of projection` is `Third angle`.

The Sheet `Scale` size is 1:5.

Check the **Display sheet format** box.

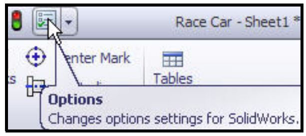
Click **OK** from the `Sheet Properties` dialog box.

The drawing sheet is displayed.

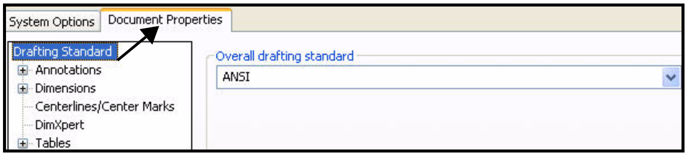


4 Set Document Properties.

Click **Tools, Options**, or click **Options** from the Menu bar toolbar.



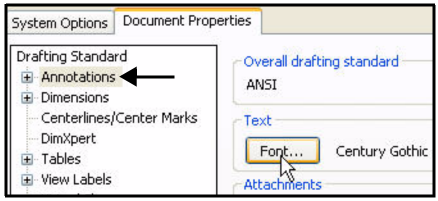
Click the **Document Properties** tab.



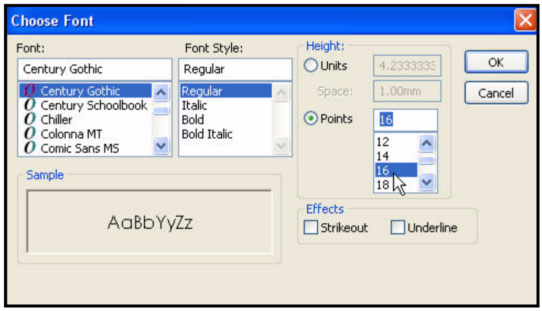
Select **ANSI** for `Overall drafting standard`.

5 Set Annotations Font.

Click the **Annotations** folder.



Click the **Font** button. The `Choose Font` dialog box is displayed. Select the drawing font.



Select **Century Gothic** from the `Font` box.

Select **Regular** from the `Font Style` box.

Check the **Points** box from the `Height` area.

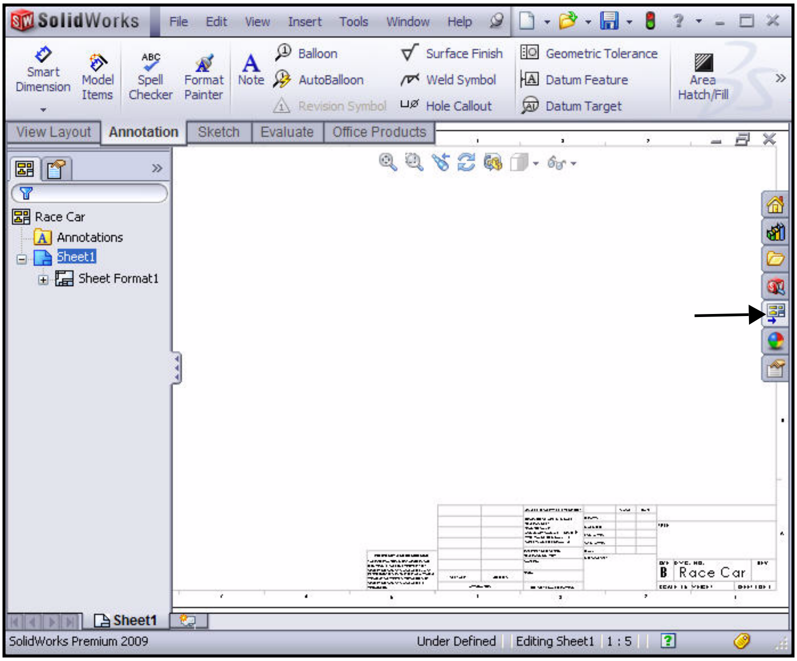
Select **16**.

6 Close the Choose Font dialog box.

Click **OK**.

7 Return to the graphics area.

Click **OK**.



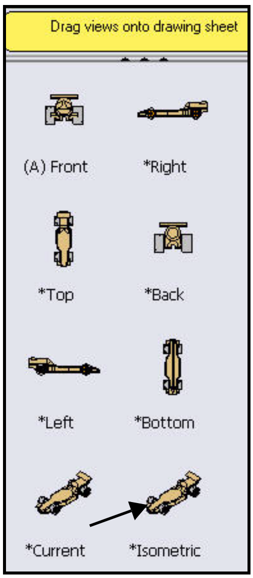
8 Insert an Isometric view.

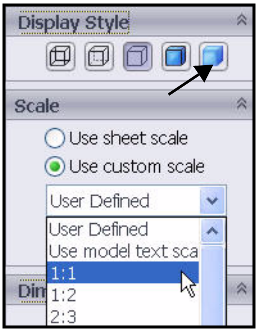
Use the View Palette to insert drawing views. The View Palette contains images of standard views, annotation views, section views, and flat patterns (sheetmetal parts) of the selected model. You can drag views into an active drawing sheet to create a drawing view.

If needed click the **View Palette** tab from the Task Pane.

Drag the ***Isometric** icon into Sheet1.

The Isometric view is displayed. The `Drawing View1` PropertyManager is displayed.





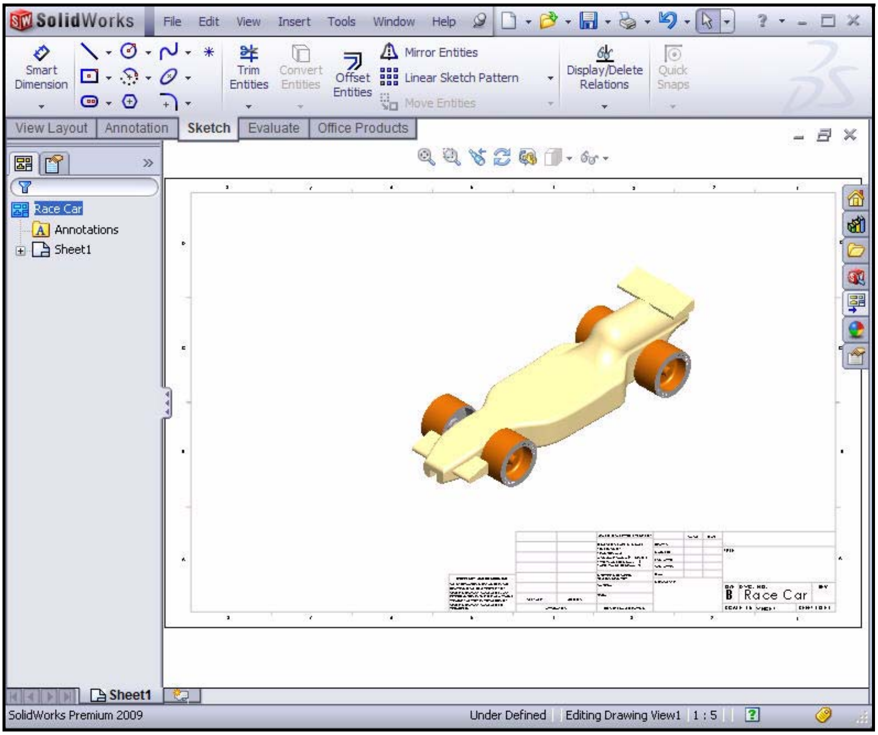
9 Modify the Sheet Scale and Display mode.
Check the **Use custom scale** box.

Select **1:1** from the drop-down menu.

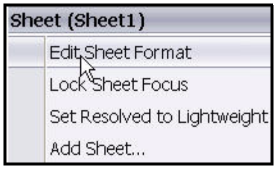
Click **Shaded** from the Display Style box.

Click **OK** from the `Drawing View1` PropertyManager.

10 Deactivate the Origins.
If needed, click **View**, un-check **Origins** from the Menu bar menu.



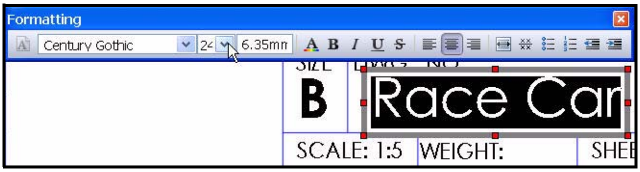
11 Edit the Title block.



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

Right-click inside **Sheet1**. Do not click inside the Isometric view.

Click **Edit Sheet Format**.



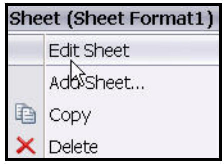
Zoom in on the Title block.

Double-click **Race Car** in the Title box.

Select **22** from the drop-down menu.

Click **OK** from the `Note` PropertyManager.

12 Return to the drawing.



Right-click **Edit Sheet**.

View the results.

13 Fit the drawing to the Sheet.

Press the **f** key.

14 Save the Drawing.

Click **Save**. Accept the default name.

Click **Save**.

Create a Bill of Materials.

Insert a Bill of Materials (BOM) into the drawing of the `Race Car` assembly. If you add or delete components in the assembly, the Bill of Materials automatically updates to reflect the changes if you select the Automatic update of BOM option under **Tools**, **Options, Document Properties, Detailing**.

Such changes include adding, deleting, or replacing components, changing component names or custom properties, and so on.

1 Create a Bill of Materials.

Click **inside** the Isometric view. The `Drawing View1` PropertyManager is displayed.

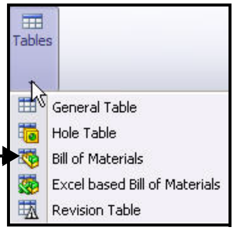
Click the **Annotation** tab from the CommandManager.

Click **Tables, Bill of Materials**. The `Bill of Material` PropertyManager is displayed. Accept the default settings. `Top level only` is selected by default. `bom-standard` is selected in the `Table Template` box.

Click **OK** from the `Bill of Material` PropertyManager.

Click a **position** in the top right corner of Sheet1.

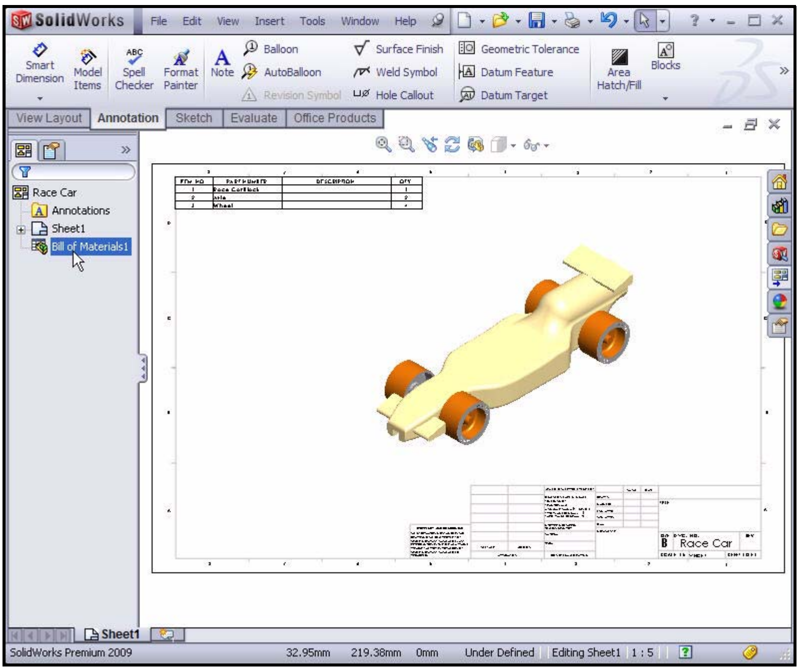
View the results.



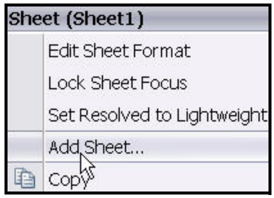
Note: You select a Sheet format when you open a new drawing. The standard Sheet formats include links to System properties and Custom properties.

2 Save the drawing.

Click **Save**.

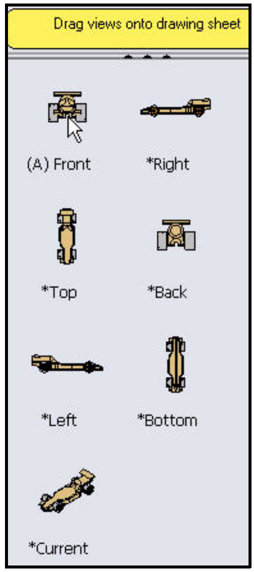


Add a Sheet to the drawing.



1 Add a Sheet in the drawing.
Right-click **Add Sheet**. Do not click inside the Isometric view. Sheet2 is displayed.

Insert a Front, Top, and Right view using the View Palette.



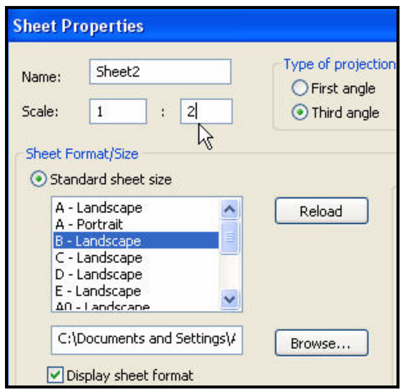
1 Insert a Front view.
Click the **View Palette** tab from the Task Pane.

Drag the ***Front** icon into Sheet2 on the bottom left corner. The Front view is displayed. The Projected View PropertyManager is displayed.

2 Insert a Top view.
Click a **position** directly above the Front view. The Top view is displayed.

3 Insert a Right view.
Click a **position** directly to the left of the Front view. The Right view is displayed.

Click **OK** from the Projected View PropertyManager. View the three views.



4 Modify the Sheet Scale.
Right-click inside **Sheet2**. Do not click inside a drawing view.

Click **Properties**.

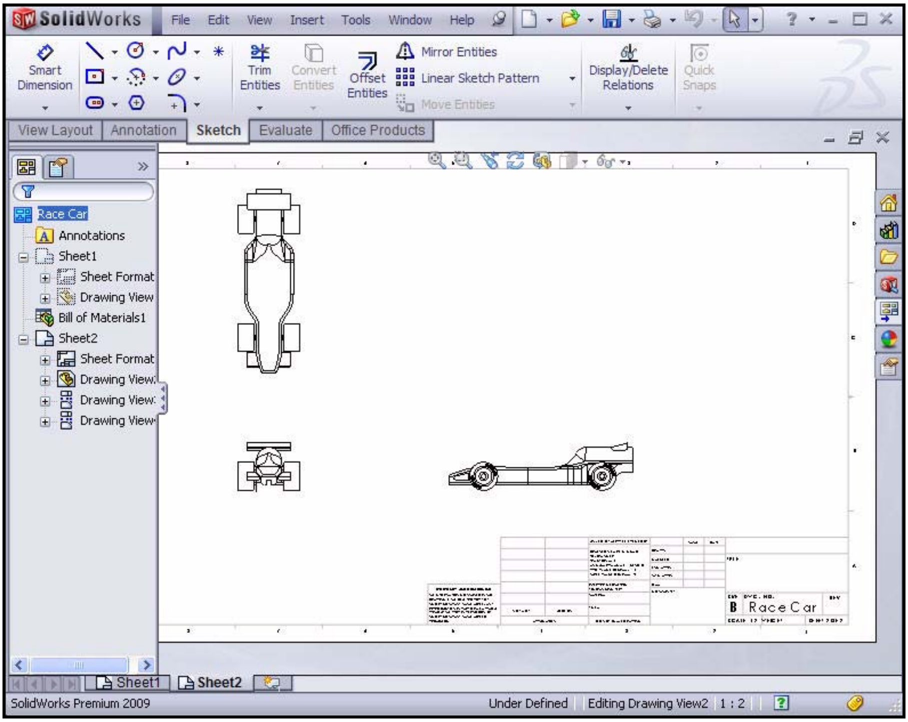
Enter **1:2** for `Scale`.

Click **OK** from the `Sheet Properties` dialog box.

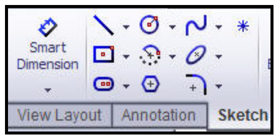
Click and drag **each view** into position.

5 Rebuild the drawing.
Click **Rebuild** from the Menu bar toolbar.

6 Save the drawing.
Click **Save**.



Insert a Right Drawing View Dimension.



1 Insert a Dimension into the Right view on Sheet2.
Zoom in on the Right View.

Click **Smart Dimension** from the Sketch toolbar.

Click the **left edge** of the `Race Car` in the Right view.

Note: Select an edge. View the icon feedback symbol.

Click the **right edge** of the `Race Car` in the Right view.

Click a **position** below the car to place the dimension. The overall dimension of the car is 210mm.

2 Insert two Dimensions into the Front view.

Press the **f** key to fit the model to the Sheet.

Zoom in on the Front View.

Click the **left front edge** of the Wheel.

Click the **right front edge** of the Wheel.

Click a **position** below the car to place the dimension.

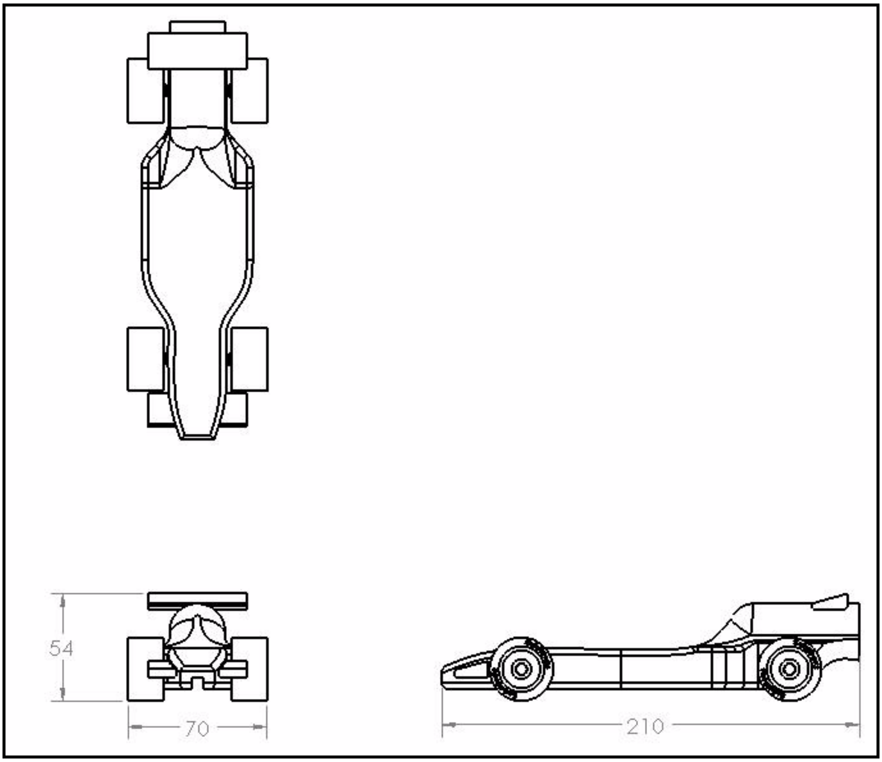
Click the **bottom** of the left front Wheel.

Click the **top** of the Top Wing.

Click a **position** to the left to place the dimension.

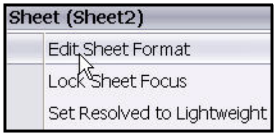
Click **OK**  from the `Dimension` PropertyManager.

Press the **f** key to fit the model to the Sheet. View the results.



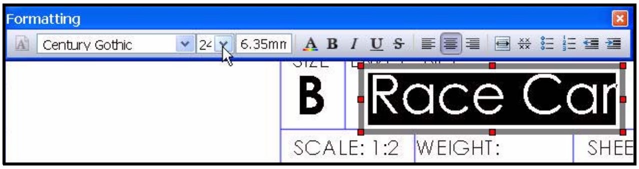
Note: The objective of this lesson is not to produce a completely dimensioned engineering drawing. Rather it is to introduce some of the basic steps engineers go through when producing documentation for a product. Add additional dimensions and information on the drawing if needed for the contest.

3 Edit the Title block on Sheet2.
The title of the drawing sheet is automatically filled in with information that is in the file properties of the assembly.



Right-click inside **Sheet2**. Do not click inside the views.

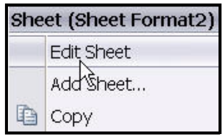
Click **Edit Sheet Format**.



Zoom in on the Title block.

Double-click on **Race Car.**

Select **22** from the drop-down menu.



Click **OK** from the `Note` PropertyManager.

Right-click **Edit Sheet**.

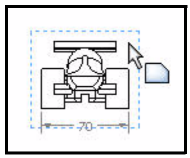
Rebuild the drawing.

4 Fit the model to the Sheet.
Press the **f** key.

5 Save the drawing.
Click **Save**.

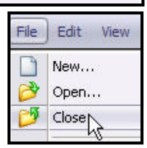
Open a Part from the Assembly

1 Open the Race Car Assembly from Sheet2.
Right-click inside the **Front** view.

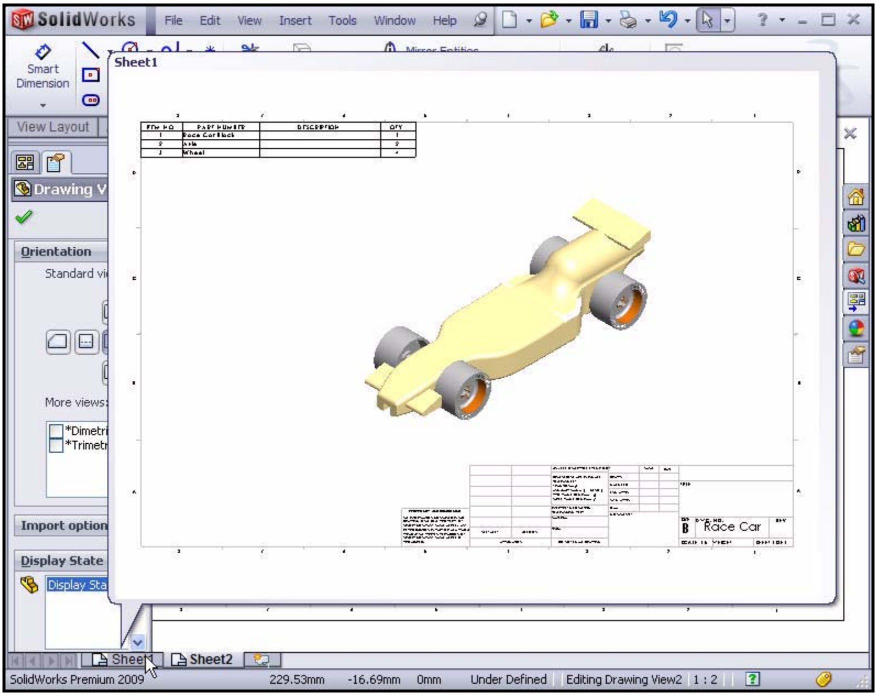


Click **Open race car.sldasm.** The `Race Car` assembly is displayed.

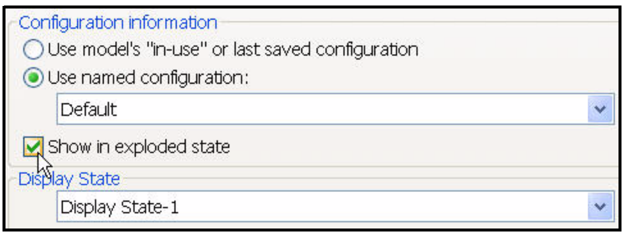
2 Return to the Race Car assembly drawing.
Click `File`, `Close` from the Menu bar menu. The `Race Car` drawing is displayed.



In the next section, return to Sheet1 and create an Exploded Isometric view.



Create an Exploded Assembly view

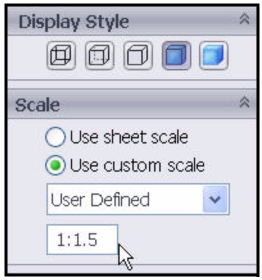


1 Return to Sheet1.
Click the **Sheet1** tab at the bottom of the graphics area to return to Sheet1.

2 Create an Exploded State.
Right-click inside the **Isometric** view.

Click **Properties**. The `Drawing View Properties` dialog box is displayed.

Check the **Show in exploded state** box.



Click **OK** from the `Drawing View Properties` dialog box.

3 Modify the View Scale.

Click inside the **Isometric** view in Sheet1. The Drawing View1 PropertyManager is displayed.

Check the **Use custom scale** box.

Select **User Defined**.

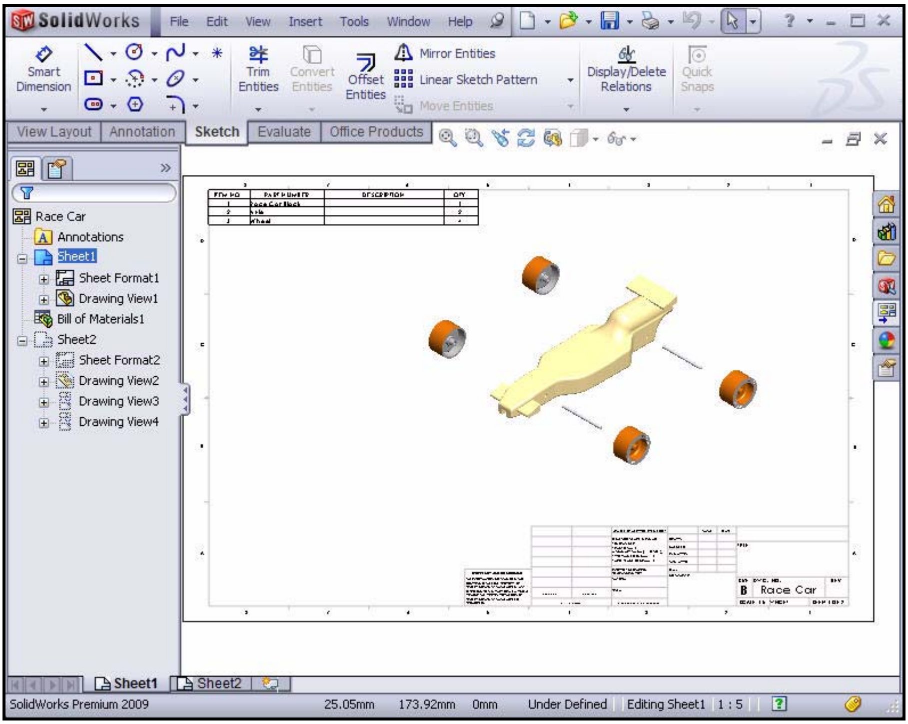
Enter **1:1.5**.

Click **OK** from the `Drawing View1` PropertyManager.

Rebuild the drawing.

4 Save the drawing.

Click **Save**. View the results. You are finished with the drawing section of this Project. You created an Exploded Isometric View with a Top level Bill of Materials on Sheet1, and created three views with inserted dimensions on Sheet2.



Lesson 4 PhotoWorks™

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

- Load the PhotoWorks `Add-in`
- Create a PhotoWorks assembly configuration
- Apply the Appearance tool to the `Race Car` assembly
- Apply the Scene tool
- Render the `Race Car` assembly
- Apply and edit the Decal tool on the `Race Car` assembly.
- Understand what makes an image look realistic and make changes to improve the realism of the rendering
- Save the PhotoWorks image

PhotoWorks

PhotoWorks is a best-in-class rendering solution for creating photorealistic images from 3D CAD models. Utilize PhotoWorks to help your colleagues visualize your designs more easily. PhotoWorks contains advanced visualization effects such as user-defined lighting, and an extensive library of appearances and textures as well as background scenery.

PhotoWorks allows you to render a model in an existing scene with lights. You select one of the studios and the scene and lights are automatically added and scaled to the size of the model. By default, images are rendered to the graphics area. You can also save images to a file in a variety of formats for printed materials and web pages.

With PhotoWorks you can define and modify the following elements of a rendering:

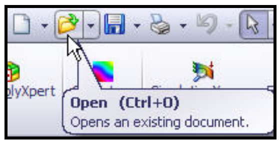
- `Scene`
- `Appearances`
- `Decals`
- `Lighting`
- `Image output formats`

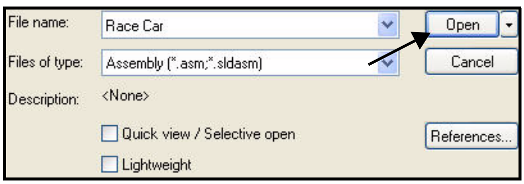
Activate PhotoWorks

Rendering is the process of applying the appearances, scene, lighting, and decal information to the model.

1 Open the Race Car Assembly.

Click **Open**  from the Menu bar toolbar.





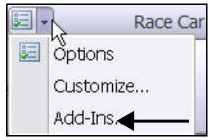
Browse to the location of the `Race Car` assembly.

Open the `Race Car` assembly.

