

Lesson 5

Making Design Changes

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

- Model free-form shapes using advanced lofting and filling techniques;
- Create a configuration in a part and use part configurations in an assembly;
- Reanalyze the model.

Changing the CO₂ Car Design

Based on the analysis of the car using COSMOSFloWorks, we conclude that the front end of the body needs to be redesigned to direct the air around the car with a more gradual change in direction. We need to make it rounder in shape.

Configurations

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

Both parts and assemblies can support configuration adjustments.

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

COSMOSFloWorks creates a configuration to store all analysis data. The name of this configuration is the name you entered in the COSMOSFloWorks Wizard. In this case it is `Test1`.

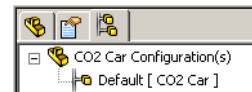
Modifying the Model

1 Open the part.

In the FeatureManager design tree right-click on the car part in the assembly file and select **Open Part**.

2 Switch to ConfigurationManager.

Click the ConfigurationManager tab  to change from the FeatureManager design tree to the ConfigurationManager.



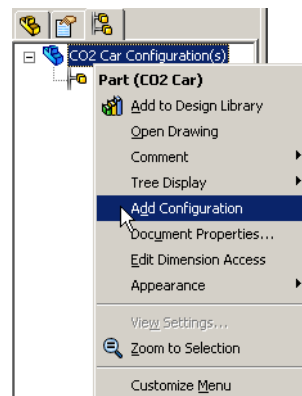
3 Add a new configuration.

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

Type `Changed Design` as the name.

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

Click **OK** to add the configuration.



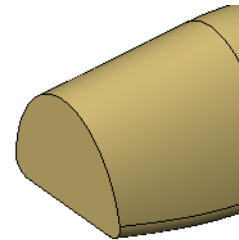
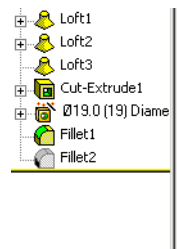
Note: **Suppress** is used to temporarily remove a feature. When a feature is suppressed, the system treats it as if it doesn't exist. This means other features that are dependent on it will be suppressed also. Suppressed features can be unsuppressed at any time. The **Suppress features** option means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

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



4 Suppress.

Click the FeatureManager design tree tab. Right-click **Fillet2** and select **Suppress**. By removing the fillet at the front of the vehicle we can now create a new, less blunt nose feature.



5 Changing the fillet value.

It will not be enough however, to just reconfigure the front of the body. To ensure a smooth front end and gradual transition between the newly created front feature and the rest of the car body, it is important to also make the body rounder. With a larger radius on **Fillet1** we will be able to blend the new surface with the model more successfully.

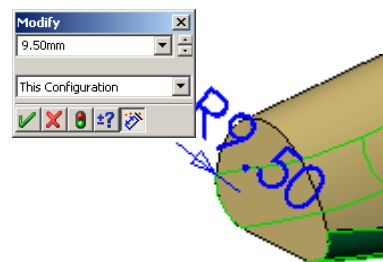
Double-click the **Fillet1** feature in the FeatureManager design tree. The fillet's dimension appears in the graphics area.

Double-click the dimension text. The **Modify** dialog box appears.

Change the radius value to **9.5mm**.

Click **Rebuild**  to apply the change.

By increasing the radius value the curves of the body will be smoother both at the front and along the length of the body.



Using Surface Functions

Rounding off the front of the car represents a bit of a challenge. Traditional solid modeling tools are not well suited to the task. Surface modeling techniques however, provide additional capabilities.

Let's Round off the Front of the Car

6 Filled surface.

Zoom in on the front of the car. The curved surface we create will replace the existing flat face.

Click **Insert, Surface, Fill**.

Note: If you have the Surfaces toolbar turned on, you can also click the **Filled Surface**  tool.

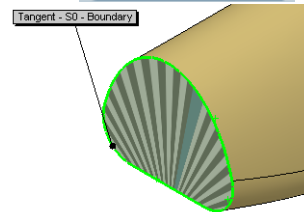