The `Race Car` assembly is displayed in the graphics area.

2 Load the PhotoWorks Add-in.

Click the **Options**, **Add-ins…** from the Menu bar toolbar. The `Add-ins` dialog box is displayed.

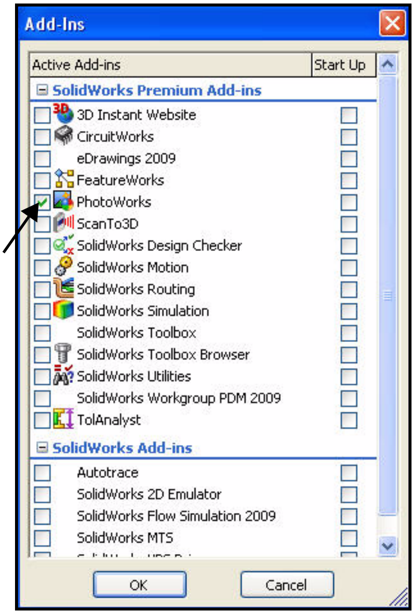


Check the **PhotoWorks** box.

Click **OK** from the `Add-Ins` dialog box.

The Render Manager tab is displayed in the FeatureManager and is updated in the Appearances/PhotoWorks tab in the Task Pane.

Click **Shaded With Edges** from the Heads-up View toolbar.



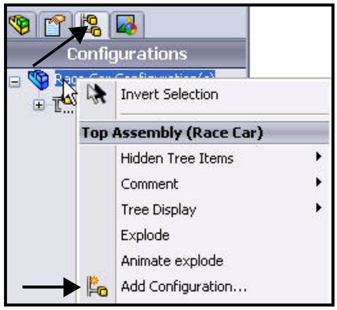
3 Display the PhotoWorks toolbar.

Click `View`, `Toolbars` from the Menu bar menu.

Check the **PhotoWorks** box. The PhotoWorks toolbar is displayed.

View the available tools and options.

Create 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.

1 Create a new configuration.

Click the **ConfigurationManager** tab.

Right-click **Race Car**.

Click **Add Configuration**. The Add Configuration PropertyManager is displayed.

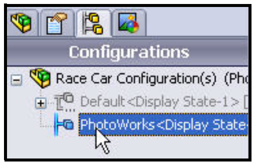
Note: The new configuration will be a copy of the active one.

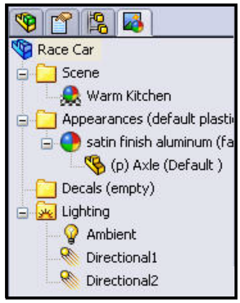
Enter **PhotoWorks** in the `Configuration name` box.

Enter **PhotoWorks** in the `Description` box.

Click **OK** from the `Add Configuration` PropertyManager.

View the new configuration.





2 View the new PhotoWorks Configuration.
Click the **PhotoWorks** Configuration in the ConfigurationManager.

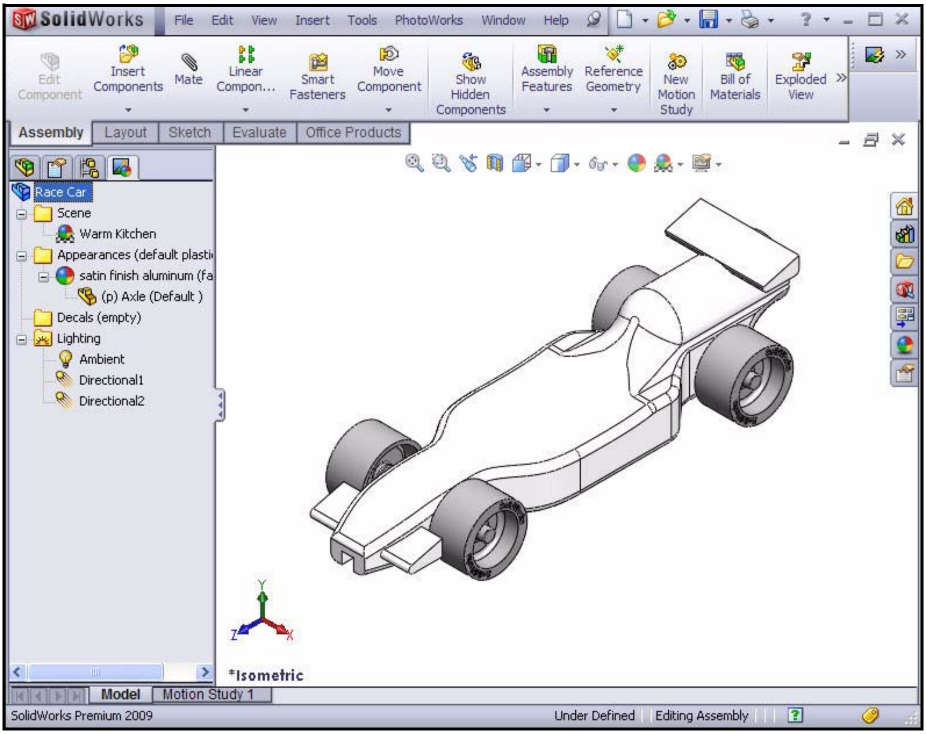
Click the **Render Manager** tab.

Expand `Scene`, `Appearances`, and `Lighting`.

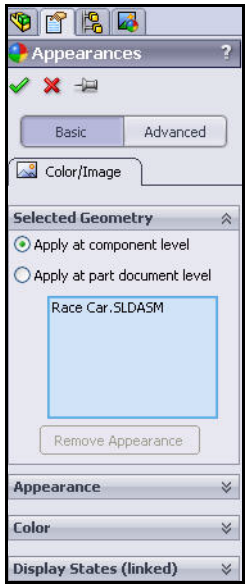
View the details.

3 Return to the FeatureManager.
Click the **FeatureManager** tab.

Note: The present configuration is PhotoWorks.



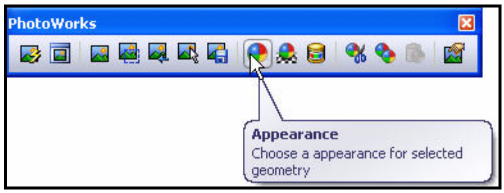
Appearance



PhotoWorks can use the appearance you applied when modeling the `Race Car` for the rendering. However, that isn't always what's best for a rendering. For example, when you modeled the `Race Car Block`, balsa material was used so we could calculate the mass. And to do that, you needed the correct material properties such as density.

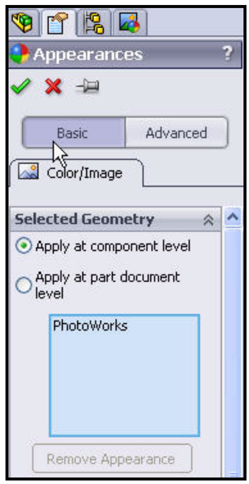
In the case of a rendering, you are more interested in what the car looks like, not what it is made of. So even though PhotoWorks can render engineering materials such as steel, copper, aluminum, and plastic, you can also apply and render materials such as rubber, leather, fabric, paint, etc.

4 **Apply Appearance to the Tires**.



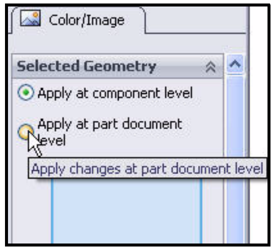
Click the **Appearance**  tool from the PhotoWorks toolbar. The `Appearances` PropertyManager is displayed.

Click the **Basic** tab from the `Appearances` PropertyManager.



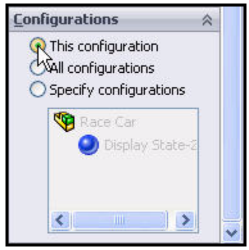
5 Apply changes at the part level.
You can apply changes at the part, feature, or assembly level.

Click the **Apply at part document level** box.

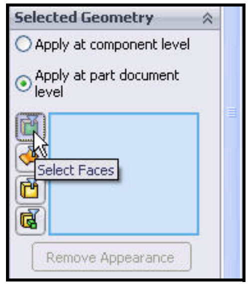


6 Apply changes to the PhotoWorks configuration.
The PhotoWorks configuration is the active configuration.

Check the **This configuration** box.

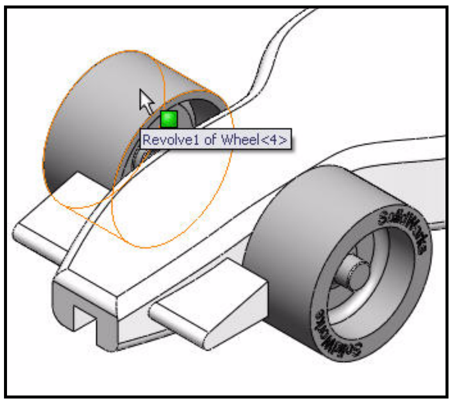


Click the **Select Faces** in the `Selected Geometry` box.



Click the **top face** of a tire in the graphics area.

The selected face is displayed in the `Selected Geometry` box.



Click the **Appearances/PhotoWorks** tab from the Task Pane as illustrated.

Expand the `Appearances` folder.

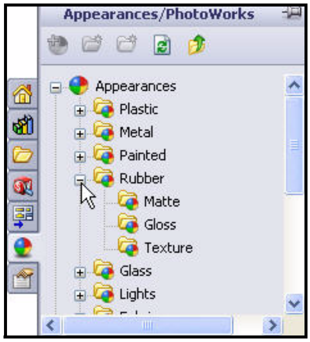
Expand the `Rubber` folder.

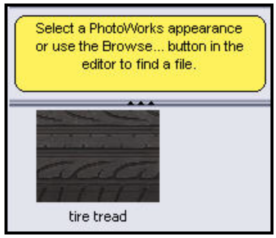
Click the `Texture` folder.

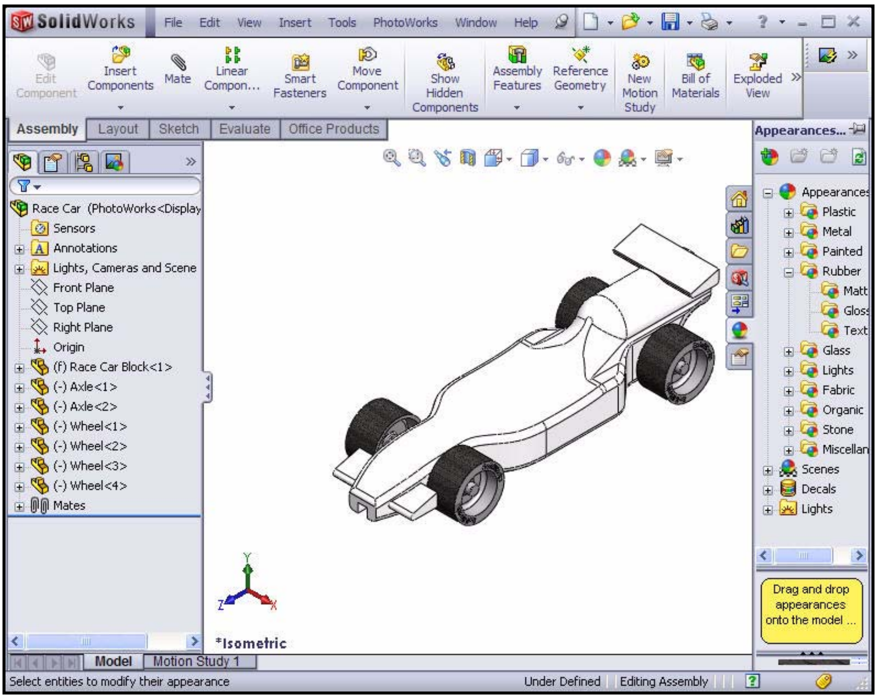
Click `tire tread`. The tire tread appearance is applied to the four tires in the graphics area.

Click **OK** from the `Appearances` PropertyManager.

View the results in the graphics area.

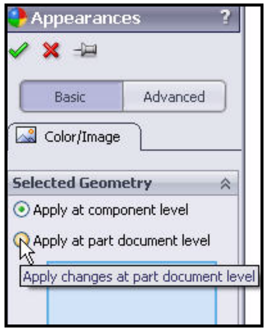






7 Apply Appearance to the Front and Rear Wing.

Click the **Edit Appearance**  tool from the Head-up View toolbar. The `Appearances` PropertyManager is displayed.

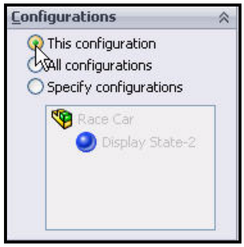


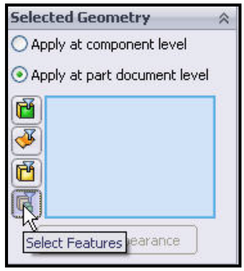
8 Apply changes at the part level. You can apply changes at the part, feature, or assembly level.

Click the **Apply at part document level** box.

9 Apply changes to the PhotoWorks Configuration. Check the **This configuration** box.

Click the **Select Features** box.





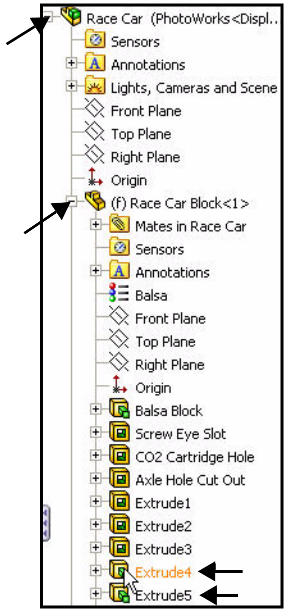
Expand `Race Car` from the fly-out FeatureManager.

Expand `Race Car Block`.

Click **Extrude4**. `Extrude4` is the front wing. `Extrude4` is displayed in the `Selected Geometry` dialog box.

Click **Extrude5.** `Extrude5` is the rear wing. `Extrude5` is displayed in the `Selected Geometry` dialog box.

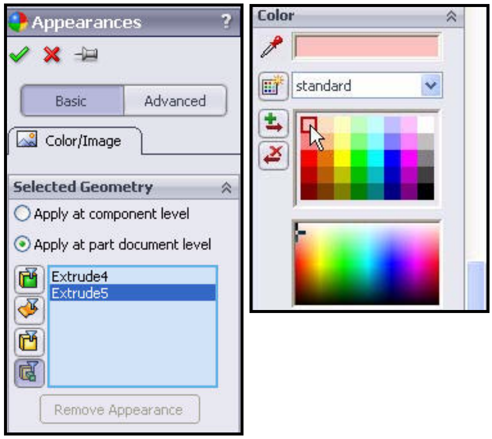
Select a **color**.

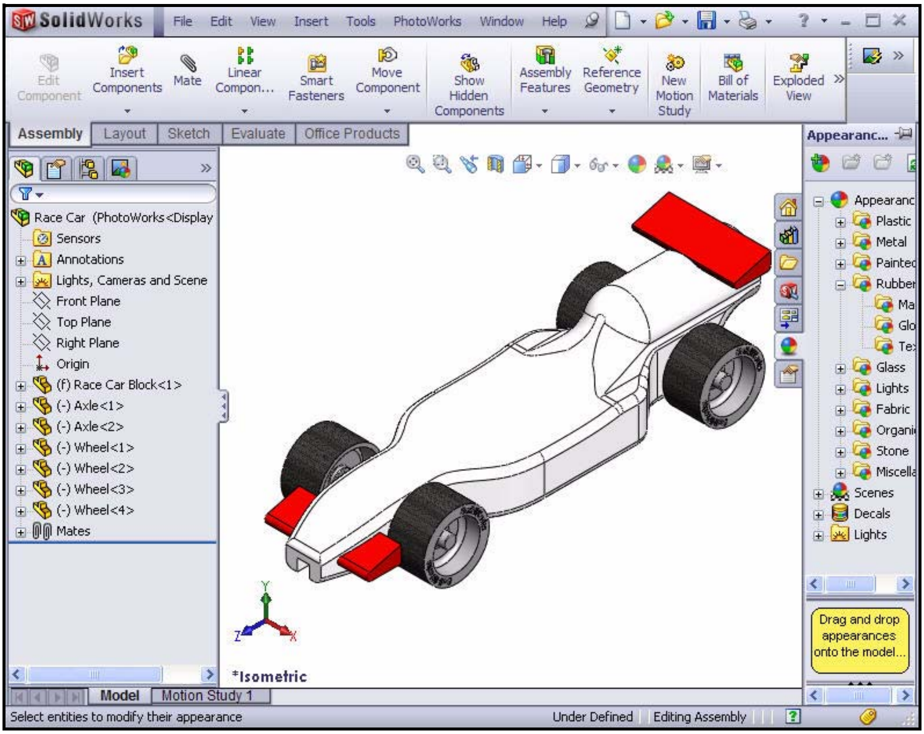


Note: A custom color can be selected and created by using the color palette in the `Color` dialog box.

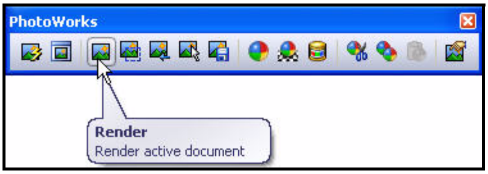
Click **OK** ✓ from the `Appearances` PropertyManager.

View the results.





Rendering



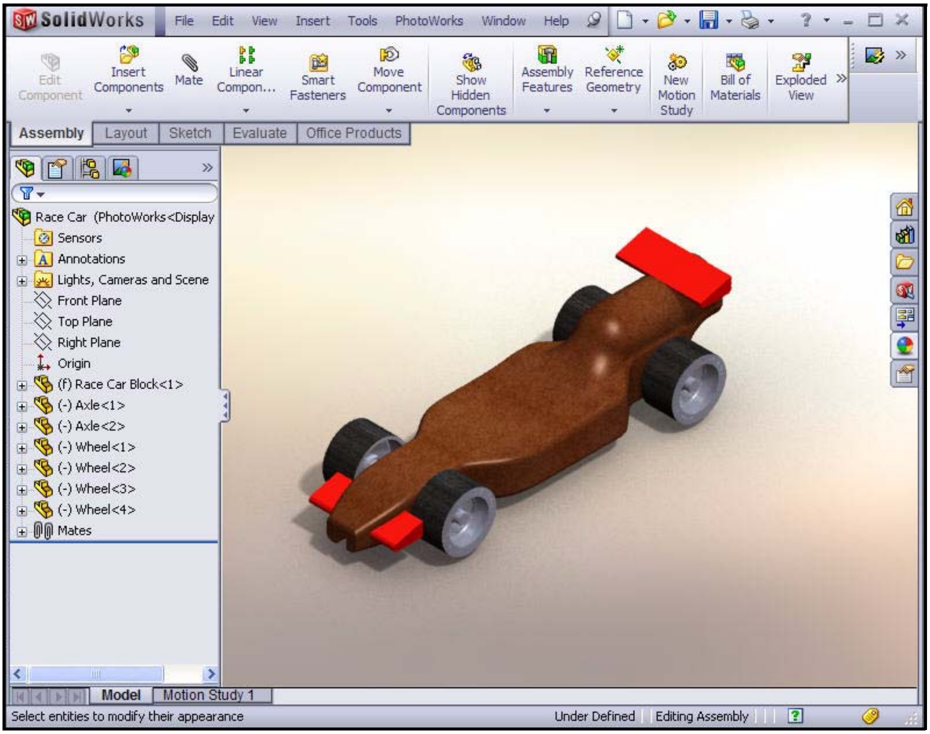
Rendering is the process of applying the appearance, scene, lighting, and decal information to the model. Full rendering applies all options set within PhotoWorks.

Note: Performing any operation that changes the view (zoom, pan or rotate) will remove the rendering.

1 Render the model.

Click the **Render** tool from the PhotoWorks toolbar.

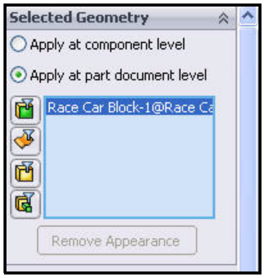
View the model in the graphics area.



Modify the Appearance

1 Modify the Appearance to the Race Car Block.

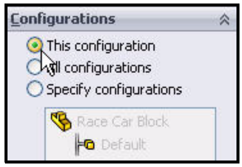
Press the **z** key to exit the render mode.

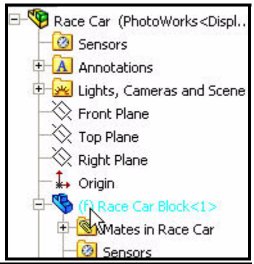


Click the **Appearance** tool. The `Appearances` PropertyManager is displayed. `Race Car` is displayed in the `Selected Geometry` box.

Click the **Apply at part document level** box.

Click **This Configuration** in the `Configurations` dialog box.





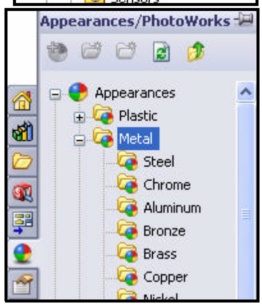
Click **Race Car Block** from the `Race Car` fly-out FeatureManager.

Expand the `Appearances` folder.

Expand the `Metal` folder.

Click `Silver`.

Click `matte silver`.

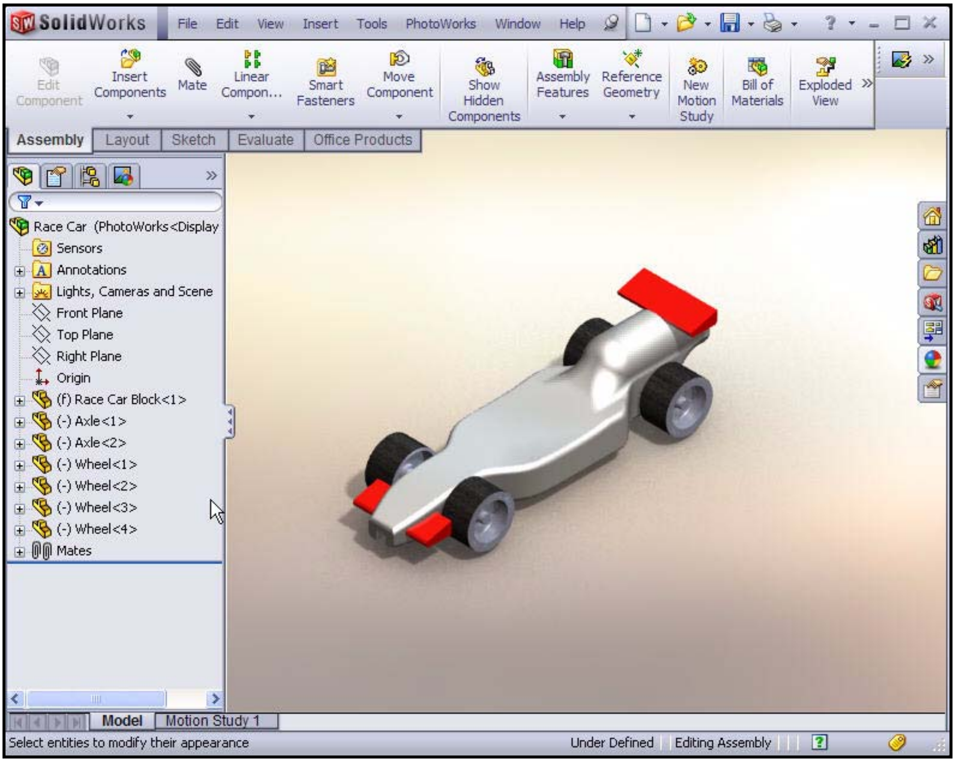


Click **OK** from the `Appearances` PropertyManager.

2 Render the model.

Click the **Render** tool from the PhotoWorks toolbar.

View the results.



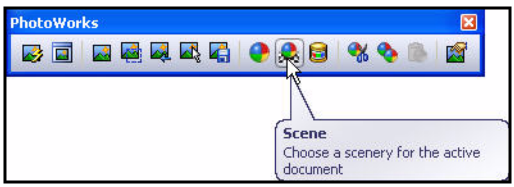
3 Save the model.

Press the **z** key to exit the render mode.

Click **Save** .

Scenes

PhotoWorks 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 box or sphere around the model. Scenes are composed of backgrounds, foreground effects, and scenery. PhotoWorks has a number of predefined scenes to make initial renderings quick and easy.



1 Apply the Scene tool.

Click the **Scene** tool from the PhotoWorks toolbar. The `Scene Editor` dialog box is displayed.

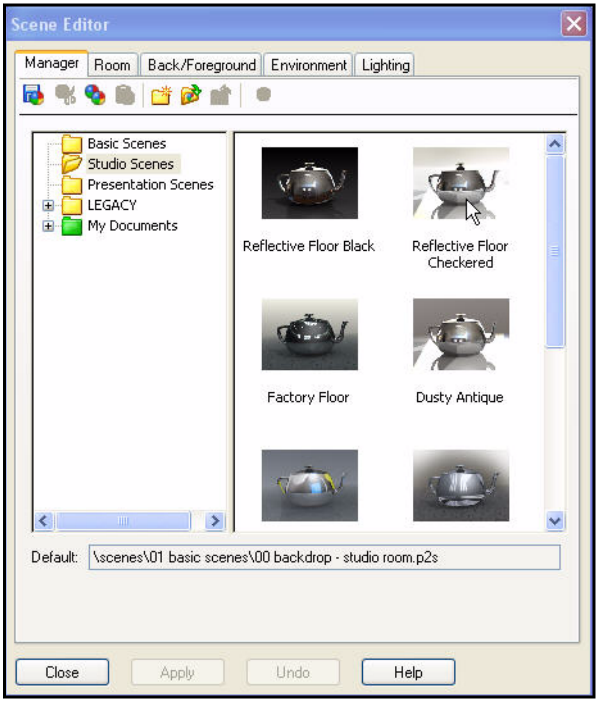
Click the **Manager** tab.

Click **Studio Scenes**.

Click **Reflective Floor Checkered**.

Click **Apply**.

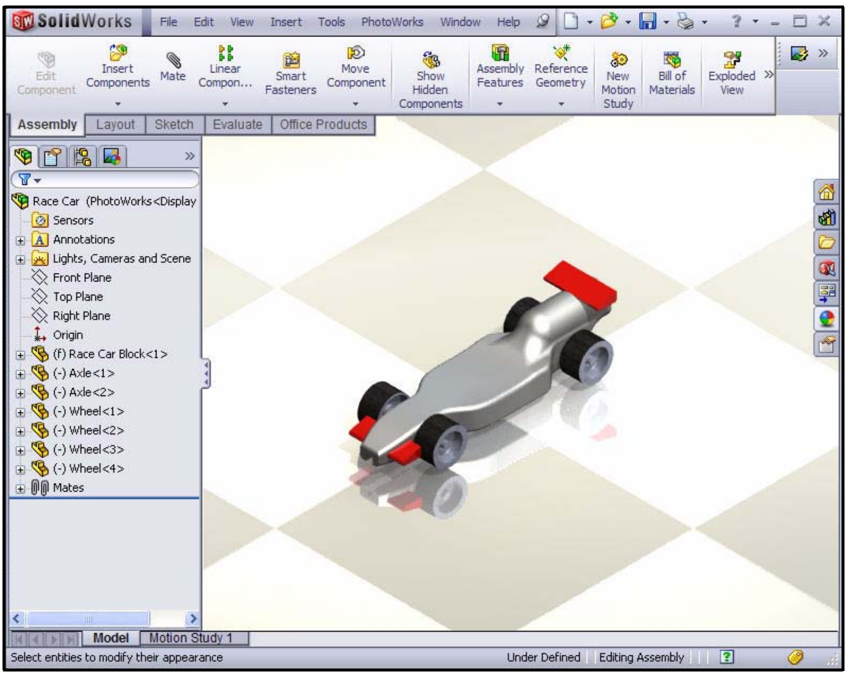
Click **Close**.



2 Render the model.

Click the **Render** tool from the PhotoWorks toolbar. View the model.

Press the **z** key to exit the render mode



Decals

Decals are artwork that are applied to the model. They are in some ways like textures 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 material 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`)

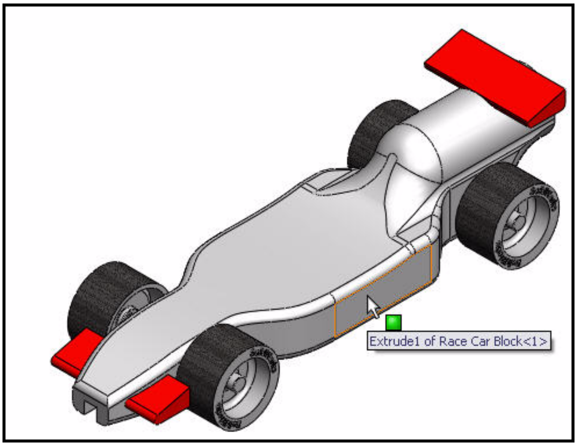
1 Apply a decal.

Click the **New Decal** tool from the PhotoWorks toolbar.

The `Decals` PropertyManager is displayed.

If required, click the **Appearances/PhotoWorks** tab in the Task Pane.

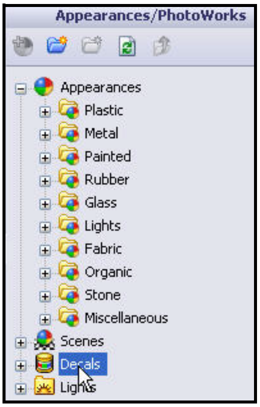
Click a position on the **right side** of the `Race Car Block` as illustrated.

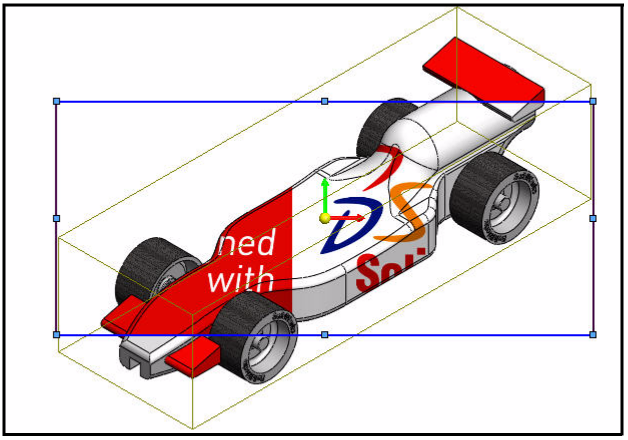


Click the `Decals` folder.

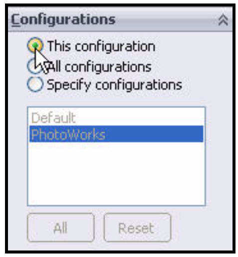
Click the `SolidWorks` decal.

The decal is displayed on the `Race Car Block`.





Check the **This configuration** box.



Position the Decal.

Click the **Mapping** tab from the Decals PropertyManager.

The decal is not positioned or scaled very well for the model.

Select **Projection** from the drop-down menu in the `Mapping` box.

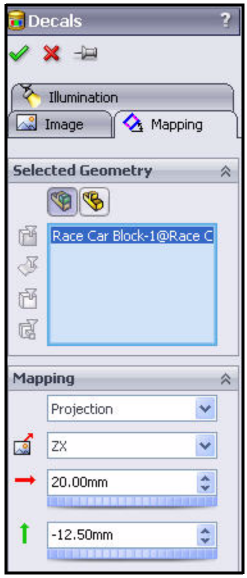
Select **ZX** from the drop-down menu for `Axis` direction.

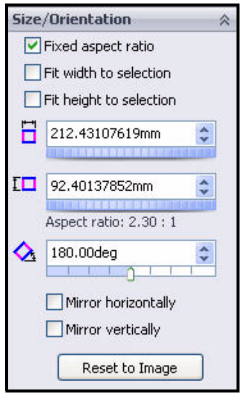
Enter **20.00**mm for `Horizontal` location.

Enter **-12.50**mm for `Vertical` location.

Enter **180.00deg** for `Rotation`.

Click **inside** the graphics area. View the results.

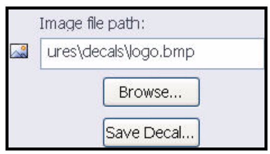


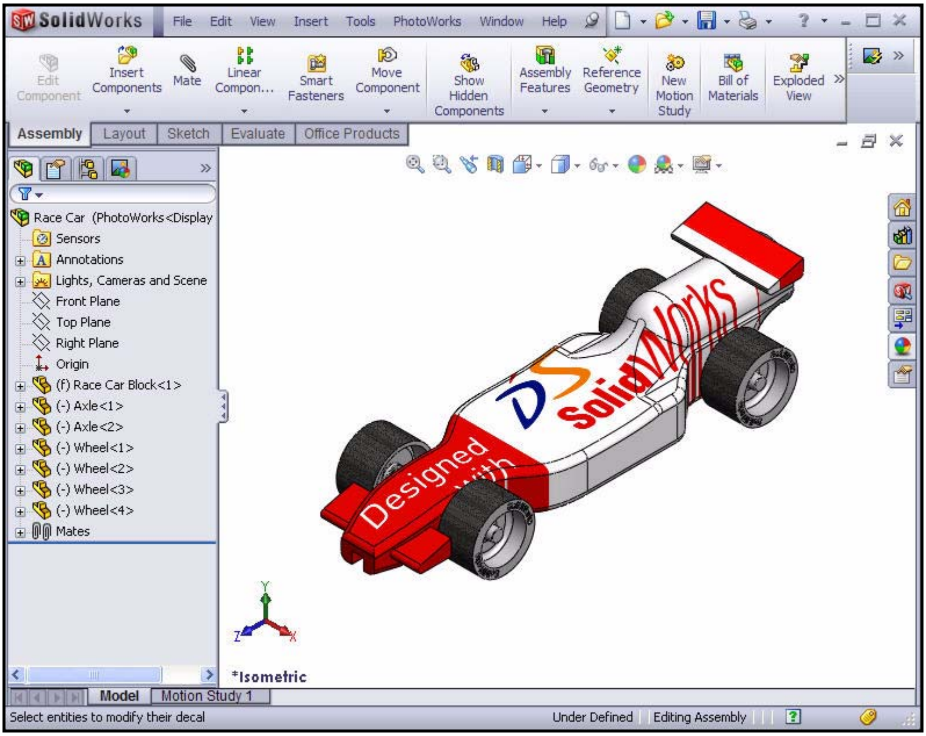


Click **OK** ✓ from the `Decal` PropertyManager.

View the results.

Tip: Create a decal from an existing file. Select the **Image** tab. Click the **Browse** button under the `Image file path`.

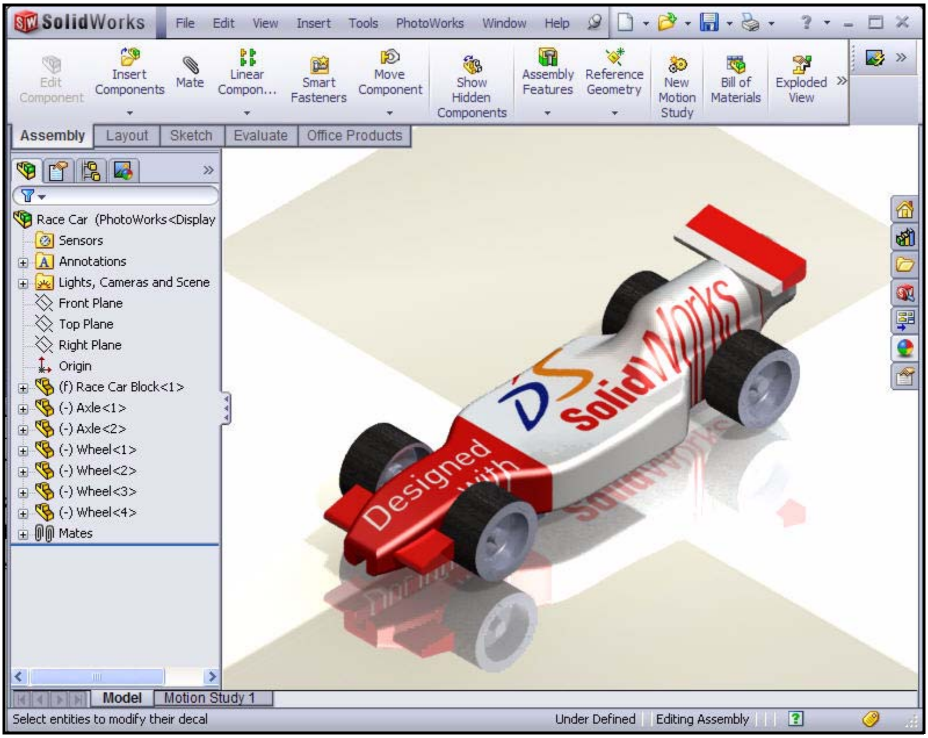




2 Render the model.

Click the **Render** tool  from the PhotoWorks toolbar.

View the model in the graphics area.



3 Save the model.

Click **Shaded** from the Heads-up View toolbar.

Press the **z** key to exit the Render mode.

Click **Save**.

4 Review the Render Manager.

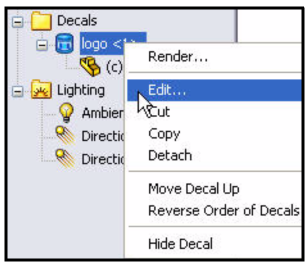
Click the **Render Manager** tab.

Expand each **folder**. View the results.

Edit the Decal

Right-click **logo <1>.**

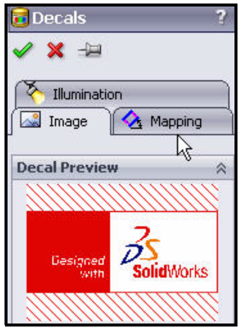
Click **Edit**. The `Decals` PropertyManager is displayed.

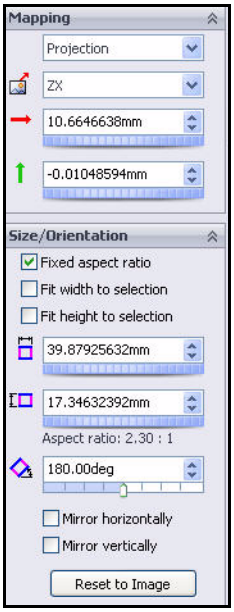


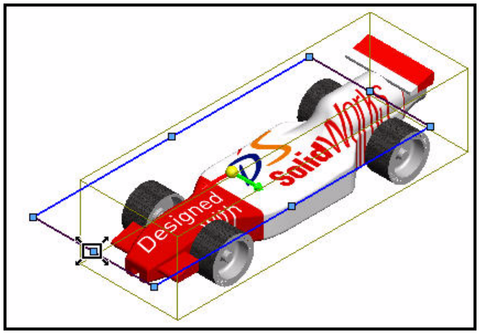
Click the **Mapping** tab.

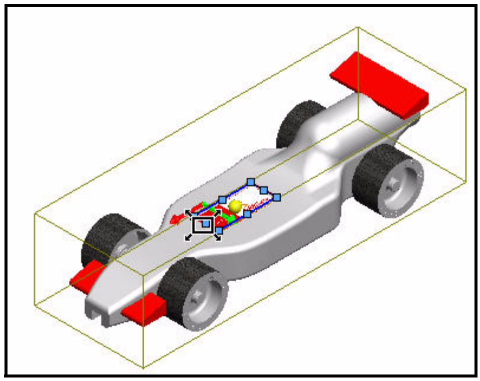
Use the **graphics view decal frame** to move, resize and rotate the decal. View the finished position of the decal from the PropertyManager.

Note: Dragging edges or anywhere inside the frame moves the image, dragging corners resizes, and dragging the center ball rotates the decal.









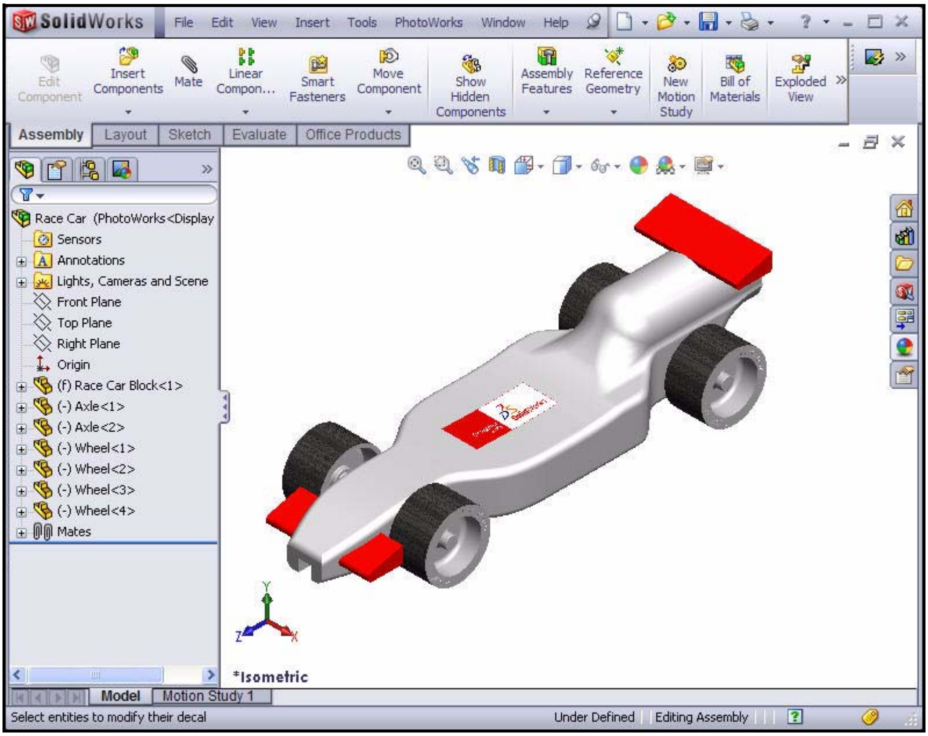
Click **OK** from the `Decals` PropertyManager.

5 Return to the FeatureManager.

Click the **FeatureManager** tab.

6 Save the model.

Click **Save**. You are finished with this section. Have fun. Explore with decals, appearances, lighting, scenes, etc.



Output Options

Rendering to the computer screen is generally done for two basic reasons:

- To visualize the effects of appearances and scenes. This is generally an intermediate step en route to the final output.
- To capture the image with screen capture software for use in other programs. The images for this manual were made as screen captures.

This is rarely the final output though.

Render to a Printer

Rendering directly to a printer is useful for creating a hard copy image of a project. This is a limited option because you cannot add captions, put multiple images on a page, or manipulate the image. Rendering to a printer is not useful for illustrations in Microsoft® Word or PowerPoint® because the hardcopy would have to be converted into a graphics file.

Some common uses of printer renderings might be for:

- Lobby displays of products before production begins;
- Display boards at conferences;
- Project reports.

To obtain rendered output from a printer, you must use the PhotoWorks print command, not the SolidWorks print command.

Rendering to a File

The most useful output method is to render the image to a 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 PhotoWorks software. This is known as the post-production phase.

File Types

Images can be rendered to the following file types:

- Windows Bitmap (`*.bmp`)
- TIFF (`*.tif`)
- TARGA (`*.tga`)
- Mental Ray Scene file (`*.mi`)
- JPEG (`*.jpg`)
- PostScript (`*.ps`)
- Encapsulated PostScript (`*.eps`)
- Silicon Graphics 8-bit RGBA (`*.rgb`)
- Portable pixmap (`*.ppm`)
- Utah/Wavefront color, type A (`*.rla`)
- Utah/Wavefront color, type B(`*.rlb`)
- Softimage color (`*.pic`)
- Alias color (`*.alias`)
- Abekas/Quantel, PAL (720x576) (`*.qntpal`)
- Abekas/Quantel, NTSC (720x486) (`*.qntntsc`)
- Mental images, 8-bit color (`*.ct`)

Methods to Increase Rendering Quality

The quality of the image file can vary depending on the options chosen in both SolidWorks and PhotoWorks. 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 PhotoWorks. For additional information about PhotoWorks, ask your teacher about getting a copy of *PhotoWorks Step-By-Step: A Self-Study Guide to Photorealistic Rendering*. It is available from your school's value-added SolidWorks reseller.

- Increase SolidWorks image quality.
  PhotoWorks 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 number of pixels rendered.
  Use a high dot per inch setting to render more pixels.
- Enable ray tracing.
  Ray tracing allows light to reflect from, and refract through, solids.
- Use a higher anti-aliasing setting.
  Higher settings for anti-aliasing reduce the jagged appearance of edges that are not vertical or horizontal.
- Increase shadow quality.
  Increasing shadow quality improves the edges of shadows.
- Enable indirect lighting.
  Indirect lighting adds light to surfaces that has been reflected by other surfaces.
- Enable caustics.
  Caustics add realism by adding the highlights caused by light refracting through transparent materials.
- Enable global illumination.
  Global illumination adds all forms of indirect illumination other than caustic effects. This includes color information and strength.

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.

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 72 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 300, 600 or 1200 dots per inch.

Multiply the printer resolution in dots per inch (dpi) times the desired size in inches.

The correct number of pixels can be calculated and entered directly, or you can specify the size of the image in inches or centimeters and the dots per inch and let PhotoWorks calculate the result.

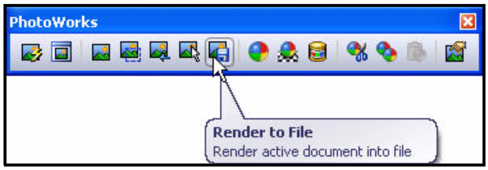
Example #1

Suppose we want to include a rendering of the `Race Car` in a Microsoft Word report which we are going to print on a 300 dpi printer. We want the image to be 5 inches wide and 3.75 inches high.

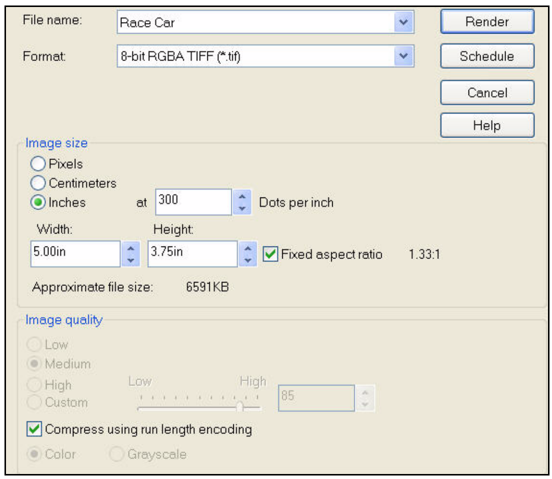
Multiplying the size of the desired image times the printer's dpi gives 1500 by 1125 pixels.

1 Render to file.

For good print quality, render this image as a TIFF file. This will result in a large file but with excellent definition.



Click **Render to File**  from the PhotoWorks toolbar.



Set the Look in directory to the `Race Car` folder.

Select **8-bit RGBA TIFF** for the Format.

Name the file `Race Car.tif`.

Select **Fixed aspect ratio**.

Select **Inches** for `Image` size.

Enter **5.00** for `Width`.

Enter **3.75** for `Height`.

Example #2

Suppose we want to incorporate our rendering into a PowerPoint presentation. PowerPoint presentations generally use images that are 96 dip. 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.

This yields a file size of about 816 KB.

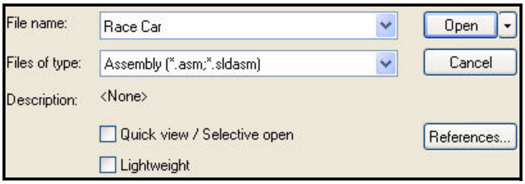
2 Save and close.
Save and **Close** all open files.

Lesson 5
Analysis

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

- Modify the Rear Wing of the `Race Car Block` to increase the mass
- Apply the Measure tool
- Apply the Mass Properties tool
- Apply SolidWorks SimulationXpress™ to the `Axle-A` part
- Modify the material on the `Axle-A` part and re-run SimulationXpress
- Save the SolidWorks SimulationXpress™ analysis
- Apply SolidWorks Flow Simulation™ to the initial `Race Car Block` assembly
- Apply SolidWorks Flow Simulation to the final `Race Car` assembly
- Compare the results
- Save the SolidWorks Flow Simulation analysis

Modify the Rear Wing



In Lesson 2, you created the Race Car assembly. You applied the Mass Properties tool and calculated the mass of the Race Car without paint, decals, sanding, etc. as 54.98grams. Increase the size of the rear wing to increase the total mass of the Race Car assembly.

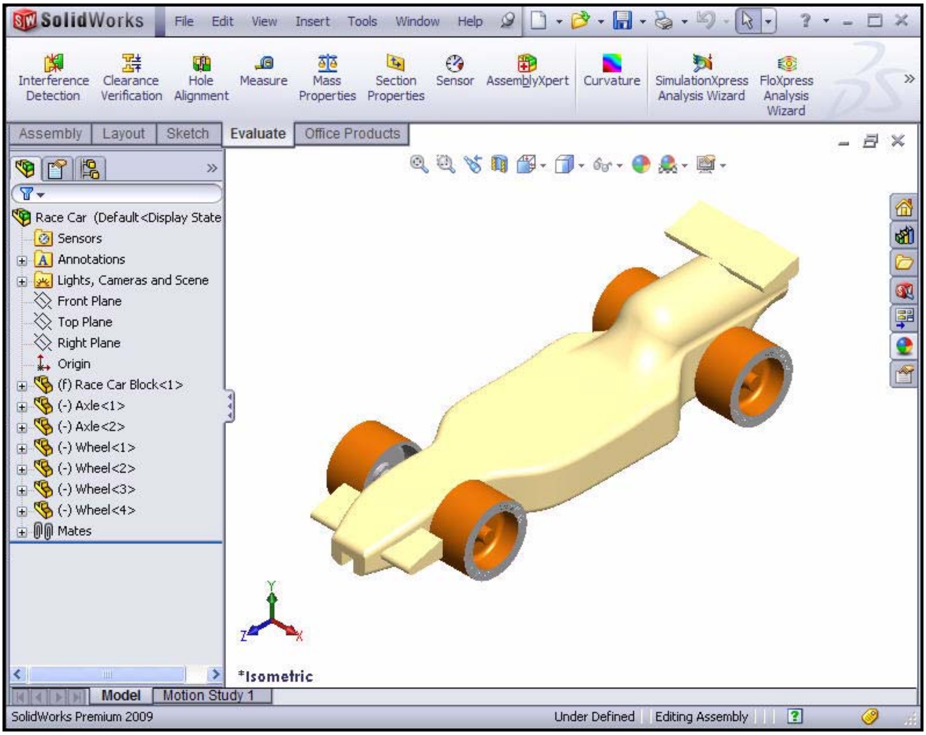
1 Open the Race Car Assembly.

Click **Open** from the Menu bar toolbar.

Browse to the location of the Race Car assembly.

Open the Race Car assembly.

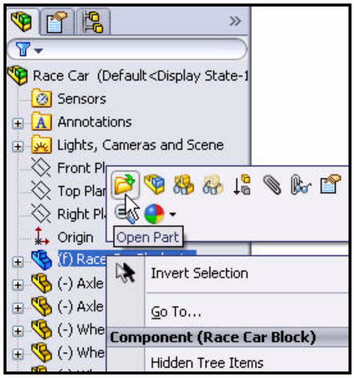
The Race Car assembly is displayed.



2 Open the Race Car Block Part.

Right-click `Race Car Block` from the FeatureManager.

Click **Open Part** from the Context toolbar. The `Race Car Block` FeatureManager is displayed.



3 Display the Rear Wind.

Click **Hidden Lines Removed** from the Heads-up View toolbar.

Click **Right** view from the Heads-up View toolbar.

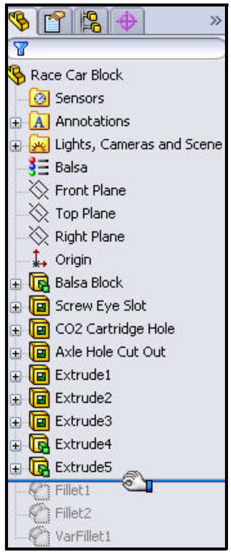
Press the **f** key to fit the model to the graphics area.

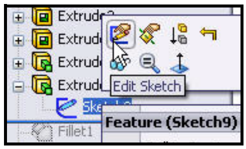
Drag the **Rollback bar** below `Extrude5`.

Expand `Extrude5`.

Right-click `Sketch9`.

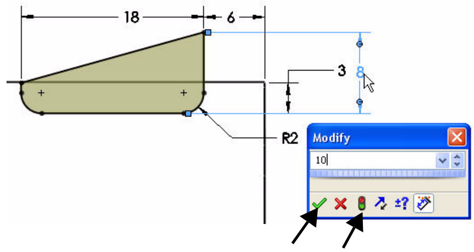
Click **Exit Sketch** from the Context toolbar.





Zoom in on the rear wing.

4 Modify the Height of the Rear Wing.



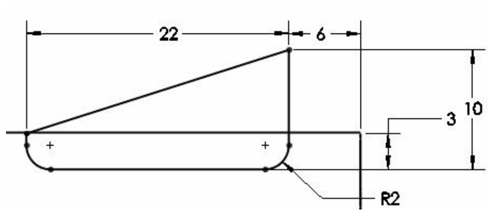
Double-click the **8** text dimension.

Enter **10** in the `Modify` dialog box.

Click the **Rebuild** tool.

Click the **Green check mark** in the `Modify` dialog box.

5 Modify the Width of the Rear Wing.



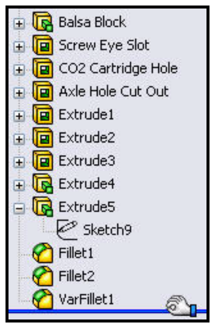
Double-click the **18** text dimension.

Enter **22** in the `Modify` dialog box.

Click the **Rebuild** tool.

Click the **Green check mark** in the `Modify` dialog box.

Click **OK** from the `Dimension` PropertyManager. View the modified rear wing dimensions.



Click **the Rebuild** tool.

Drag the **Rollback bar** below `VarFillet1` in the FeatureManager as illustrated.

Click **Shaded** from the Heads-up View toolbar.

6 Save the model.

Click **Isometric** view from the Heads-up View toolbar.

Click **Save** from the Menu bar toolbar.

7 Return to the Race Car Assembly.

Click `File`, `Close` from the Menu bar menu. The `Race Car` assembly is displayed.

Click **Yes** to rebuild.

Calculate the new Mass

You modified the height and width of the rear wing. Compare the original design to the modified design. Apply the Mass Properties tool. Measure the overall mass of the `Race Car` assembly.

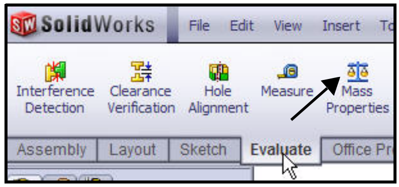
1 Apply the Mass Properties tool.
Click the **Evaluate** tab.

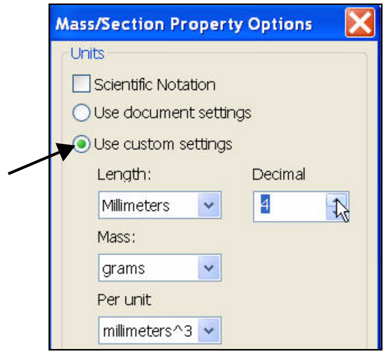
Click the **Mass Properties** tool from the Evaluate toolbar. The `Mass Properties` dialog box is displayed.

Click the **Options** button.

Check the **Use custom settings** box.

Select **4** for `Decimal` place.



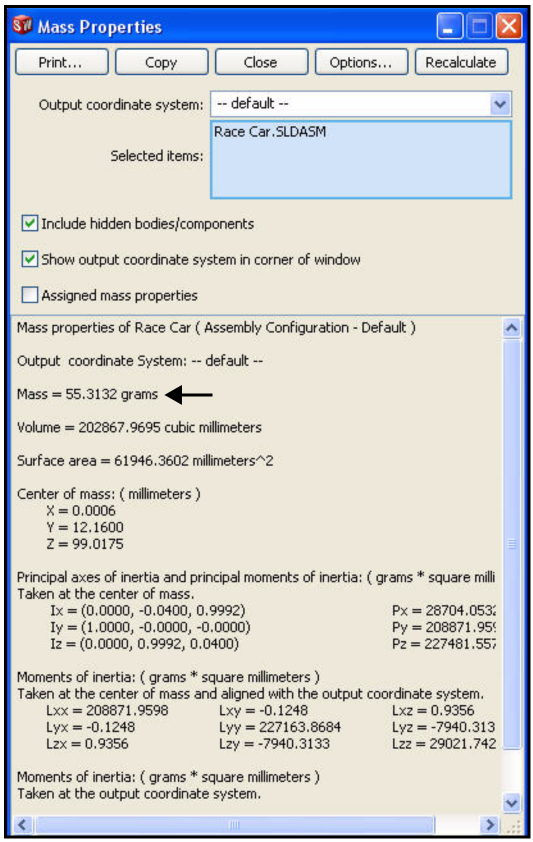


Click **OK** from the `Mass/Section Property Options` box.

View the new mass of the `Race Car` assembly. The new mass is approximately 55.31grams vs. 54.98.

Click **Close** from the `Mass Properties` dialog box.

Explore design changes to your `Race Car` assembly. Make sure that your final configuration meets the race contest requirements.

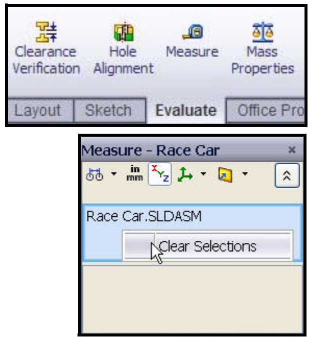


Apply the Measure tool

Apply the Measure tool to measure your modifications to the rear wing. You modified the rear wing in the `Race Car Block`.

Confirm your modified dimensions.

1 Apply the Measure tool.

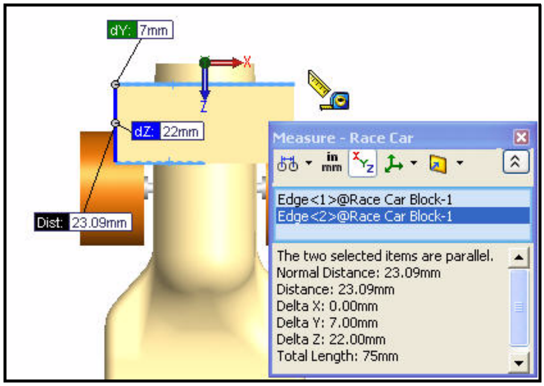


Click the **Measure** tool from the Evaluate toolbar. The `Measure - Race Car` dialog box is displayed.

Right-click **Clear Selections** in the `Selections` box.

Click **Top** view from the Heads-up View toolbar.

2 Measure the width of the rear wing.



Click the **front edge** of the rear wing.

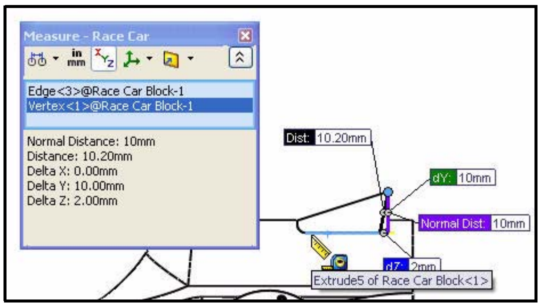
Click the **back edge** of the rear wing. 22mm is displayed.

3 Measure the height of the rear wing.

Right-click **Clear Selections** in the `Selections` box.

Click **Right** view.

Click **Hidden Lines Removed** from the Heads-up View toolbar.



Click the **bottom edge** of the rear wing.

Click the **top point** of the rear wing. View the dimensions.

Close the `Measure - Race Car` dialog box.

Click **Shaded With Edges** from the Heads-up View toolbar.

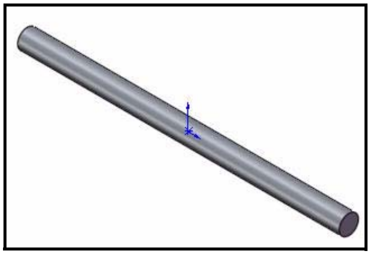
Click **Isometric** view.

4 Save the model.

Click **Save** from the Menu bar toolbar.

Click **Window**, **Close All** from the Menu bar menu. All models are closed.

Stress Analysis of the Axle



In this section, you will use SolidWorks SimulationXpress™ to quickly analyze the `Axle-A` part which is used in the `Race Car` assembly. Performing an analysis is very quick and easy to do. There are only five steps required:

1. Define material on the part.
2. Apply restraints.
3. Apply loads.
4. Analyze the part.
5. Optimize the part, (Optional).
6. View the results.

After performing a first-pass analysis on the `Axle-A` part and assessing its safety, you will change the material and rerun the analysis.

Design Analysis

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

- Will the part break?
- How will it deform?
- 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 of testing costly physical prototypes. 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 studies of many design options and aids in developing optimized designs.

Stress Analysis

Stress analysis or static analysis is the most common design analysis test. It predicts how the model deforms under loading. It calculates displacements, strains, and stresses throughout the part based on material, restraints, and loads. A material fails when the stress reaches a certain level. Different materials fail at different stress levels. SolidWorks SimulationXpress™ uses linear static analysis, based on the Finite Element Method (FEM), to calculate stresses.

Linear static analysis makes the following assumptions to calculate stresses in the part:

- **Linearity Assumption**. Means that the induced response is directly proportional to the applied loads.
- **Elasticity Assumption**. Indicates that the part returns to its original shape if the loads are removed.
- **Static Assumption**. Implies that loads are applied slowly and gradually until they reach their full magnitudes.

User Interface

SolidWorks SimulationXpress guides you through six steps to define material properties, restraints, loads; to analyze the part; optimize the part; and to view your results. The SolidWorks SimulationXpress interface consists of the following components:

Welcome tab: Allows you to set the default units and to specify a folder for saving the analysis results.

Material tab: Applies material properties to the part. The material can be assigned from the material library or you can input the material properties.

Restraint tab: Apply restraints to faces of the part.

Load tab: Apply forces and pressures to faces of the part.

Analyze tab: You can select to analyze with the default settings or change the settings.

Optimize tab: Optimize a model dimension based on a specified criterion.

Results tab: View analysis results in the following ways:

- Show critical areas where the factor of safety is less than a specified value.
- Display the stress distribution in the model with or without annotation for the maximum and minimum stress values.
- Display resultant displacement distribution in the model with or without annotation for the maximum and minimum displacement values.
- Show deformed shape of the model.
- Generate an HTML report.
- Generate eDrawings files for the analysis results.

Start Over button: Click this button to delete existing analysis data and results and start a new analysis session.

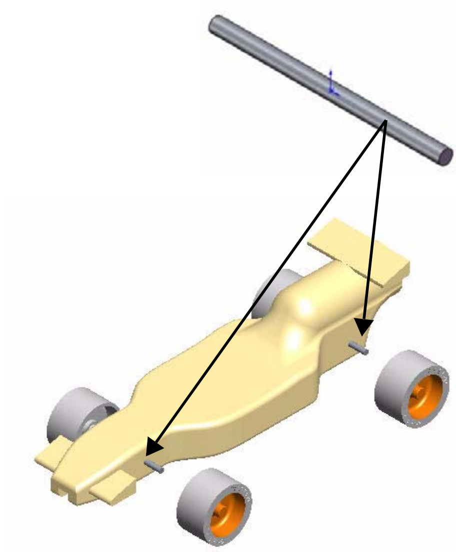
Update button: Runs SolidWorks SimulationXpress analysis if the restraints and loads are resolved. Otherwise, it gives a message and you need to resolve the invalid restraints or loads. The Update button appears if you change geometry after applying loads or restraints. It also appears if you change material properties, restraints, loads, or geometry after completing the analysis. Once any of these is changed, exclamation marks appear on the Analyze and Results tabs. An exclamation mark on the Restraint or Load tab indicates that a restraint or load became invalid after a change in geometry.

Analyze the Axle-A Part

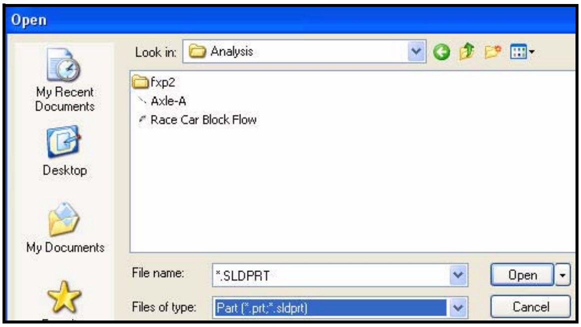
Browse to the downloaded Analysis folder and open the `Axle-A` part in this section.

Perform a stress analysis on the `Axle-A` part.

The `Axle-A` part is a renamed part of Axle that is used in the `Race Car` assembly.



Open the Axle-A Part



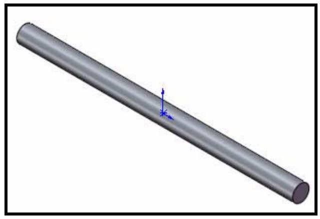
1. **Open the Axle-A part.**

   Click **Open** from the Menu Bar toolbar.

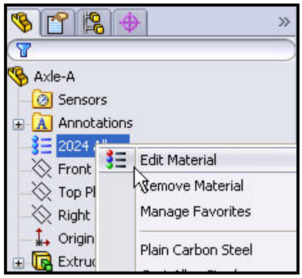
   Select the **folder** in which you downloaded the Analysis folder.

   Set `Files of type`: **Part**.

   Double-click **Axle-A**. The `Axle-A` part is displayed in the graphics area.

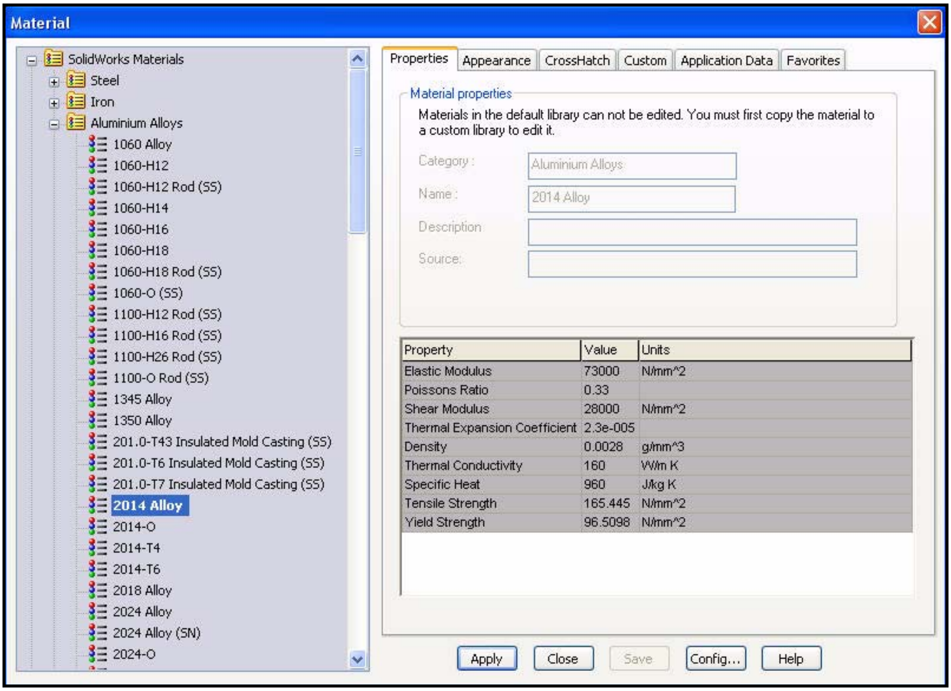


2. **Change the view orientation.**
   If the part is not displayed in an Isometric view, click **Isometric** view from the Heads-up View toolbar.



3. **Review the material.**
   Right-click **2024 Alloy** in the FeatureManager.

   Click **Edit Material**. The physical material properties are displayed in the `Materials` dialog box.



Note: The 2024 Alloy material properties are used in SimulationXpress.

4 Return to the FeatureManager.
Click **Close** from the `Materials` dialog box.

SolidWorks SimulationXpress

Once the part is open in SolidWorks, you can launch the SolidWorks SimulationXpress application and start your analysis right away. On the Options dialog box, you can set the default system of units and the destination folder for the analysis results.

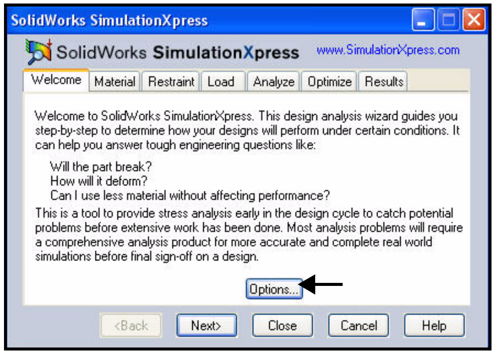
Systems of Units

The following table lists the quantities used by SimulationXpress and their units in different systems of units:

		SI	English (IPS)	Metric
Loads	Force	N (Newton)	lb (pound)	Kgf
	Pressure	N/m^2	psi (lb/in^2)	Kgf/cm^2
Material Properties	Ex: Elastic modulus	N/m^2	psi (lb/in^2)	Kgf/cm^2
	NUXY: Poisson's ratio	No units	No units	No units
	SIGYLD: Yield Strength	N/m^2	psi (lb/in^2)	Kgf/cm^2
	DENS: Mass density	Kg/m^3	lb/in^3	Kg/cm^3
Results	Equivalent Stress	N/m^2	psi (lb/in^2)	Kgf/cm^2

Table 1: Systems of units used in SimulationXpress

Running SimulationXpress and Setting Analysis Options



1. **Run SolidWorks SimulationXpress.**
   Click **Tools**, **SimulationXpress** from the Menu Bar menu.

   The SolidWorks SimulationXpress application starts with the Welcome tab selected.

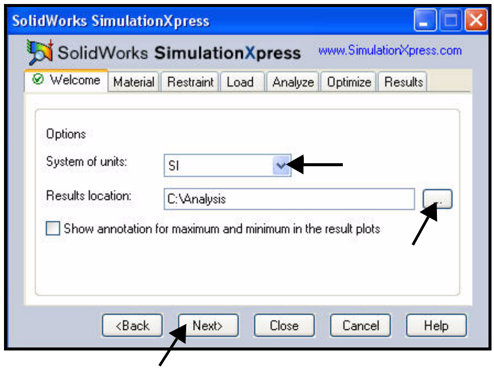
Tip: You can quickly run SimulationXpress by clicking **SimulationXpress Analysis Wizard** from the Evaluate tab in the CommandManager.

2. **Set System units.**
   Click the **Options** button from the `Welcome` screen.

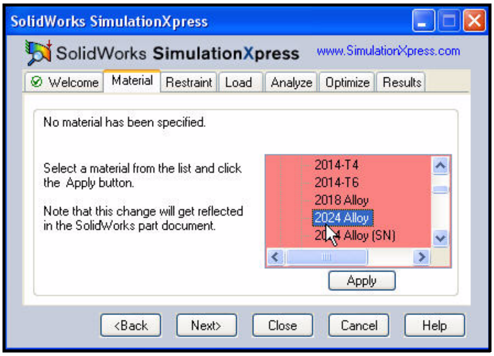
   Set `System of units` to **SI**, **(MMGS)**.

   Set the `Results location` to the **Analysis** folder.

   Click **Next>**.



Assigning Material



The response of the part depends on the material it is made of. SimulationXpress must know the elastic properties of the material of your part. You can pick a material from the SolidWorks material library or define your own material properties. SimulationXpress uses the following material properties to perform stress analysis.

Elastic Modulus (EX). For a linear elastic material, the elastic modulus is the stress required to cause a unit strain in the material. In other words, stress divided by the associated strain. The modulus of elasticity was first introduced by Young and is often called the Young's Modulus.

Poisson's Ratio (NUXY). Extension of the material in the longitudinal direction is accompanied by shrinking in the lateral directions. For example, if a body is subjected to a tensile stress in the X-direction, then Poisson's Ratio NUXY is defined as the ratio of lateral strain in the Y-direction divided by the longitudinal strain in the X-direction. Poisson's ratios are dimensionless quantities. If not defined, the program assumes a default value of 0.

Yield Strength (SIGYLD). SimulationXpress uses this material property to calculate the factor of safety distribution. SimulationXpress assumes that the material starts yielding when the equivalent (von Mises) stress reaches this value.

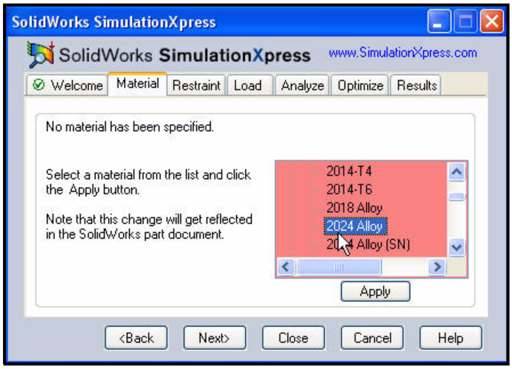
Mass Density (DENS). The density is mass per unit volume. Density units are lb/in^3 in the English system, and kg/m^3 in the SI system. SimulationXpress uses the mass density to include mass properties of the part in the report file.

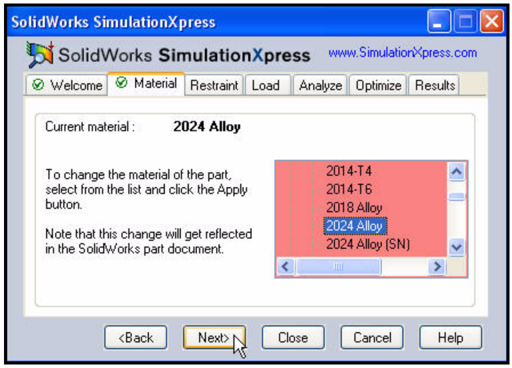
Assigning Material

1 Assign Material.
Click **2024 Alloy** as illustrated.

Click **Apply**.

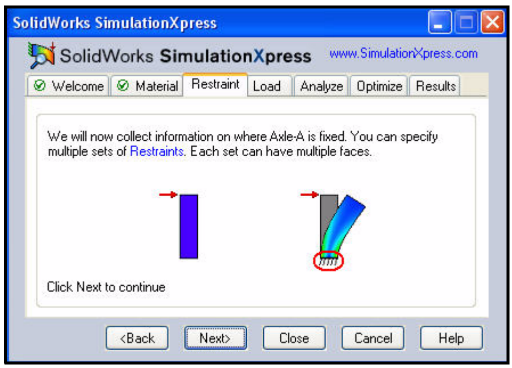
Click **Next>**.





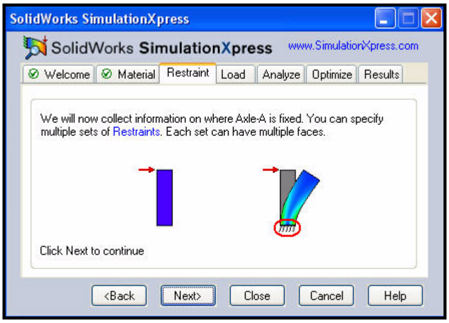
Applying Restraints

A part that is not restrained will travel indefinitely in the direction of the applied load as a rigid body. In the Restraint section, you define how the `Axle-A` part is fastened in the analysis. The restrained faces are fixed in space. You must restrain one face of the part to prevent the analysis program from stopping, due to instability caused by rigid body motion.



Applying a Restraint

1 Apply a restraint.
The Restraint tab is activated. The Restraint section collects information on where the `Axle-A` part is fixed. You can specify multiple sets of restraints. Each set can have multiple faces.

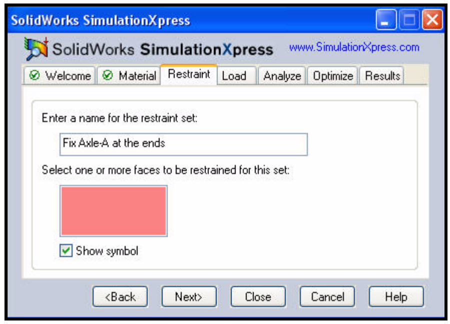


2 Name the restraint.
Click **Next>**.

Delete the **Restraint1** text.

Enter **Fix Axle-A at the ends**.

Note: It is recommended that you use meaningful names for the restraints.



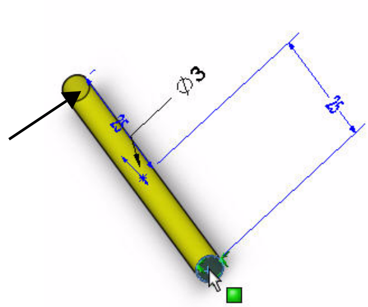
3 Select the restrained faces.
Click the **outside right face** of the `Axle-A` part.

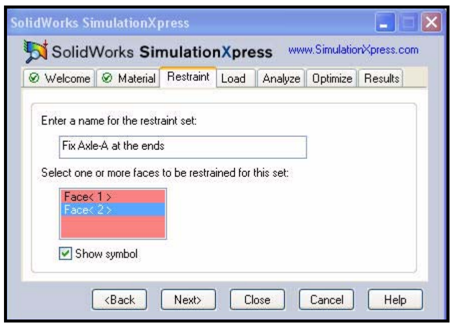
Click the **outside left face** of the `Axle-A` part as illustrated.

`Face <1>` and `Face<2>` are displayed in the `Selection` box.

Click **Next>**.

Note: To add a new restraint set, click **Add**. To edit or delete an existing set, perform the proper functions in the box.

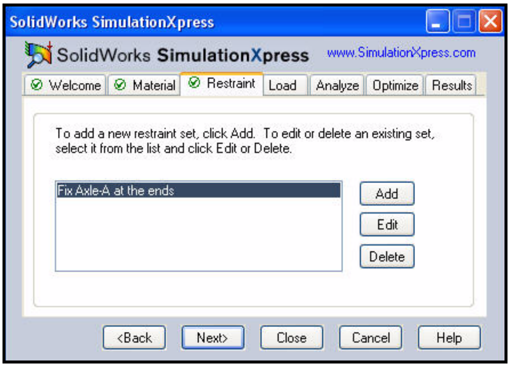


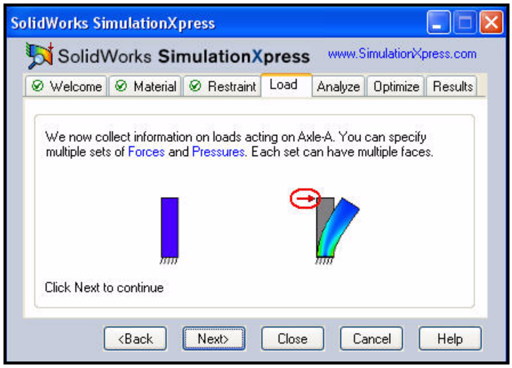


Applying a Load

Using the Load tab, you can specify the loads acting on the part. A load can either be a force or a pressure.

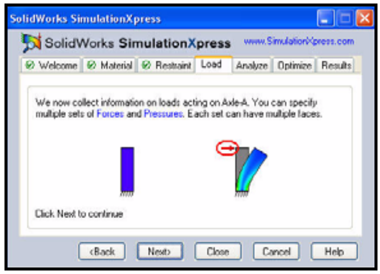
You can apply multiple loads to a single face or to multiple faces. The direction of a force can be specified with respect to planes or normal to selected faces. The pressure is always applied normal to selected faces.





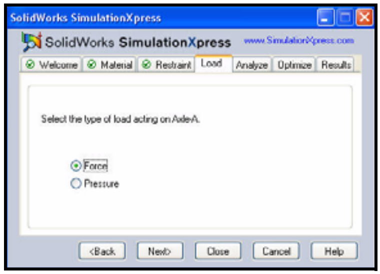
Applying a Load

1 Apply a load.
Click **Next>**. Collect information on loads acting on the Axle-A part. You can specify multiple sets of forces or pressures. Each set can have multiple faces.



Click **Next>**.

2 Select a load type.
Select **Force**.

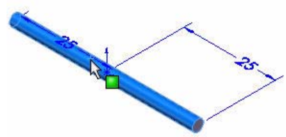


Click **Next>**.

3 Enter a name for the force.
Delete the **Load1** text.

Enter **1N** for `load` text.

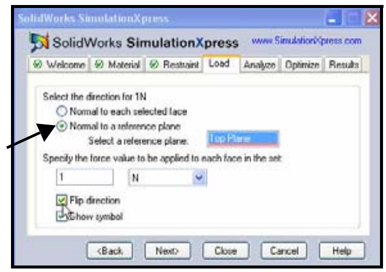
4 Select the face to which the force is applied.
Click the **cylindrical face** of the `Axle-A` part.



`Face <1>` is displayed in the selection box.

Click **Next>**.

5 Specify the direction and magnitude of the force.
Check the **Normal to a reference plane** box.

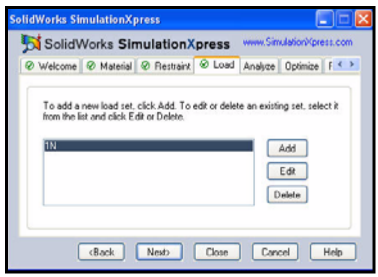


Click **Top Plane** from the FeatureManager.

Check the **flip direction** box. The force arrows point downwards.

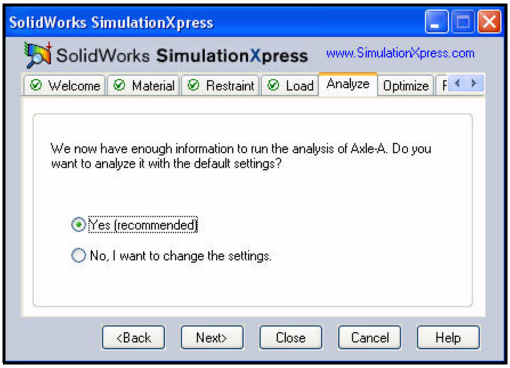
Click **Next>**.

6 Analyze the results.
Click **Next>**.



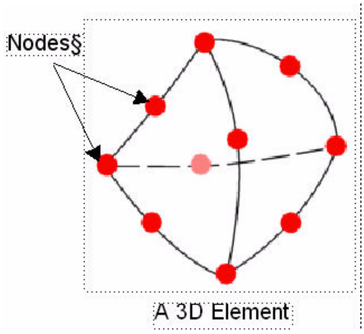
The Analyze tab opens.

Running the Analysis



The Analyze tab allows you to run the analysis. SimulationXpress prepares the model for analysis and then calculates displacements, strains, and stresses.

The first phase in the analysis is meshing. Meshing is basically splitting the geometry into small, simple-shaped pieces called finite elements.



Design analysis uses finite elements to calculate the model's response to the applied loads and restraints. SimulationXpress estimates a default element size for the model based on its volume, surface area, and other geometric details. You can instruct SimulationXpress to use the default element size or you can use a different element size.

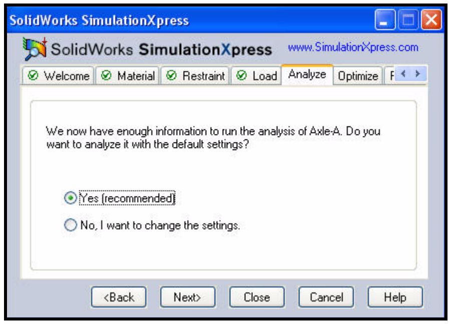
After meshing the model successfully, the second phase starts automatically. SimulationXpress formulates the equations governing the behavior of each element taking into consideration its connectivity to other elements. These equations relate the displacements to known material properties, restraints, and loads. The program then organizes the equations into a large set of simultaneous algebraic equations. The solver finds the displacements in the X, Y, and Z directions at each node.

Using the displacements, the program calculates the strains in various directions. Finally, the program uses mathematical expressions to calculate stresses.

Running the Analysis

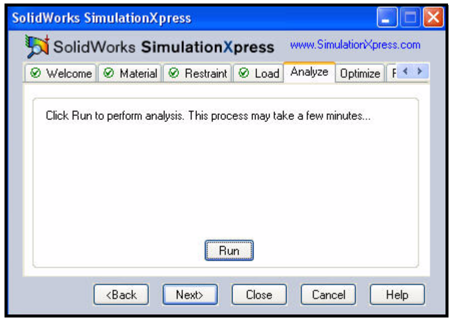
1 Use the default settings.
Check **Yes (recommended).**

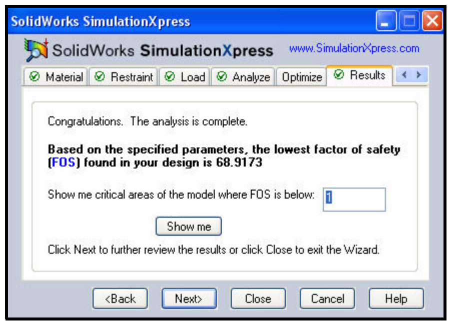
Click **Next>**.



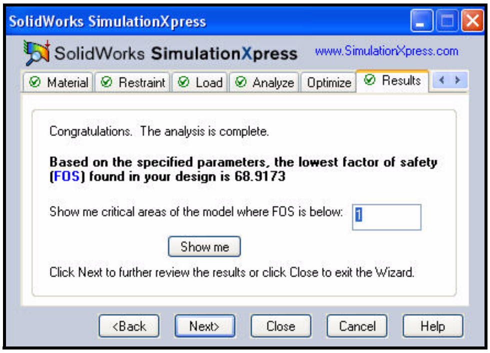
2 Run the analysis.
Click **Run**.

The analysis starts. When the analysis is complete, a check mark is displayed on the Analyze tab and the Results tab.





Viewing Results



Viewing results is an essentialstep in the analysis process. This is the step in which you evaluate how good your design is at withstanding the specified working conditions. This step should lead you to make important decisions about whether to accept the design and move to prototyping, make further improvements on the design, or try additional sets of loads and restraints.

The first screen of the Results tab lists the lowest factor of safety (FOS) of the model under the specified load and restraint.

SimulationXpress uses the maximum von Mises stress criterion to calculate the factors of safety. This criterion states that a ductile material starts to yield when the equivalent stress (von Mises stress) reaches the yield strength of the material. The yield strength (SIGYLD) is defined as a material property. SimulationXpress calculates the factor of safety (FOS) at a point by dividing the yield strength by the equivalent stress at that point.

Interpretation of factor of safety values:

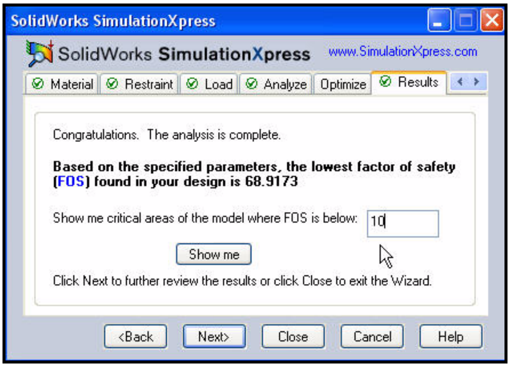
- A factor of safety less than 1.0 at a location indicates that the material at that location has yielded and that the design is not safe.
- A factor of safety of 1.0 at a location indicates that the material at thatlocation has just started to yield.
- A factor of safety greater than 1.0 at a location indicates that the material at that location has not yielded.
- The material at a location will start to yield if you apply new loads equal to the current loads multiplied by the resulting factor of safety.

Viewing the Results

1 View the results.

The factor of safety of the `Axle-A` part is approximately 68.92.

This indicated that the current design is safe or over-designed.



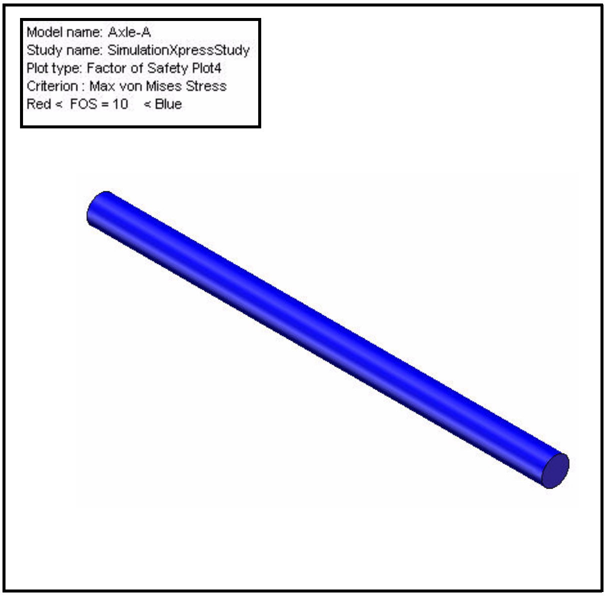
2 Modify the factor of safety.

Enter **10** in the Show me critical areas of the model where FOS is below box.

Click the **Show me** button.

The following plot is displayed. Regions in blue have factors of safety greater than 10 (over-designed regions).

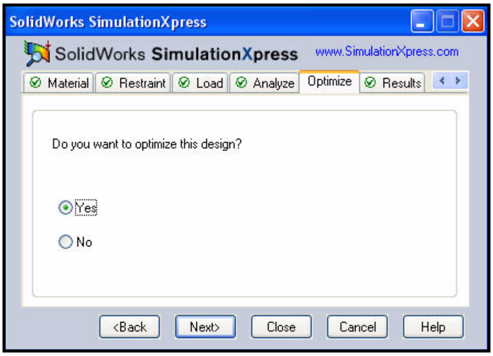
Regions in red have factors of safety less than 10. All areas are displayed in blue.



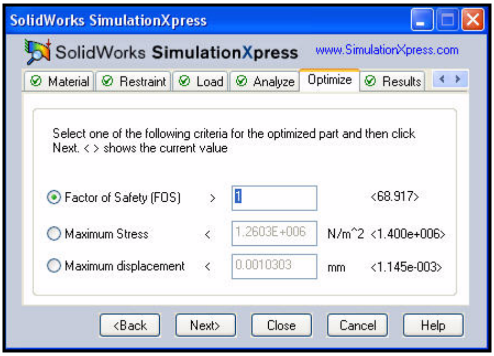
Optimizing the model

The Optimize tab allows you to perform an optimization analysis after completing the stress analysis on the Analyze tab. The software tries to find the optimal value for one model dimension while satisfying a specified criterion:

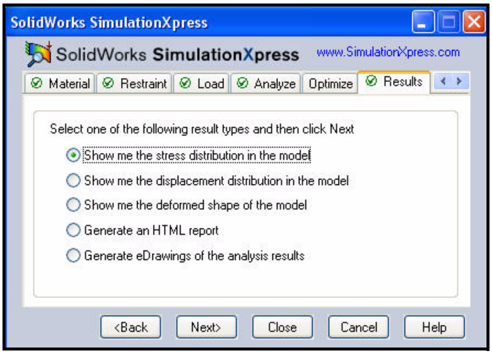
- Factor of Safety
- Maximum Stress
- Maximum displacement



You can either input your desired Factor of Safety or allow SimulationXpress to calculate factor of Safety based on the upper and lower limits. Do not perform the Optimization process at this time.



Stresses

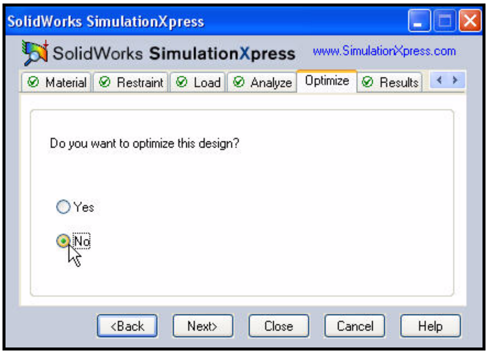


When loads are applied to a body, the body tries to absorb its effects by developing internal forces that, in general, vary from one point to another. The intensity of these internal forces is called stress. The unit of stress is force per unit area.

In SimulationXpress you can view a stress quantity called the equivalent (or von Mises) stress. While the equivalent stress at a point does not uniquely define the state of stress at that point, it provides adequate information to assess the safety of the design for many ductile materials.

The equivalent stress has no direction. It is fully defined by magnitude with stress units (i.e., force/area). SimulationXpress uses the von Mises Yield Criterion to calculate the factors of safety at different points of the model.

Stress Distribution



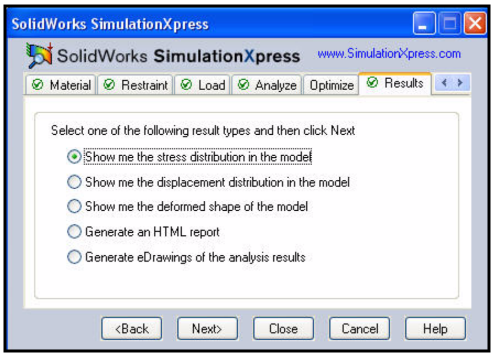
1 **View the Stress distribution of the part.**
Click **Next>**.

Click the **No** box. Do not perform the optimization process at this time.

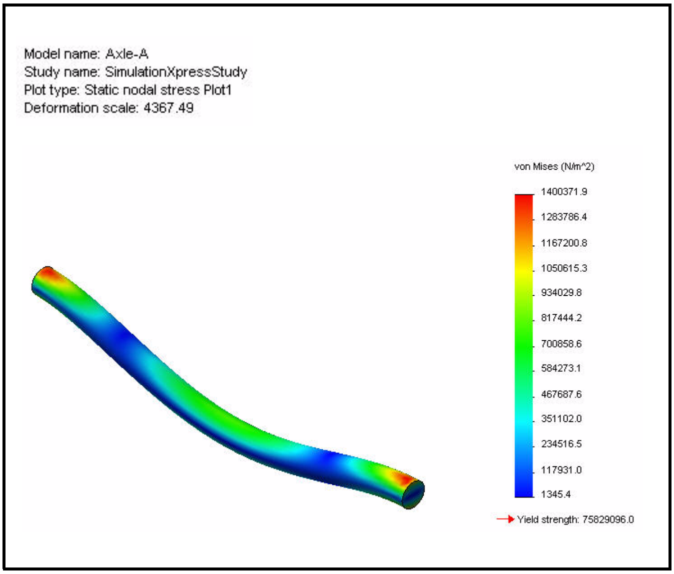
Click **Next>**.

Check the **Show me the stress distribution in the model** box.

Click **Next>**.



2 View the stresses.
The stress plot is displayed in the graphics area.



Animating the Stress Plot

Start the animation.

Click **Play**.

Stop the animation.

Click **Stop**.

Save the animation.

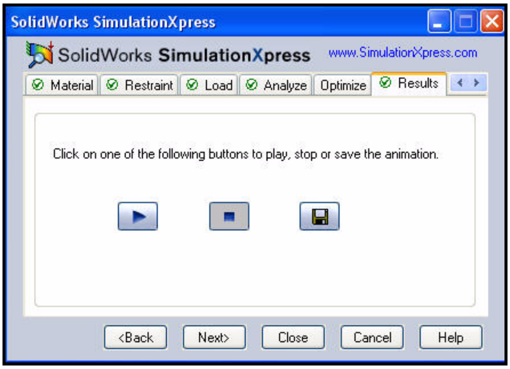
Click **Save**. The Save As dialog box opens.

Click **Save** from the Save As dialog box to accept the file type and name for the animation file.

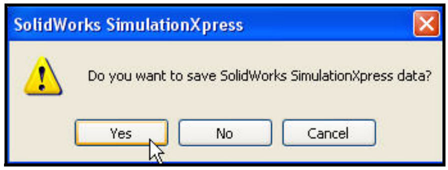
Note: SimulationXpress saves the animation file in the folder specified in the Options dialog box unless you change this folder when you save the animation in the previous step.

Save the Analysis Data and Close SimulationXpress

1 Close SimulationXpress.
Click **Close**.



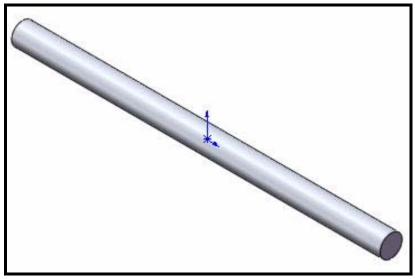
2 Save the analysis data.
Click **Yes to** the message window.



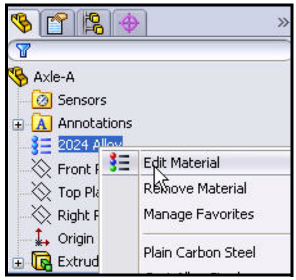
Modify the Axle-A Material

SimulationXpress provides the ability to ask the question, can you reduce the `Axle-A` material thickness without affecting performance or even modify the material?

Modify the material from 2014 Alloy to AISI 304. Rerun the analysis and compare the results.



1 Modify the material in the FeatureManager.
Right-click **2024 Alloy** in the FeatureManager.



Click **Edit Material**. The `Materials` dialog box is displayed.

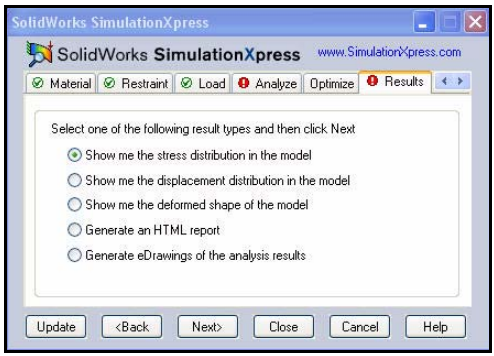
Expand **Steel**.

Click **AISI 304**.

Click **Apply**.

Click **Close**.

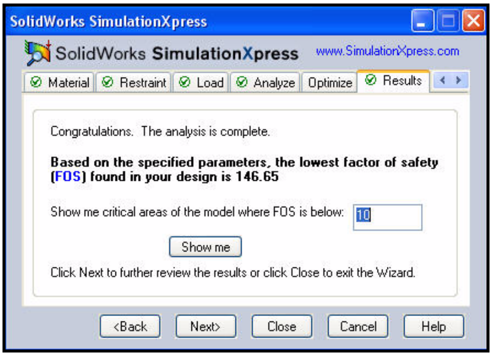
2 Run Simulationpress.
Click **Tools**, **SimulationXpress** from the Menu Bar menu. Exclamation marks are displayed on the Analyze and Results tabs to indicate that you need to reanalyze the model and that existing results do not belong to the current model geometry. An Update button is displayed at the lower left corner of the SimulationXpress window.



3 Update the analysis.
Click the **Update** button.

The analysis starts. When the analysis is complete, the Results tab opens and the Factor of Safety for the modified model is now approximately 146.65.

Click **Next>**.



Run the Optimization process

1 Run Optimization.
Check the **Yes** box.

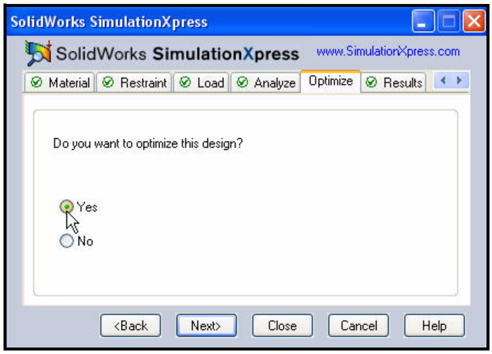
Click **Next>**.

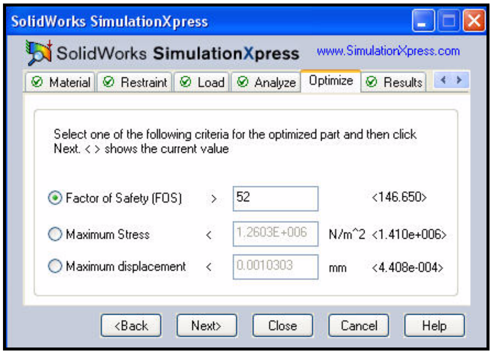
Enter **52** as illustrated.

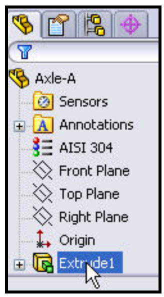
Note: Remember, approximately 68.92 was calculated using 2014 Alloy.

Click **Next>**.

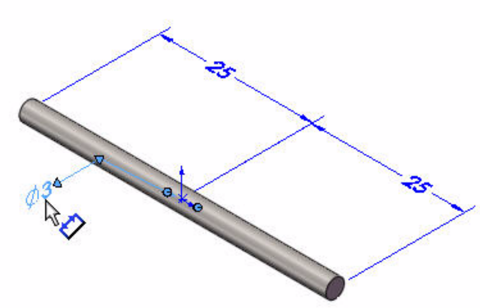
Click `Extrude1` in the FeatureManager. View the dimensions in the graphics area.



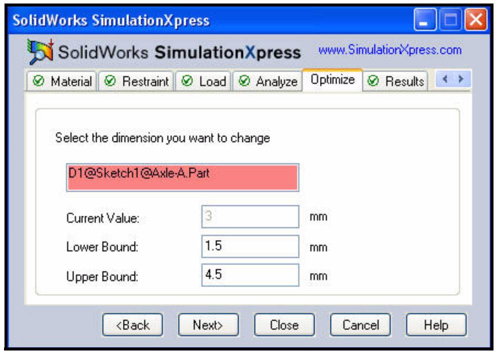




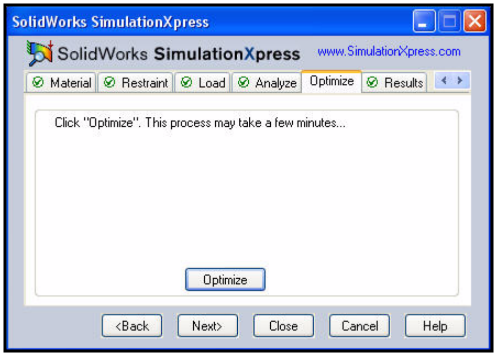
Click the 3 diameter in the graphics area. `D1@Sketch1@Axle-A Part` is displayed.



Click **Next>**.

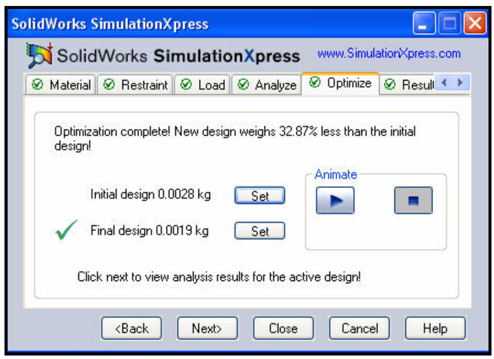


Click the **Optimize** button. This may take a few minutes.



Note: In Lesson 2 the mass of the Axle with 2014 Alloy was .9898 grams. The diameter was 3.00mm.

The Optimized design with the Axle using AISI 304 material has a 2.46mm diameter or 32.87% less material with a Safety Factor of 52.



Click **Close**.

Click **No**.

2 View the new diameter of the model.

Double-click `Extrude1` from the FeatureManager to view the new diameter.

3 Close all models.

Click `Window`, `Close All` from the Menu bar menu. You are finished with this section.

SolidWorks Flow Simulation

During this lesson, you will use SolidWorks Flow Simulation to analyze the aerodynamics of the initial `Race Car Block` assembly and the final `Race Car` assembly. Think of SolidWorks Flow Simulation as a virtual wind tunnel in this section.

Note: The initial `Race Car Block` assembly configuration was created for you to save time and is located in the Flow Simulation folder which you downloaded.

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.

There are several steps to the analysis:

1. Create a design in SolidWorks.
   SolidWorks Flow Simulation can analyze parts, assemblies, sub-assemblies and multi-bodies.
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. View the SolidWorks Flow Simulation results which include:
   Results Plots:
   - Vectors, Contours, and Isolines
   - Cut Plots, Surface, Flow Trajectories, and 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, oxygen, air, etc. 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.

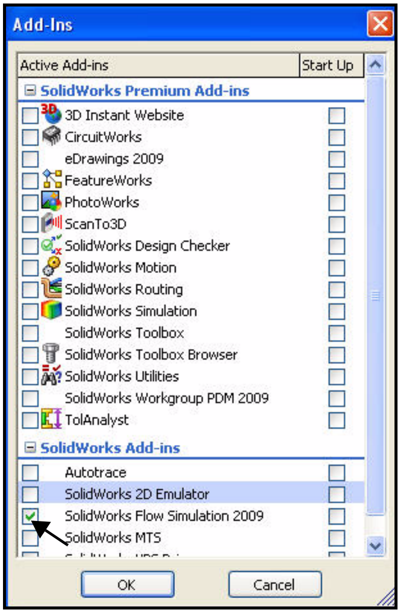
Check Before Using SolidWorks Simulation Flow

Make sure SolidWorks Flow Simulation software is installed.

Click **Tools, Add-ins…** from the Menu bar menu.

Check the **SolidWorks Flow Simulation 2009** box.

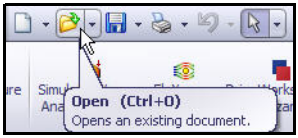
Click **OK** from the `Add-Ins` dialog box.



Note: The Flow Simulation tab is display in the CommandManager with an active document.

Tip: Select tools from the Flow Simulation CommandManager.

Let's Analyze the Initial Race Car Block

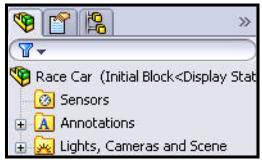


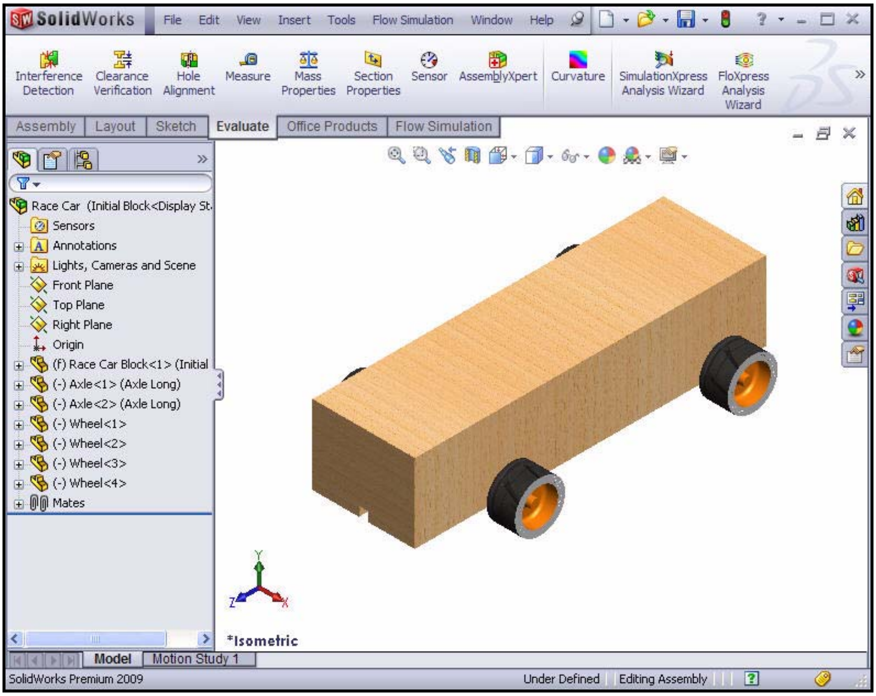
1 Open the Race Car assembly from the Flow Simulation folder.

Click **Open** from the Menu bar toolbar.

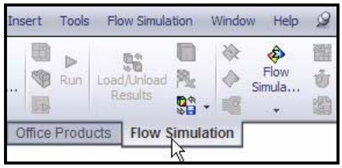
Browse to the `Flow Simulation` folder.

2 Double-click **Race Car.** The `Race Car` assembly (Initial Block) configuration is displayed in the graphics area. The `Race Car` (Initial Block) assembly configuration was created for you to save time.



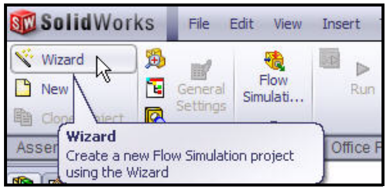


Create a Simulation Flow Project



3 Click the **Flow Simulation** tab from the CommandManager.

Click **Wizard** from the Flow Simulation CommandManager. The `Wizard` dialog box is displayed. View your options.

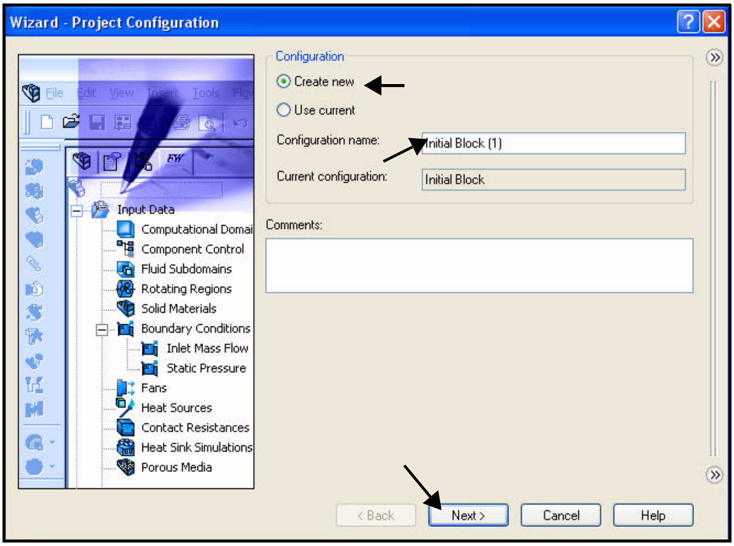


4 **Configuration a project name.**

Click the **Create new** box.

Accept the `Configuration name`: **Initial Block (1).**

Click **Next>**.

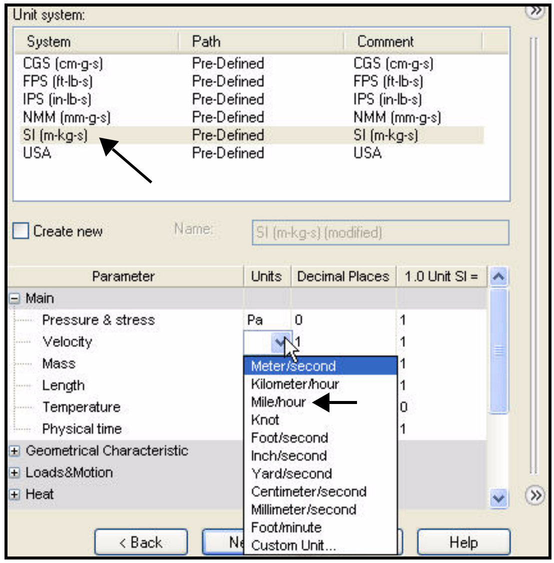


Note: All required analysis data for this project is saved in this SolidWorks model configuration.

5 Set the Unit System.
Click **SI(m-kg-s)** in the `Unit system` box.

Click inside the **Velocity/ Units** box.

Select **Mile/hour**.



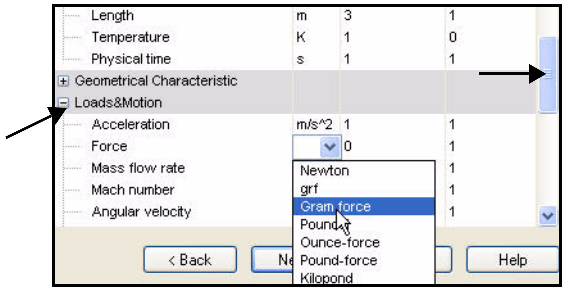
Scroll down to view the **Loads&Motion** option.

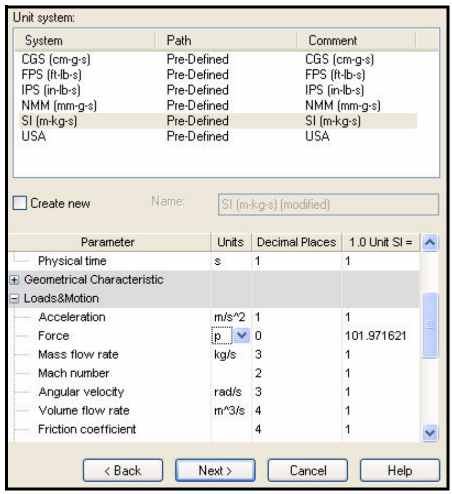
Expand the **Loads&Motion** folder.

Click inside the **Force/Units** box.

Select **Grams force**.

Click **Next>**.





Gram-force

Gram-force is a unit of force, approximately equal to the weight of 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.

6 Set Analysis Type and Physical Features.

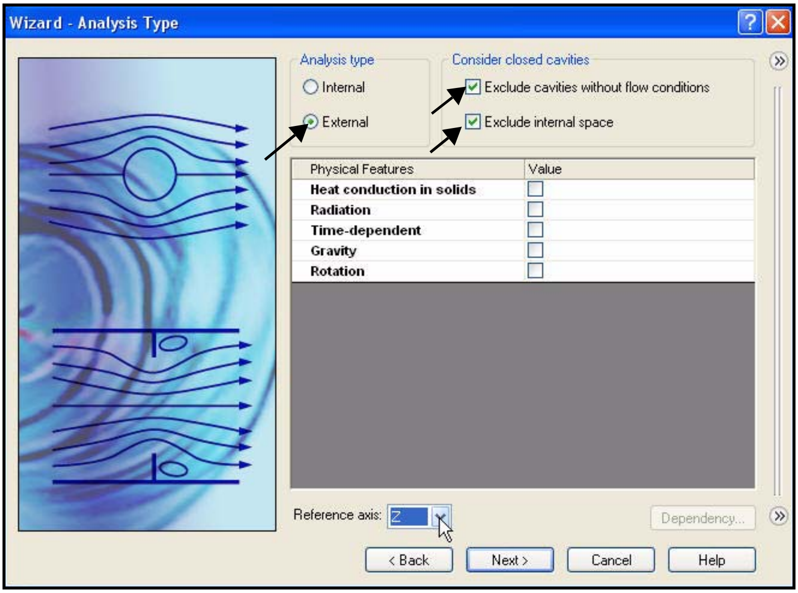
Click **External** as the `Analysis` type.

Check the **Exclude cavities without flow conditions** box.

Check the **Exclude internal space** box.

Select **Z** for `Reference axis`.

Note: The Reference axis is chosen so that an angular velocity vector can be aligned with the Reference axis.



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>**.

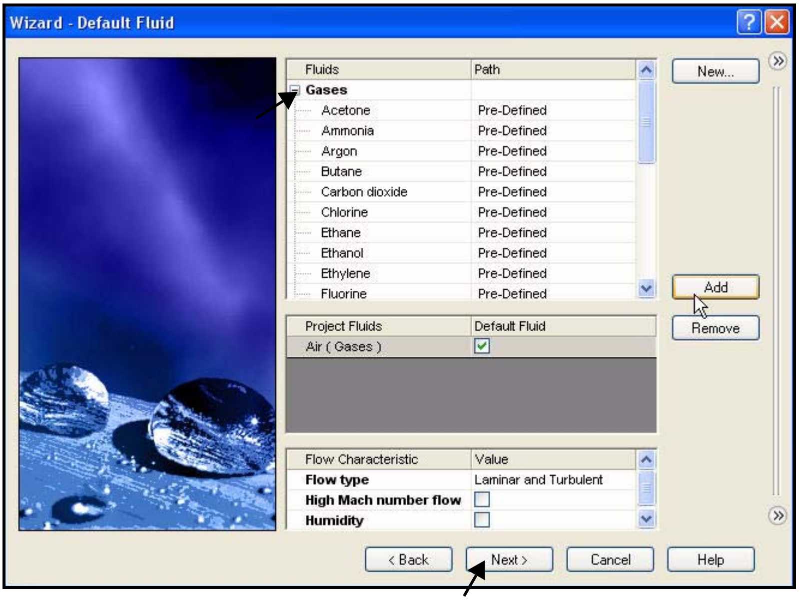
7 Set Default Fluid.

Expand the **Gases** folder.

Click **Air**.

Click the **Add** button.

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



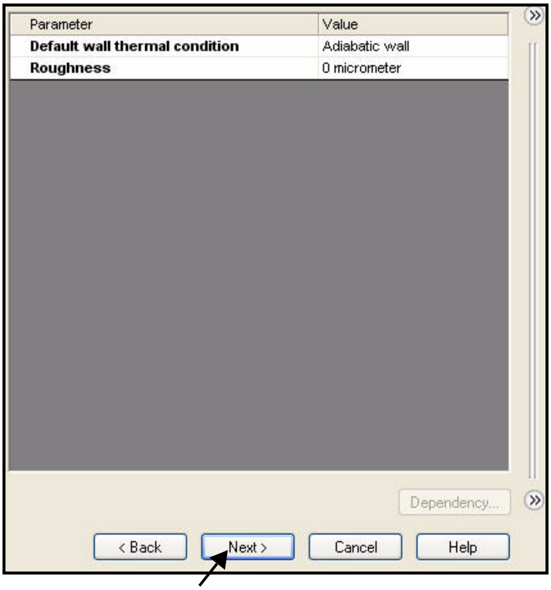
Note: SolidWorks Flow Simulation has a database library of several liquids and gases which is called the Engineering Database. With this database you can create your own materials.

SolidWorks 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>**.

8 Set Wall Conditions.
Accept the default values: **Adiabatic wall** and **Roughness = 0 micrometer**.

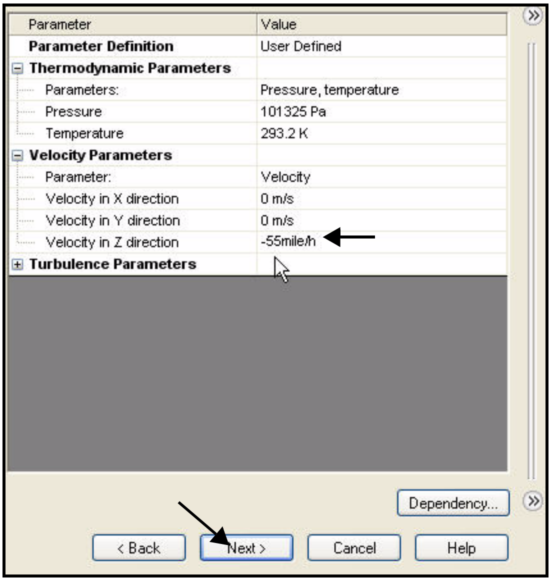
Click **Next>**.



9 Set Initial and Ambient Conditions.
Double-click inside the **Velocity in Z direction** Value box.

Enter **-55 mile/h**. Approximately -24.58 m/s.

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

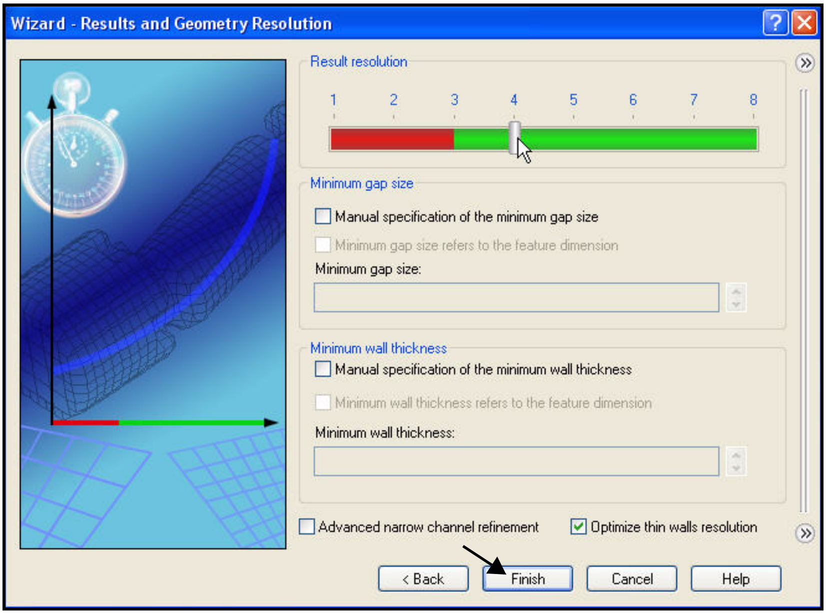


In the real world, the car would be moving through stationary air. In a wind tunnel, the car is stationary and the air is moving. You can think of this Flow Simulation example as a virtual wind tunnel. The car is stationary and the air is moving.

Click **Next>**.

10 Set Results and Geometry Resolution.

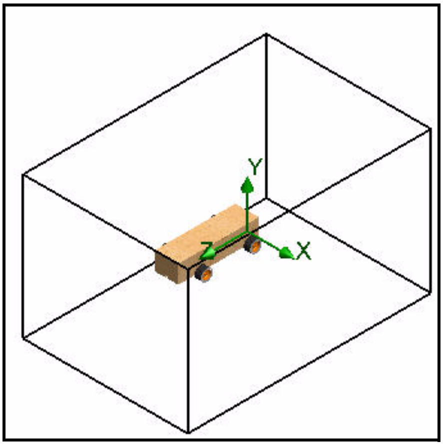
Set the Result resolution to **4.** This will yield acceptably accurate results in a reasonable amount of time.



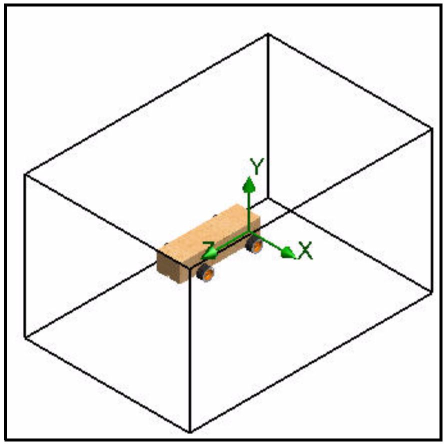
Click the **Finish** button.

11 View the model in the Graphics area.

Zoom out to view the `Computational Domain` in the graphics area.



Computational Domain



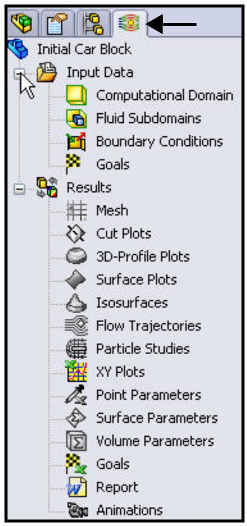
SolidWorks Flow Simulation calculations are performed inside a volume called the Computational Domain. The boundaries of this volume are parallel to the global coordinate system planes. For external flows, the size of the Computational Domain is automatically calculated based on the size of the model.

In the illustration at the right, the black box represents the Computational Domain.

Modifying the Computational Domain

Why modify the Computational Domain:

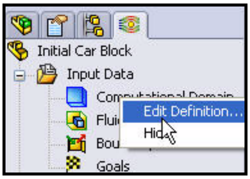
- **Size**
  We are going to reduce the size of the Computational Domain in order to reduce solving time, at the expense of accuracy. A smaller domain means there are fewer fluid cells to calculate. Using the default sizes for the domain could result in solving times in excess of 1.5 hours even on a moderately fast computer. Such solving times are not practical in a school environment.



1 **Display the Flow Simulation analysis tree.**
Click the **Flow Simulation analysis tree** tab.

Expand the **Input Data** folder.

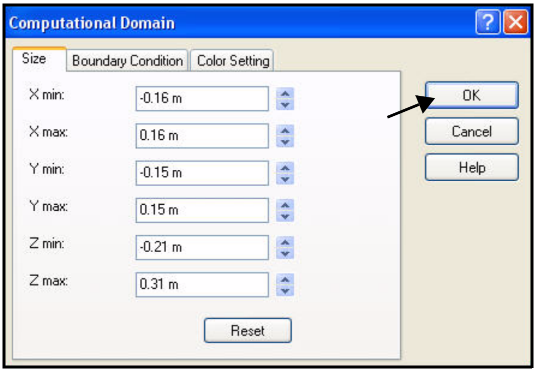
2 Set the Computational Domain size.
Right-click the **Computational Domain** folder.



Click **Edit Definition**.

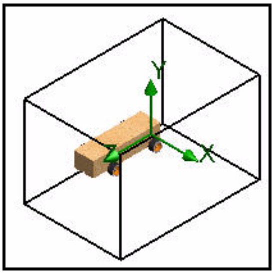
Input the following **values** on the Size tab:

- **X min = - 0.16 m**
- **X max = 0.16 m**
- **Y min = - 0.15 m**
- **Y max = 0.15 m**
- **Z min = - 0.21 m**
- **Z max = 0.31 m**



Click **OK**.

3 Results.
The resulting Computational Domain is displayed in the graphics area.



Setting Goals

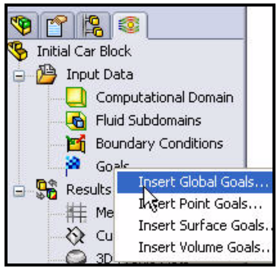
You can specify the following four engineering goals:

- **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 the **Goals** folder.

Click **Insert Global Goals**. The `Global Goals` PropertyManager is displayed.



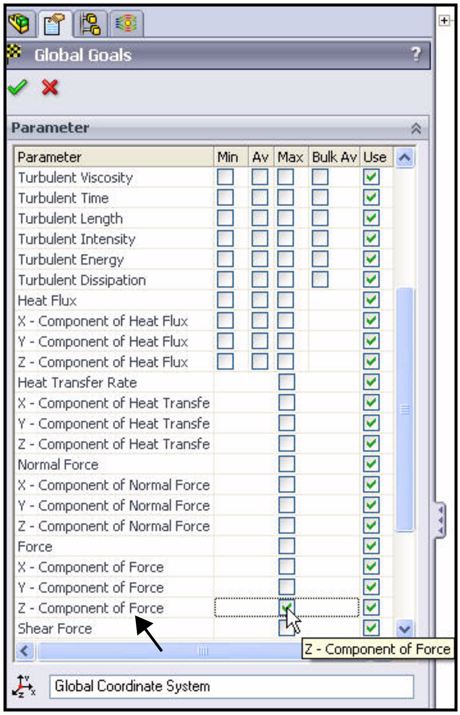
Tip: Drag the boundary of the PropertyManager window to the right to make it wider. This makes it easier to read the parameter names.

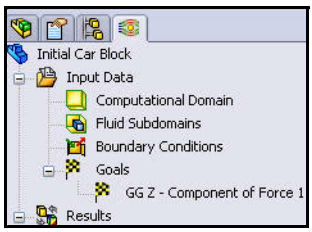
5 Set the goal for drag.

Scroll down to view **Z - Component of Force** under the `Parameters` column.

Check the **Max** (Maximum) box.

Click **OK** from the `Global Goals` PropertyManager. View the update in the FeatureManager.





6 Insert a second Global Goal.

Right-click the **Goals** folder.

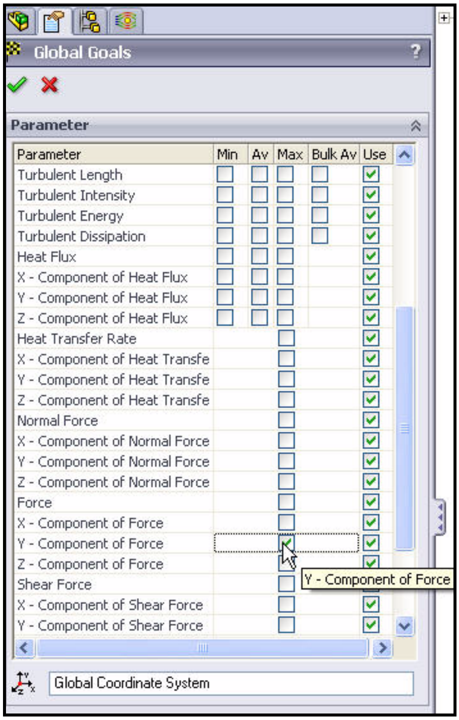
Click **Insert Global Goals** in the Flow Simulation analysis tree.

7 Set the Goal for lift.

Scroll down to view **Y - Component of Force** under the `Parameters` column.

Check the **Max** (Maximum) box.

Click **OK** from the `Global Goals` PropertyManager. View the update in the FeatureManager.

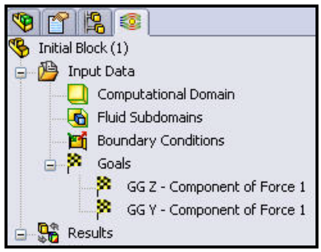


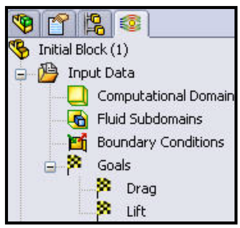
8 Rename the goals.

Two goal icons are displayed in the Flow Simulation analysis tree.

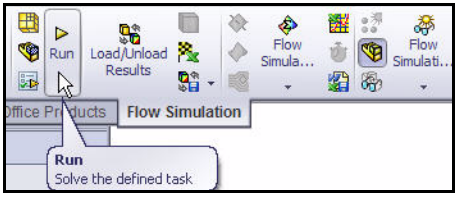
Rename the **GGZ - Component of Force 1** to **Drag**.

Rename the **GGY - Component of Force 1** to **Lift**.



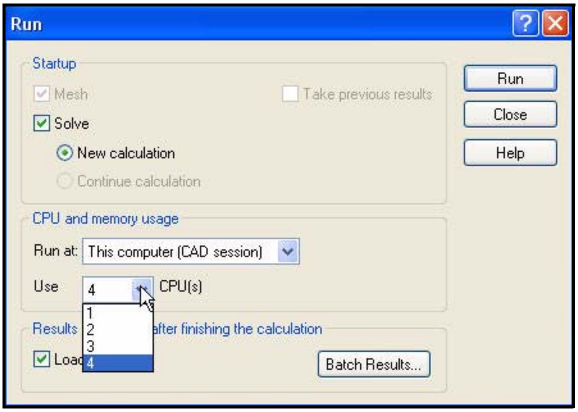


Running the Analysis



1 Run the analysis.
Click **Run** from the Flow Simulation CommandManager. The `Run` dialog box is displayed. View the options.

Click the **Run** button.



2 Solver information.
The `Solver` dialog box is displayed. 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.

Note: The analysis can take up to 25 minutes.

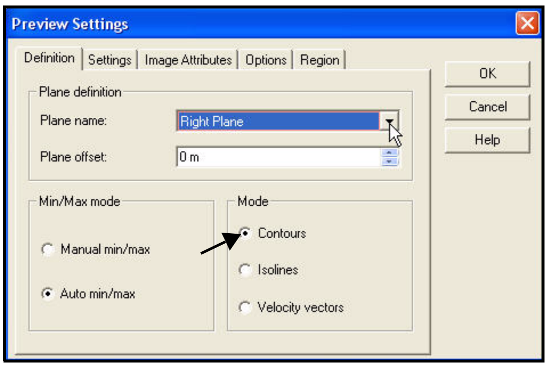
Parameter	Value
Fluid cells	71050
Partial cells	3563
Iterations	60
Last iteration fini...	09:16:39
CPU time per last...	00:00:04
Travels	0.706197
Iterations per 1 t...	84
Cpu time	0 : 3 : 10
Calculation time left	0 : 17 : 47
Status	Calculation

3 Pause the calculation.
After about 60 interrelations, click the **Suspend** button on the `Solver` toolbar. This suspends the calculations so you can explore some of the different types of previews.

4 Preview the Velocity.
Click the **Insert Preview** tool on the `Solver` toolbar. The `Preview Settings` dialog box is displayed.

Select **Right Plane** for `Plane name`.

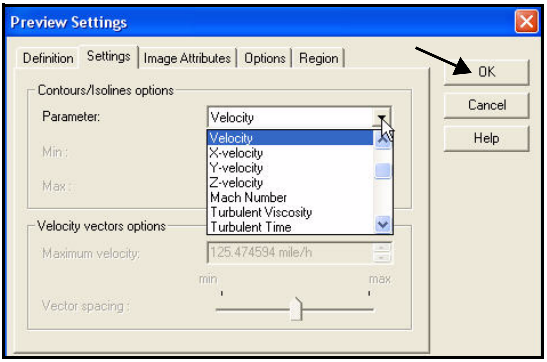
Select **Contours** for `Mode`.



Click the **Settings** tab from the `Preview Settings` dialog box.

Select **Velocity** for `Parameter`. View your options.

Click **OK**.

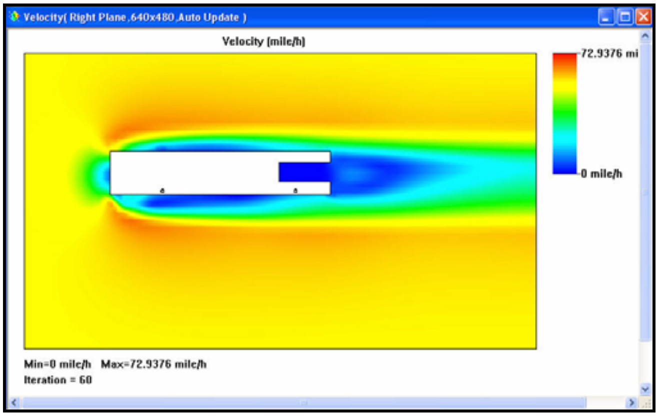


5 View the Preview box.

The Plot preview is displayed in its own window.

View the results.

Close the Preview window.



6 Preview the Pressure.

Click the **Insert Preview** tool on the `Solver` toolbar. The `Preview Settings` dialog box is displayed.

Select **Right Plane** for `Plane name`.

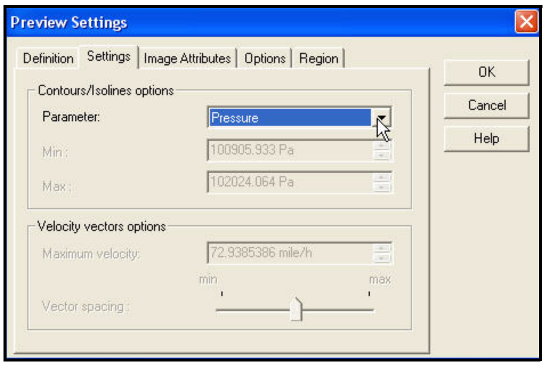
Select **Contours** for `Mode`.

Click the **Settings** tab.

Select **Pressure** for `Parameter`.

Click **OK.**

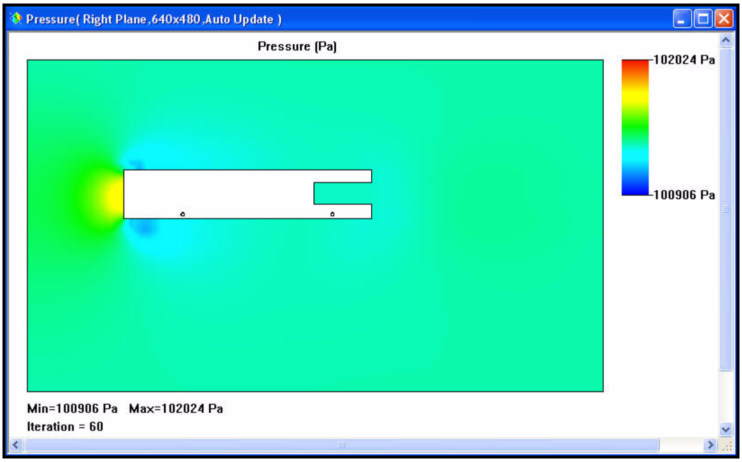
View the results.



7 Resume the Calculation.

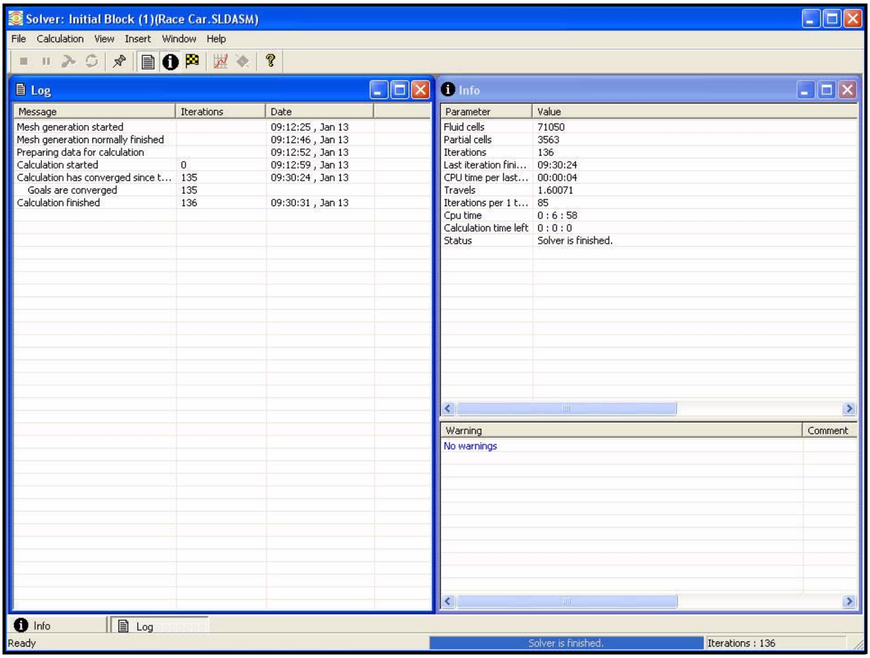
Close the Preview window.

Click the **Suspend** button on the `Solver` toolbar.

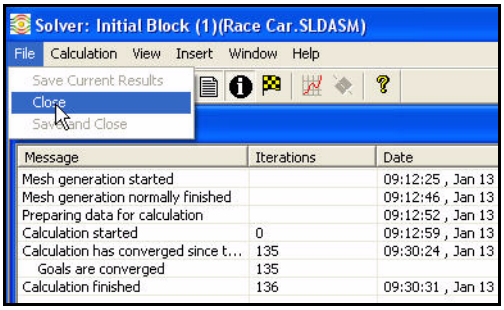


8 Completion.

The status bar at the bottom of the window indicates when the `Solver` is finished.

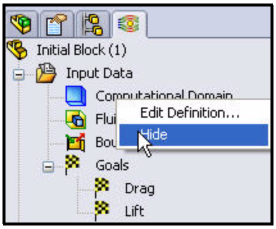


9 Close the Solver window.
Click **File**, **Close** from the `Solver` dialog box.



10 Hide the Computational Domain.
Right-click the **Computational Domain** folder.

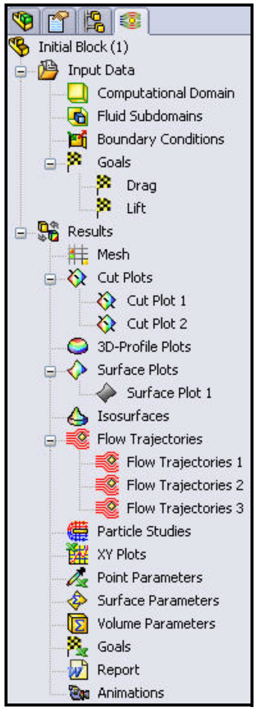
Click **Hide**.



11 Save the document.

Click **Save** from the Menu bar toolbar.

Viewing the Results

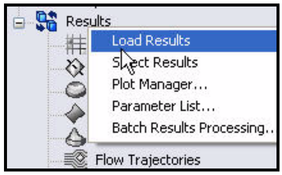


Once the calculation finishes, you can view the saved calculation results through numerous Flow Simulation options in a customized manner directly within the graphics area. The results options are:

- `Cut Plots` (section view of parameter distribution)
- `Section Plots` (generates contours of the results on the specified sections)
- `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 `Section plots`, `Surface plots` and `Flow trajectories` next.

Accessing the Results

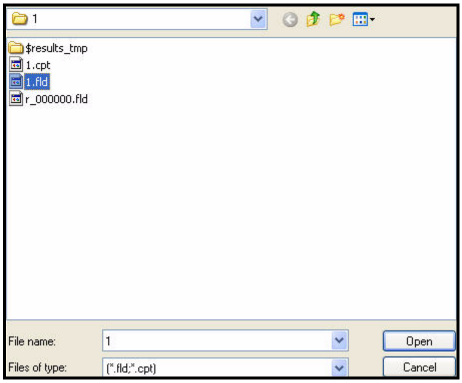


1 **If needed, load the results.**
Right-click the **Results** folder in the Flow Simulation analysis tree.

Click **Load Results**. The `Load Results` dialog box is displayed.

Note: If `Unload Results` is displayed, the results have already been loaded.

Double-click `1.fld`.

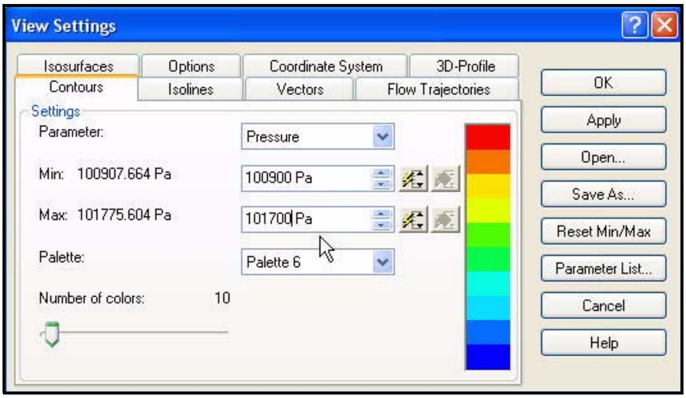


2 **Change the View settings.**
Right-click the **Results** folder.

Click **View Settings**.

Click **Pressure** from the drop-down menu for `Parameter` Setting.

Enter **100900** for `Min`.

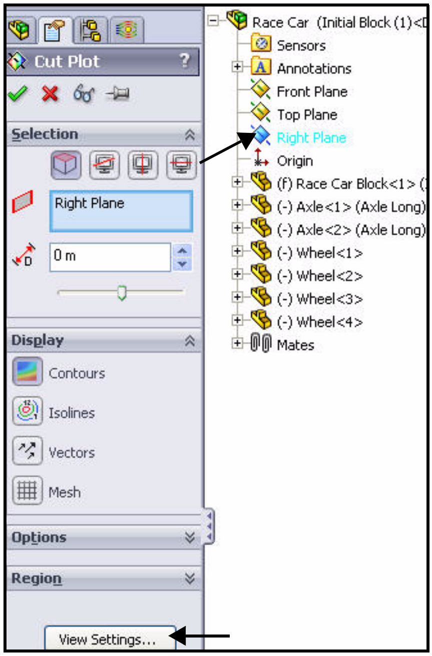


Enter **101700** for `Max`.

Click the **Apply** button.

Click **OK** from the `View Settings` dialog box.

Note: The reason we do not use the default values is because if we make a design change to the car and re-run 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.



3 Create a Section Plot.

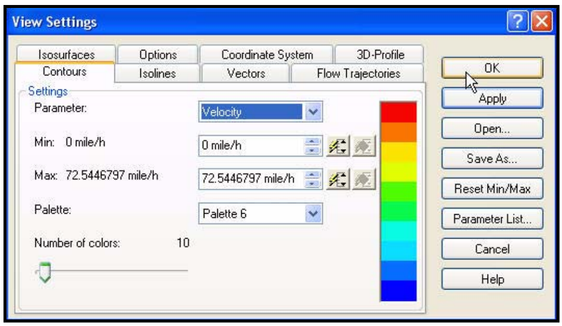
Right-click the **Cut Plots** folder.

Click **Insert**. The `Cut Plot` PropertyManager is displayed. Front Plane is selected by default.

Expand `Race Car` from the fly-out FeatureManager. View the features.

Click `Right Plane` from the fly-out FeatureManager. Right Plane is displayed in the `Selection Plane or Planer Face` box.

Click the **View Settings** button in the `Cut Plot` PropertyManager. The `View Settings` dialog box is displayed.



Click the **Contours** tab.

Click **Velocity** from the drop-down menu for the `Parameter Setting`.

Click the **Apply** button.

Click **OK** from the `View Settings` dialog box.

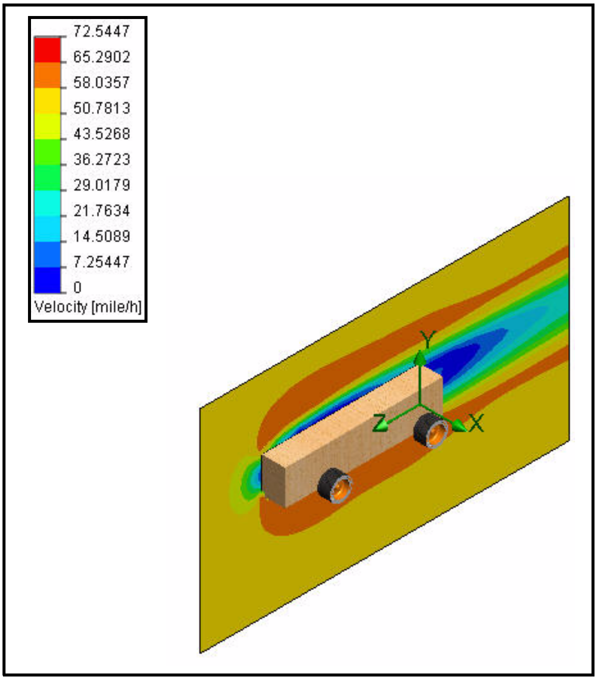
4 View the Section Plot.

Click **OK**  from the `Cut Plot` PropertyManager. View the plot in the graphics area.

Note: You may need to click the **Hide FeatureManager Tree Area** tab to view the total plot.

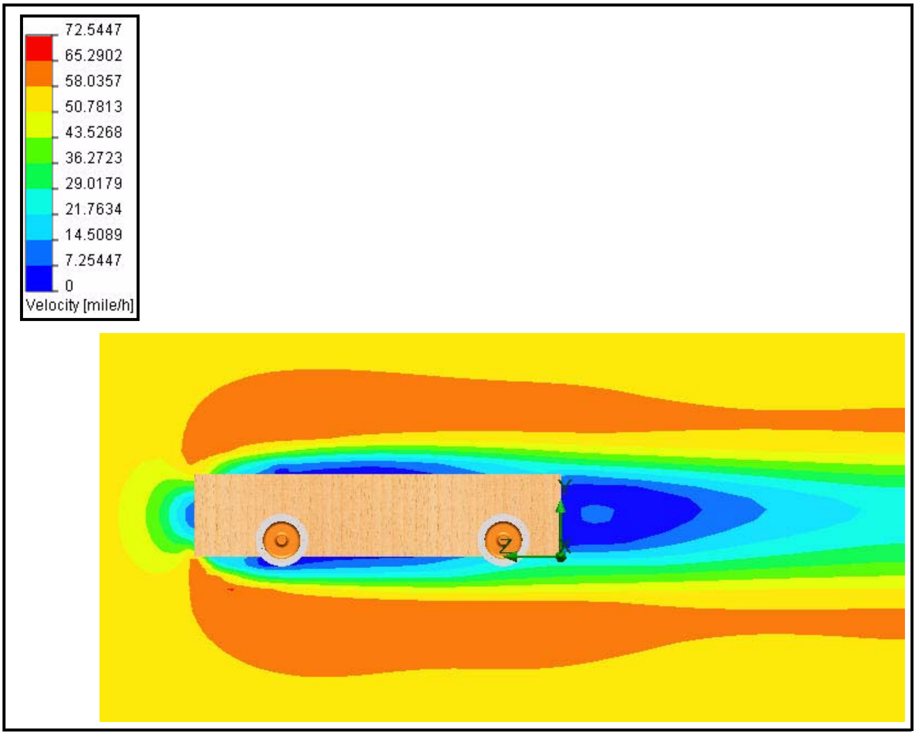
5 View the results.

Click **Right** view from the Heads-up View toolbar. View the results.



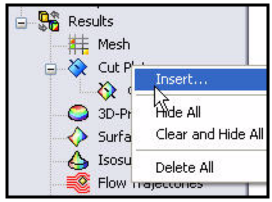
Note: Click the **FeatureManager tree** tab if needed to view the Velocity scale in the graphics area.

Note: View the high velocity areas around the model in red and orange.



6 Create a second Cut Plot.
Right-click the **Cut Plots** folder.

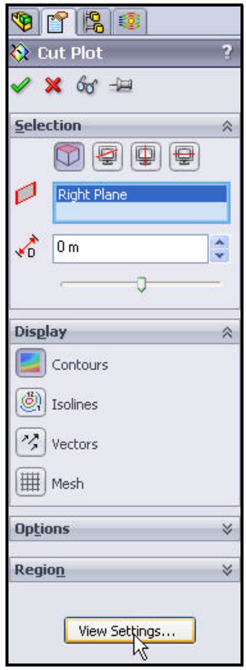
Click **Insert**. Front Plane is selected by default.



7 Change the Selected Plane.
Expand the `Race Car` assembly from the fly-out FeatureManager.

Click `Right Plane` from the fly-out FeatureManager. Right Plane is displayed in the `Selection Plane or Planer Face` box.

Click the **View Settings** button.



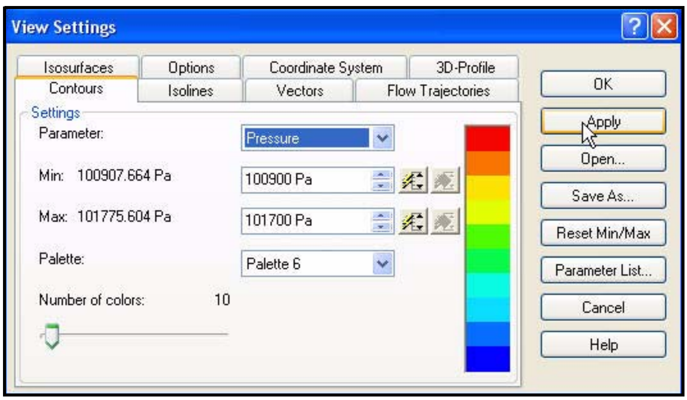
Click the **Contours** tab.

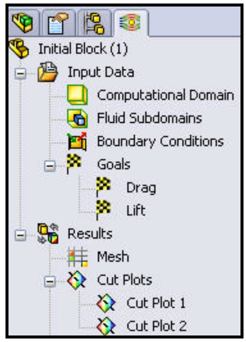
8 View the Settings.
Click **Pressure** from the drop-down menu for `Parameter Setting`.

View the Min **100900** value.

View the Max **101700** value.

Click the **Apply** button.





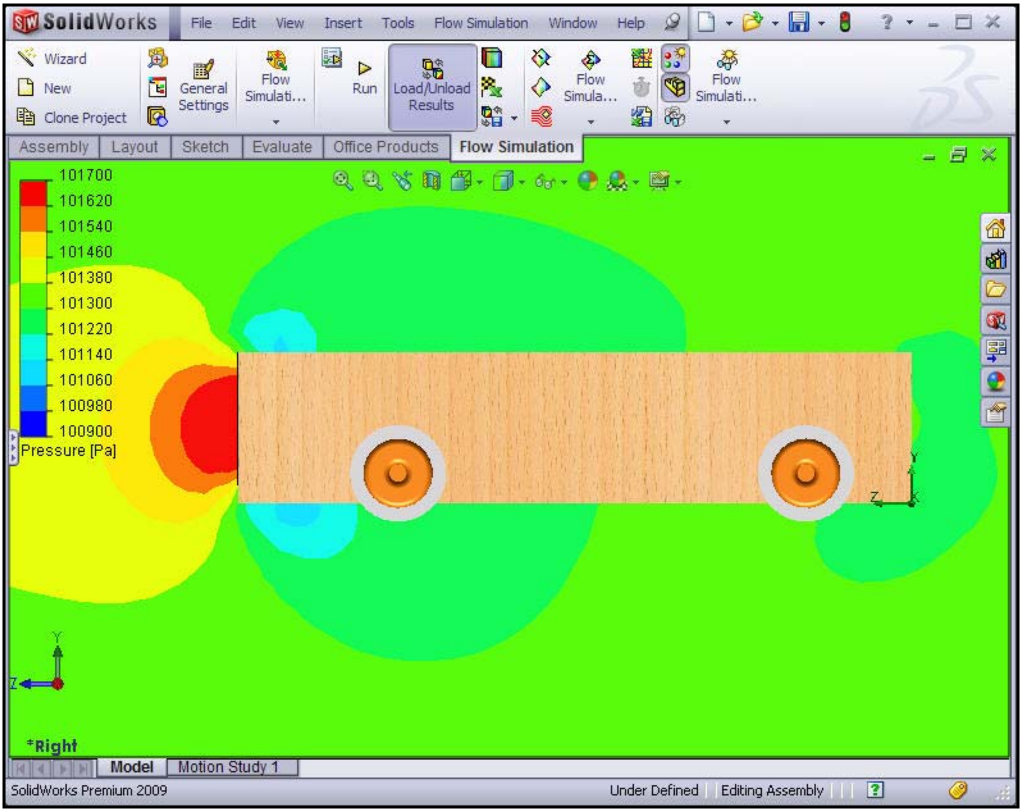
Click **OK** from the `View Settings` dialog box.

Click **OK** from the `Cut Plot` PropertyManager. `Cut Plot 2` is displayed in the Flow Simulation analysis tree.

Note: If needed, click the **FeatureManager tree** tab as illustrated to view the full Graphics area.

9 View the second Plot.

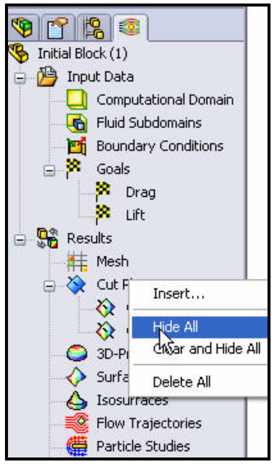
Click **Right** view from the Heads-up View toolbar. View the plot.



10 Hide the Section Plots.

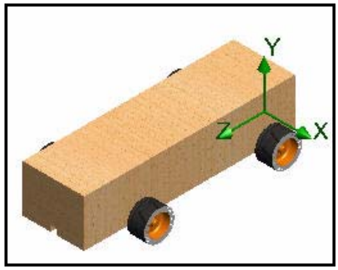
Right-click the `Cut Plots` folder.

Click **Hide All**. View the model in the graphics area.



11 Display an Isometric view.

Click **Isometric** from the Hands-on View toolbar.



12 Save the document.

Click **Save** from the Menu bar toolbar.

13 Insert a Surface Plot.

Right-click `Surface Plots` in the Flow Simulation analysis tree.

Click **Insert**.

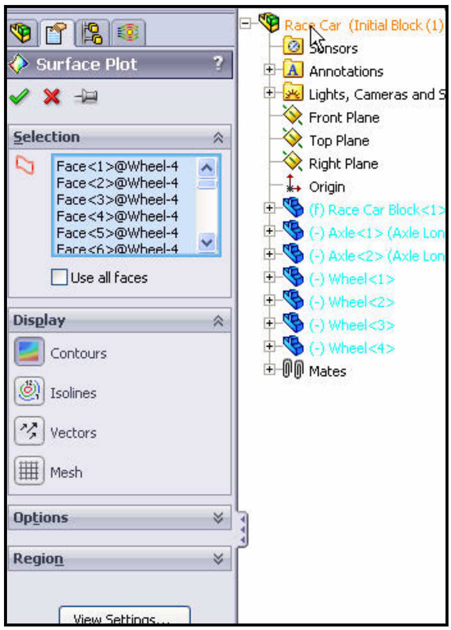
Un-check the **Use all faces** box.

Click `Race Car` from the fly-out FeatureManager. View the selected surfaces. Note: This may take 1 - 2 minutes.

Click the **Contours** box.

Click **OK** from the `Surface Plot` PropertyManager.

A status box is displayed.

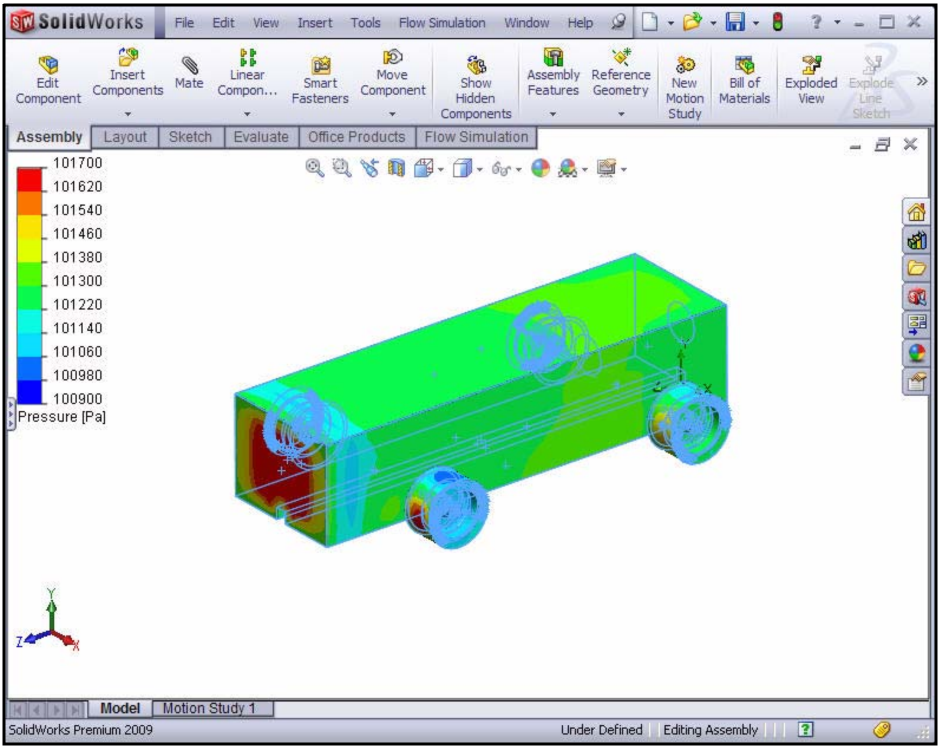


14 Surface plot results.

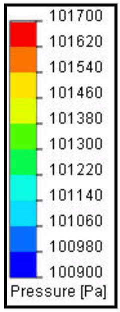
The **Surface Plot** displays the pressure distribution on the selected model faces or SolidWorks surfaces.

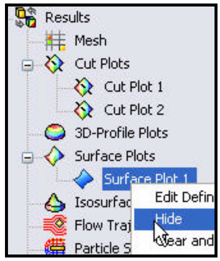
Rotate the model in the graphics area to view the surfaces and their color.

Note: The drag force is equal to the pressure multiplied by the area. You can see in the Surface Plot 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 `Race Car`. Adding a front wing would reduce the high pressure in front of the Wheels and provides a down force for the `Race Car`.



Interpreting the Results





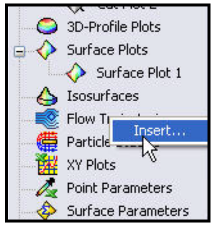
Red indicates areas of high pressure. Blue indicates areas of low pressure. By looking at the Surface plot we can see that the pressure is highest on the front of the Initial `Car Block` and the front surface of the front wheels.

15 Hide the Surface plot.
Right-click `Surface Plot 1`.

Click **Hide**. This hides the Surface plot so we can more easily see the flow trajectories.

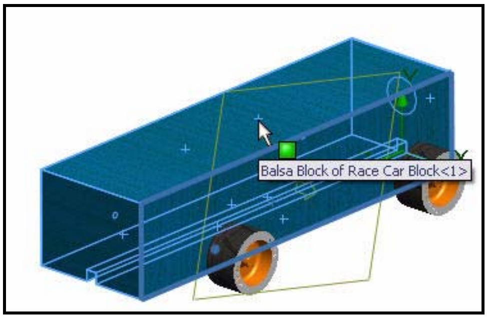
Note: Click **Show** to display the Surface plot.

Flow Trajectories



Flow trajectories are displayed as flow streamlines. Flow streamlines are curves where the flow velocity vector is tangent to that curve at any point on the curve.

Tip: They are analogous to the streamers of smoke in a wind tunnel.



1 Inserting a flow trajectory.
Right-click `Flow Trajectories` in the Flow Simulation analysis tree.

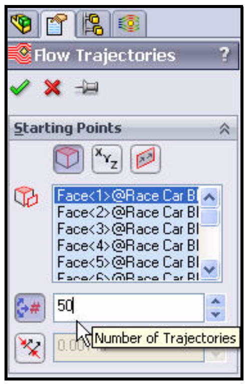
Click **Insert**. The Reference options is active.

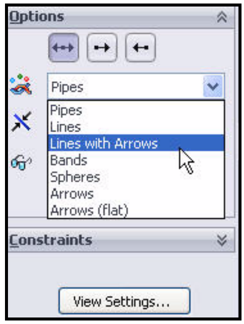
Right-click **Clear Selections** in the `Selection` box.

Click the ten flat surfaces of the **Race Car Block**.

Click the face of the four **Wheels**.

Enter **50** for the `Number of trajectories`.





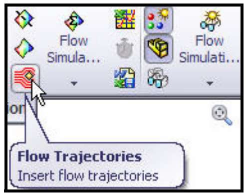
Select **Line with Arrow** from the `Draw trajectories` drop-down menu.

Click **OK** from the `Flow Trajectories` PropertyManager.

2 View the flow trajectory.

This type of display helps visualize how the air flows around the car.

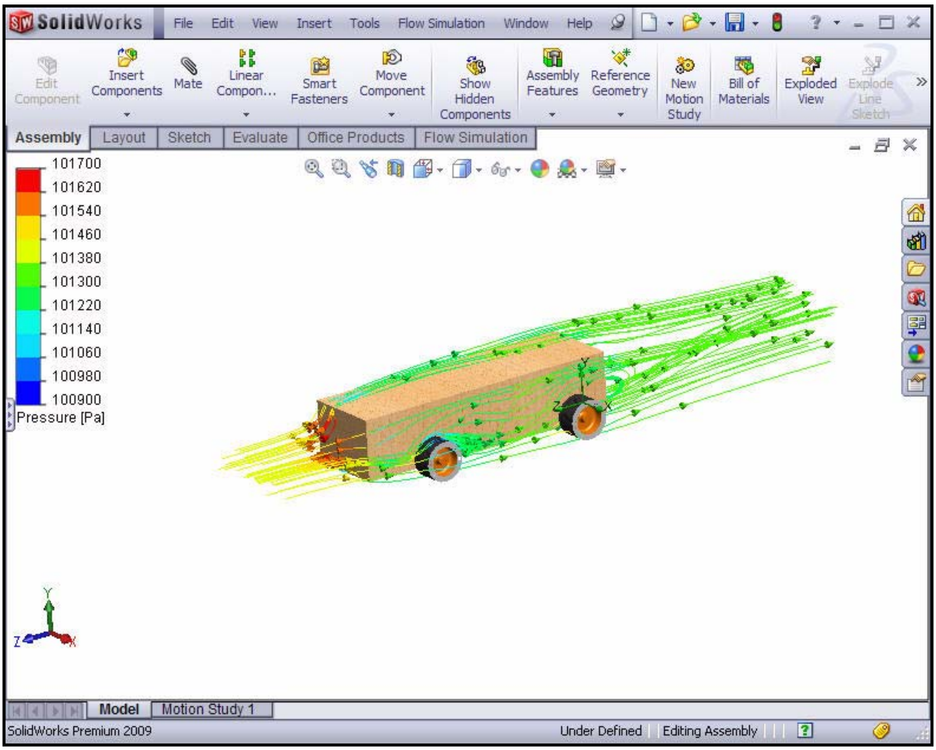
Rotate the model in the graphics area to view the turbulence around the front wheels and behind the block.



Note: Click the **Flow Trajectories** tool in the Simulation toolbar to insert a new trajectory.

3 Save the document.

Click **Save** from the Menu bar toolbar.

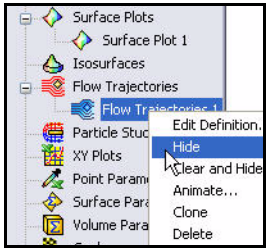


Experiment With Other Flow Trajectories

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.



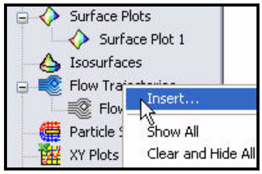
We will create some other flow trajectories.

4 Hide the flow trajectory.

Right-click `Flow Trajectories 1`.

Click **Hide**.

5 Insert a new Flow Trajectory plot.



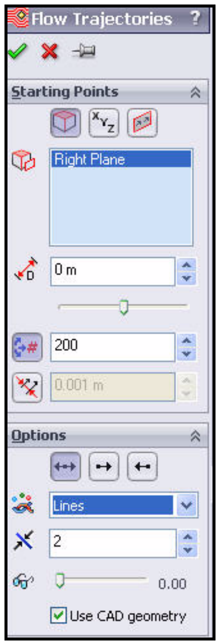
Right-click the `Flow Trajectories` folder.

Click **Insert**.

Right-click **Clear Selections**.

Click `Right Plane` from the fly-out FeatureManager.

Enter **200** for the `Number of trajectories`.

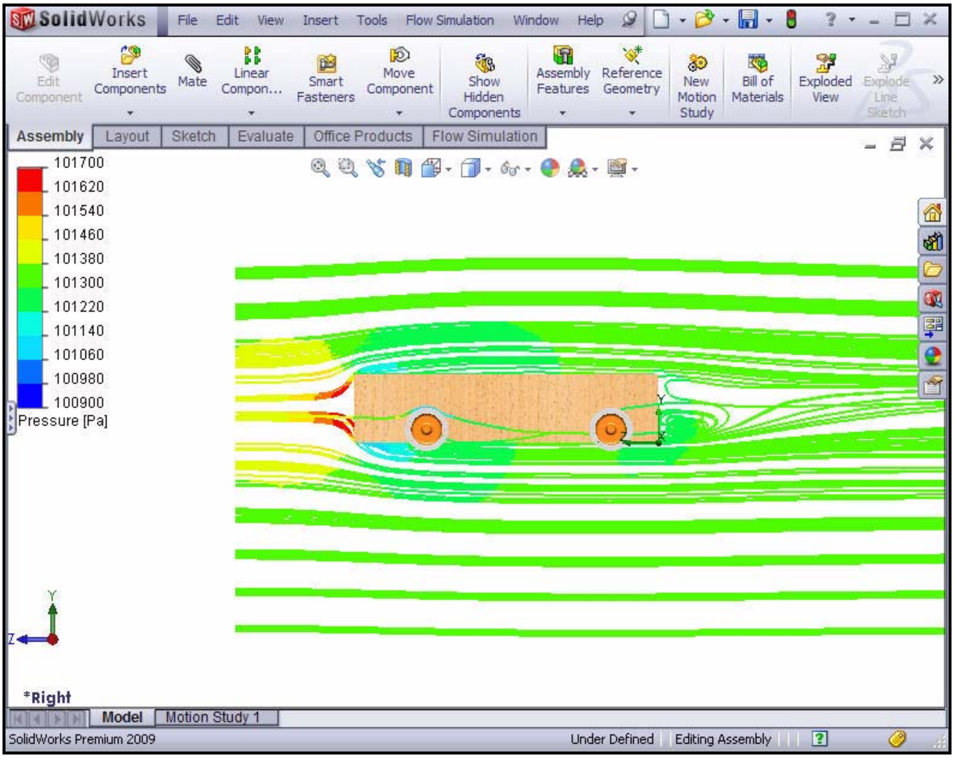


Select **Lines** from the `Draw trajectories` drop-down menu.

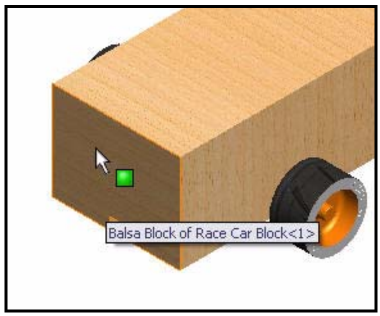
Click **OK**  from the `Flow Trajectories` PropertyManager.

6 Display the Right view.

Click **Right**  view from the Heads-up View toolbar.



Note: Notice the turbulence in front and behind the body of the block.



1 Insert another new Flow Trajectory plot.
Right-click `Flow Trajectories 2`.

Click **Hide**.

Right-click the `Flow Trajectories` folder.

Click **Insert**.

Right-click **Clear Selections.**

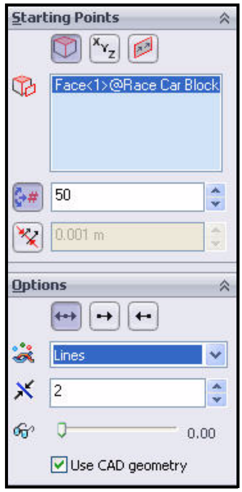
Click **Isometric** view from the Heads-up View toolbar.

Click the **front face** of the `Race Car`.

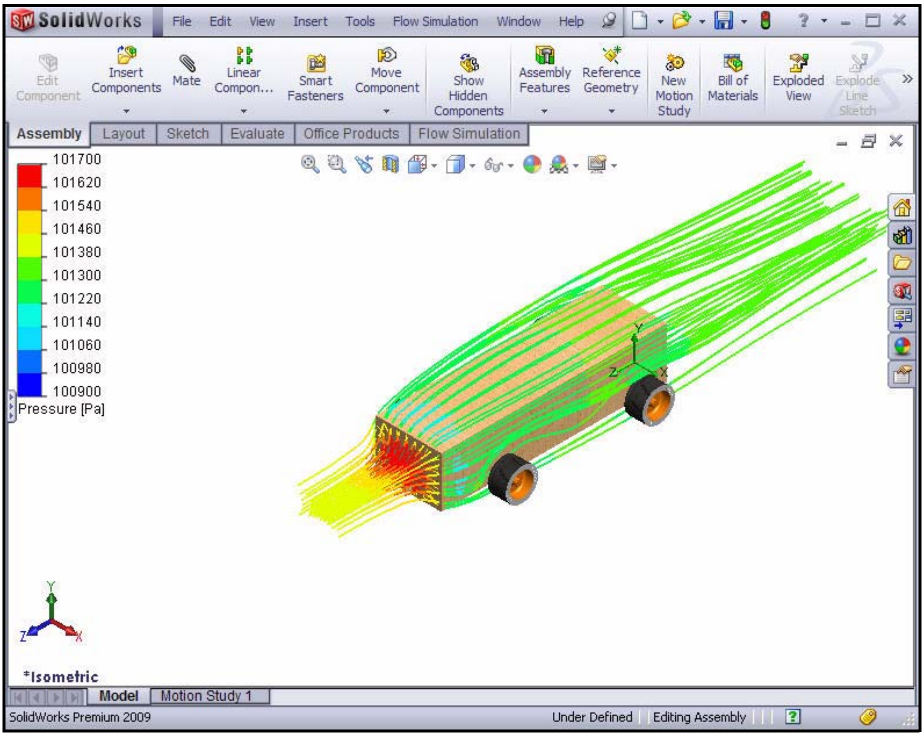
Enter **50** for the `Number of trajectories`.

Select **Lines** from the `Draw trajectories` drop-down menu.

Click **OK** from the `Flow Trajectories` PropertyManager.



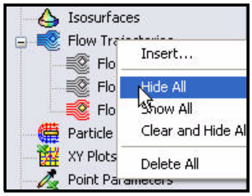
Tip: The lower number of trajectory lines makes it easier to see if there is significant turbulence surrounding the model.



The flow trajectories reveal several conditions:

- The red color of the trajectories on the front body of the `Race Car` assembly indicates an area of high pressure. This pressure will effect the speed of the `Race Car`.
- The flow trajectories behind the wheels are fairly smooth indicating a lack of turbulence.

2 Hide all Flow Trajectories.



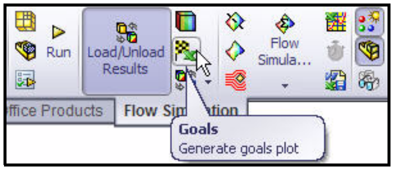
Right-click the `Flow Trajectories` folder.

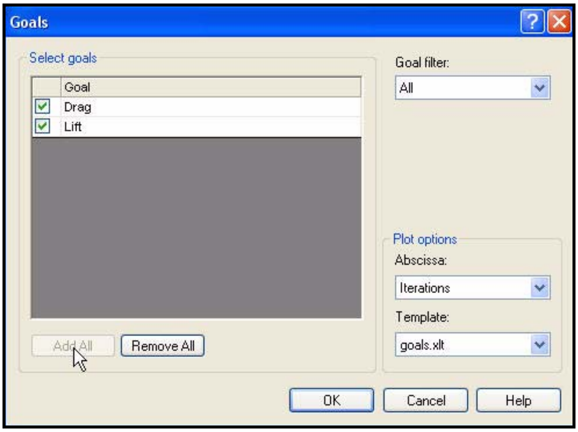
Click **Hide All**.

3 Save the document.

Click **Save** from the Menu bar toolbar.

Quantitative Results





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

Note: Microsoft® Excel is needed for the next section.

1 Create a Goals plot.

Click the **Goals** tool from the `Flow Simulation` tab. The `Goals` dialog box is displayed.

Click the **Add All** button.

Click **OK**.

2 Excel spreadsheet.

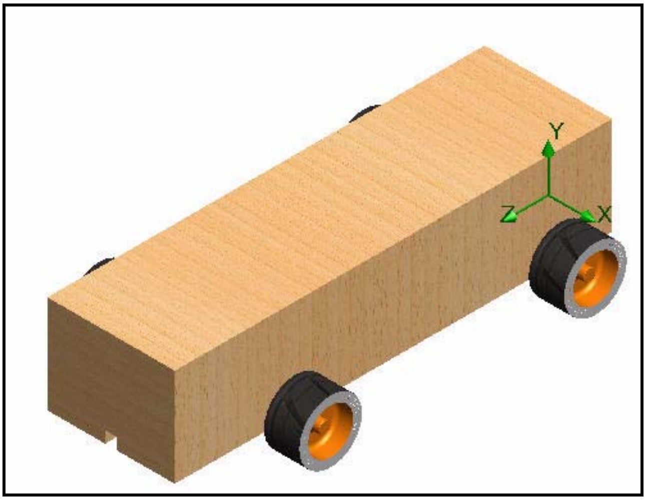
Microsoft® Excel is launched and a spreadsheet opens. Pay particular attention to the first three columns. They show the name of the goal, the units (gram-force, in this case) and the value.

goals1

Race Car.SLDASM [Initial Block (1)]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Valu
Drag	[p]	-150.3421045	-150.193293	-150.3421045	-150.073694
Lift	[p]	10.01262452	9.823998331	9.207361991	10.0916489

Iterations: 137
Analysis interval: 43



Note: Numbers may vary slightly.

3 Save and close the assembly.
Click **File**, **Save**. Accept the default name.

Click **Save**.

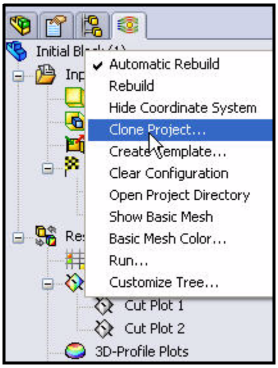
Close the **Excel** spreadsheet.

Units, Values, and Interpreting the Results

As we discussed, 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 -150.34 grams.

The correct way to state the results is to say we have a drag force of approximately 150.34 grams-force and a downward lift force of approximately 10.01 grams-force.

Changing the Design



Based on the analysis of the `Race Car` (Initial Block) assembly configuration using SolidWorks Flow Simulation, we conclude that the shape of the body can be greatly improved.

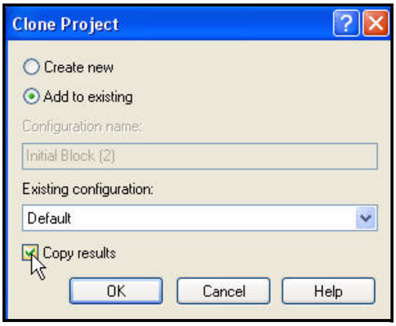
The easiest way to redo an analysis is to clone the SolidWorks Flow Simulation project we created for the Initial Block design. This way we don't have to repeat the work of adding the goals and defining the Computational Domain, but we can not reuse plots were new features were created on the final Default `Race Car` configuration.

To save time, the Final Default configuration for this section is provided. 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.

Tip: An configuration may be changed to dimension of the different value. Both parts and assemblies can support configuration adjustments.

Note: Some of the Referenced faces of the car body don't exist in the Final Default configuration. They were eliminated whenthe cut features andfillets were applied to the body. Therefore, we must redefine the reference before we can display any plots. The Axle part was also modified in the Initial Block configuration to fix the assembly.

4 **Clone the Project.**



Right-click the `Initial Block (1)` configuration in the Flow Simulation analysis tree.

Click **Clone Project**.

Click **Add to existing**.

Select **Default** for Existing configuration.

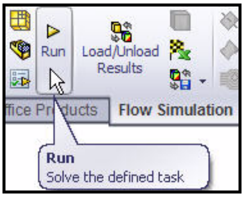
Check the **Copy results** box.

Click **OK**. The system will ask you if you want to reset the Computational Domain.

Click **No**.

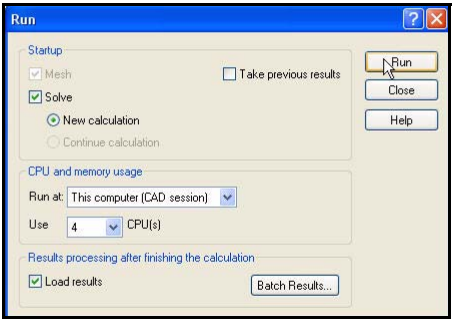
Note: To make it easier to do meaningful comparisons between the two sets of results, we want to use the same size Computational Domain. Also, resetting the domain would require us to redefine the symmetry conditions. That would be extra work.

5 Reset Mesh settings.
Do you want to reset mesh settings? Click **Yes**.



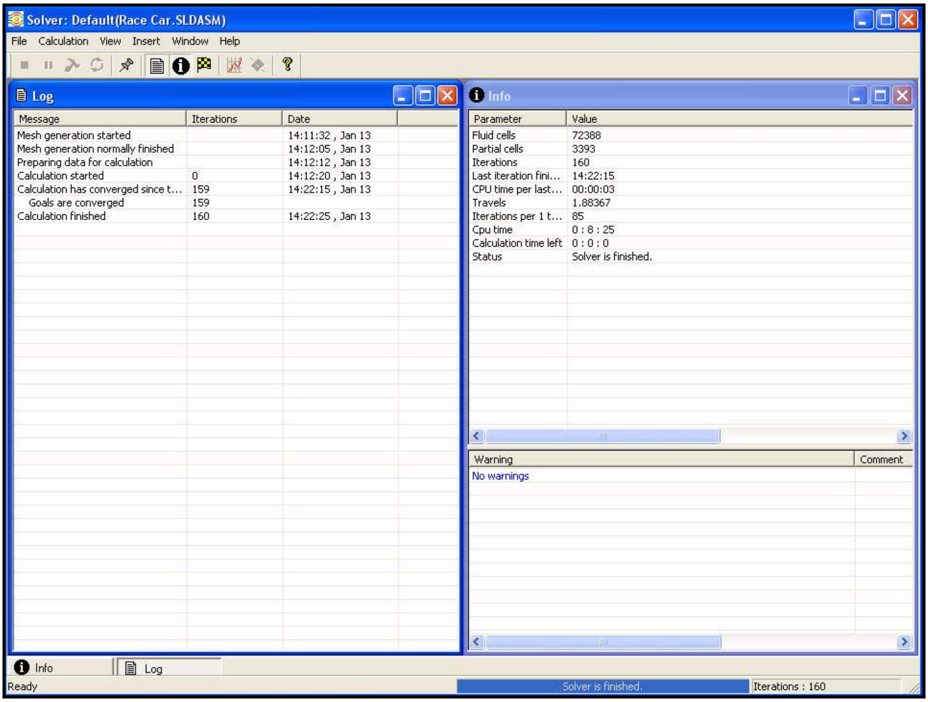
6 Run the solver.
Click **Run** ▷ from the Flow Simulation CommandManager toolbar.

Click **Run** from the `Run` dialog box. This can take 10 - 15 minutes.



7 Completion.
The status bar at the bottom of the window indicates when the `Solver` is finished.

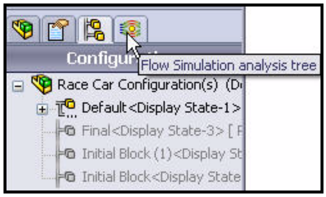
Close the `Solver` dialog box.

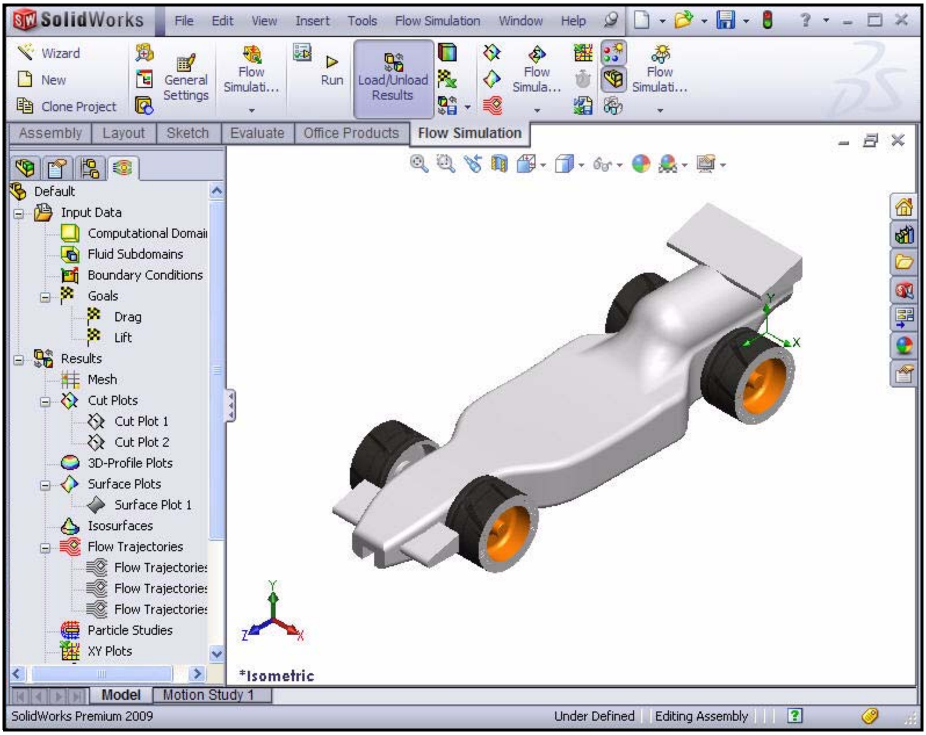


Examine the Results

1 Load the results.

Click the **Flow Simulation analysis tree** tab. Examine the results for the Default configuration. The Default configuration is the final configuration of the `Race Car` assembly.

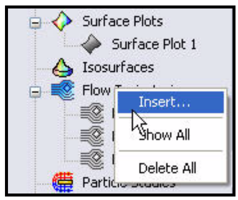




1 Create a Flow Trajectory plot.

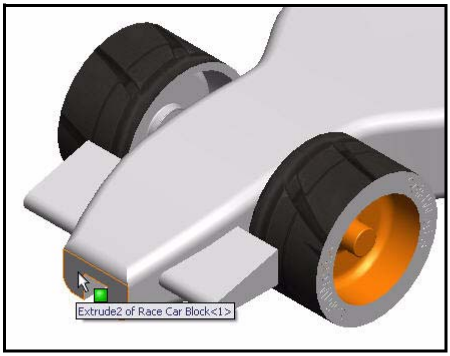
Right-click the `Flow Trajectories` **folder.**

Click **Insert**.



Click **Isometric** view from the Heads-up View toolbar.

Click the **front face** of the `Race Car`.

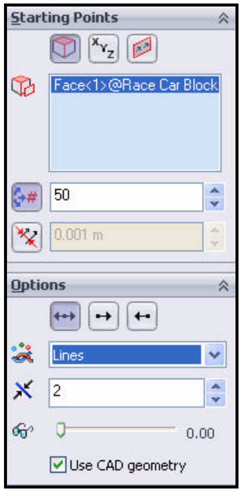


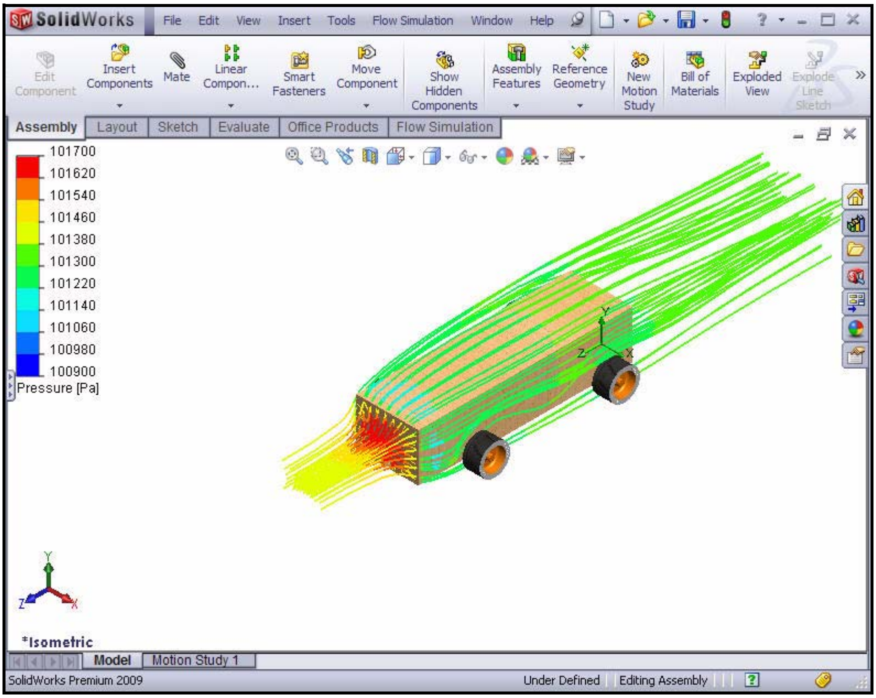
Enter **50** for the `Number of trajectories`.

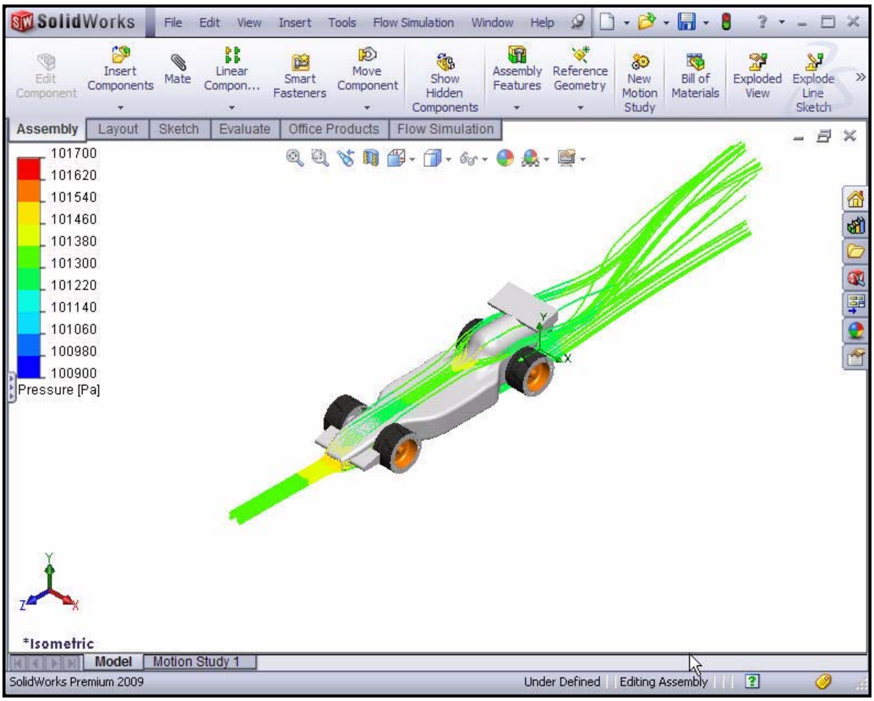
Select **Lines** from the `Draw trajectories` drop-down menu.

Click **OK** from the `Flow Trajectories` PropertyManager.

Below are the two Flow Trajectory plots `Race Car` (Initial Block) vs. the final Default `Race Car` configuration. View the pressure areas.




SolidWorks
File Edit View Insert Tools Flow Simulation Window Help
Edit Component
Insert Components
Mate
Linear Compon...
Smart Fasteners
Move Component
Show Hidden Components
Assembly Features
Reference Geometry
New Motion Study
Bill of Materials
Exploded View
Explode Line Sketch
Assembly
Layout
Sketch
Evaluate
Office Products
Flow Simulation
101700
101620
101540
101460
101380
101300
101220
101140
101060
100980
100900
Pressure [Pa]
*Isometric
Model
Motion Study 1
SolidWorks Premium 2009
Under Defined
Editing Assembly


SolidWorks
File Edit View Insert Tools Flow Simulation Window Help
Edit Component
Insert Components
Mate
Linear Compon...
Smart Fasteners
Move Component
Show Hidden Components
Assembly Features
Reference Geometry
New Motion Study
Bill of Materials
Exploded View
Explode Line Sketch
Assembly
Layout
Sketch
Evaluate
Office Products
Flow Simulation
101700
101620
101540
101460
101380
101300
101220
101140
101060
100980
100900
Pressure [Pa]
*Isometric
Model
Motion Study 1
SolidWorks Premium 2009
Under Defined
Editing Assembly

2 Modify the Flow Trajectory Plot.

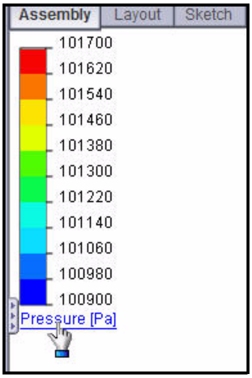
Place your mouse pointer over **Pressure (Pa)** in the Graphics area as illustrated.

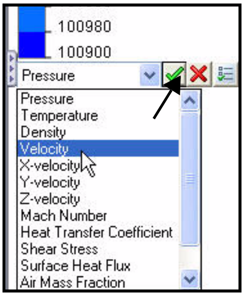
Click **Pressure (Pa)**. View the drop-down menu.

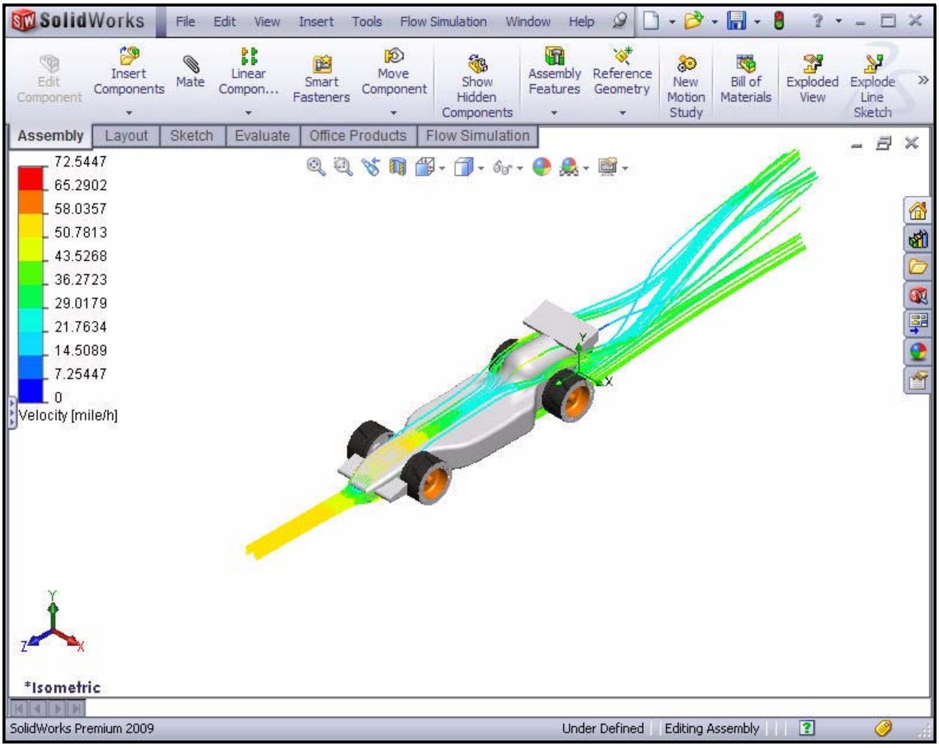
Click **Velocity**.

Click the **green check mark**.

View the new Flow Trajectory Plot.



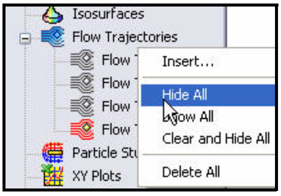




3 Hide all Flow Trajectories.

Right-click the `Flow Trajectories` folder.

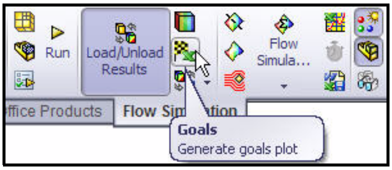
Click **Hide All**.



4 Save the document.

Click **Save** from the Menu bar toolbar.

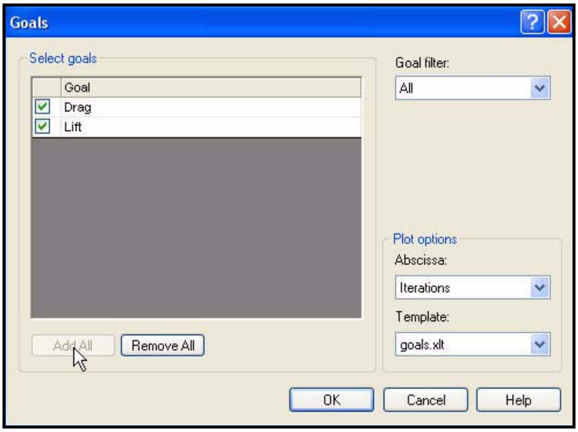
Quantitative Results



Note: Microsoft® Excel is needed for the next section.

1 Create a Goals plot.

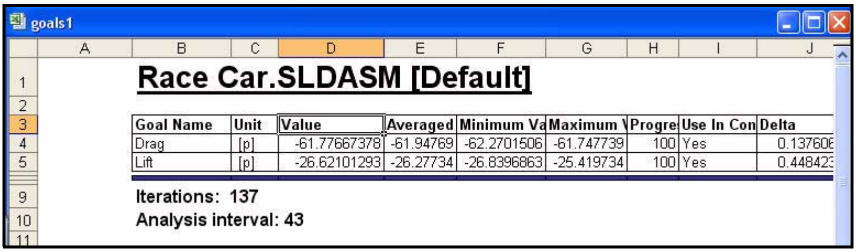
Click the **Goals** tool from the `Flow Simulation` tab. The `Goals` dialog box is displayed.



Click the **Add All** button.

Click **OK**.

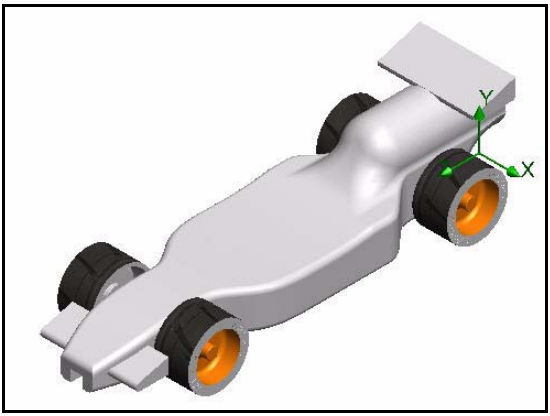
2 Excel spreadsheet. Microsoft® Excel is launched and a spreadsheet opens. Pay particular attention to the first three columns. They show the name of the goal, the units (gram-force, in this case) and the value.

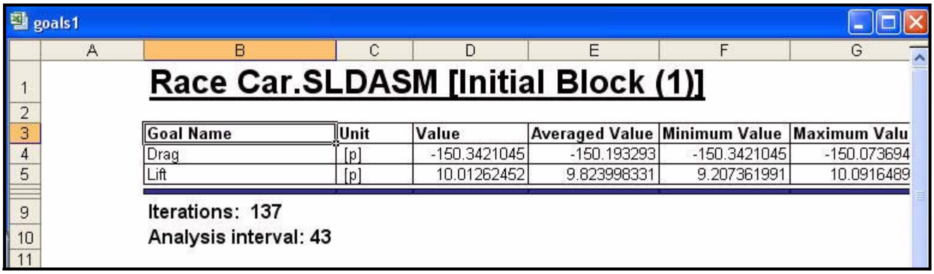

goals1

Race Car.SLDASM [Default]

Goal Name	Unit	Value	Averaged	Minimum Va	Maximum V	Progre	Use In Con	Delta
Drag	[p]	-61.77667378	-61.94769	-62.2701506	-61.747739	100	Yes	0.137606
Lift	[p]	-26.62101293	-26.27734	-26.8396863	-25.419734	100	Yes	0.448423

Iterations: 137
Analysis interval: 43



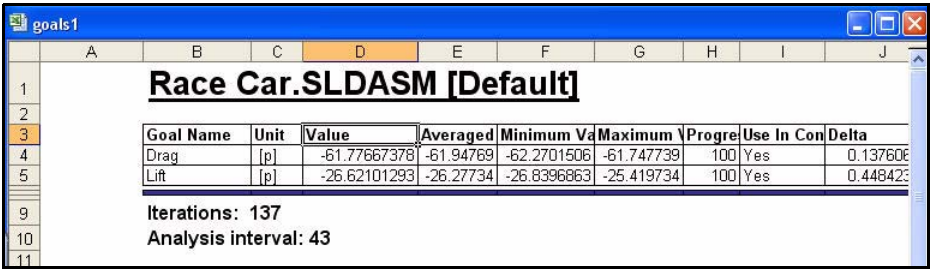


goals1

Race Car.SLDASM [Initial Block (1)]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Valu
Drag	[p]	-150.3421045	-150.193293	-150.3421045	-150.073694
Lift	[p]	10.01262452	9.823998331	9.207361991	10.0916489

Iterations: 137
Analysis interval: 43



goals1

Race Car.SLDASM [Default]

Goal Name	Unit	Value	Averaged	Minimum Va	Maximum V	Progre	Use In Con	Delta
Drag	[p]	-61.77667378	-61.94769	-62.2701506	-61.747739	100	Yes	0.137606
Lift	[p]	-26.62101293	-26.27734	-26.8396863	-25.419734	100	Yes	0.44842

Iterations: 137
Analysis interval: 43

The drag value for the new design is 61.77 grams-force. The drag value of the original block was 150.32 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:

The changes yielded about a 58.91% improvement in drag!

What About Lift?

It is interesting to note that the Initial Block design had a *upward lift* force of approximately 10.01 grams-force. The modified design has an *downward lift* force of about 26.62 grams-force. This is the effects of the Front Wing to keep the font end of the car down at high speeds.

3 Save and close Excel.
Click **Save**.

Close the Excel spreadsheet.

4 Save the document.

Click **Save** from the Menu bar toolbar.

5 Close all models and dialog boxes.

Click **File**, **Close**.

More to Explore

Using what you have learned, explore some additional design modifications. Or, better yet, start developing your own car body design. Using SolidWorks 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 SolidWorks Flow Simulation together you can easily explore many design variations. Have fun!

SolidWorks Flow Simulation

During this short session on using SolidWorks Flow Simulation, you have had a brief exposure to the main concepts of fluid-flow simulation. SolidWorks Flow Simulation gives you insight into parts and assemblies related to fluid flow, heat transfer, and forces on immersed or surrounded solids.

The only fluid-flow simulation product fully integrated with SolidWorks, SolidWorks Flow Simulation is incredibly easy to use; you simply tell the software what you're interested in instead of having to translate analysis design goals into numerical criteria and iteration numbers.

Access physical fluid models for engineering applications. SolidWorks Flow Simulation can analyze a wide range of real fluids such as air, water, juice, ice cream, honey, plastic melts, toothpaste, and blood, which makes it ideal for engineers in nearly every industry.

Simulate real-world operating conditions. SolidWorks Flow Simulation includes several types of boundary conditions to represent real-life situations.

Automate fluid-flow tasks. SolidWorks Flow Simulation utilizes a number of automation tools to simplify the analysis process and help you to work more efficiently.

Interpret results with powerful and intuitive visualization tools. Once you have completed your analysis, SolidWorks Flow Simulation offers a variety of results visualization tools that allow you to gain valuable insight into the performance of your model.

Collaborate and share analysis results. SolidWorks Flow Simulation makes it easy to collaborate and share analysis results effectively with everyone involved in the product development process.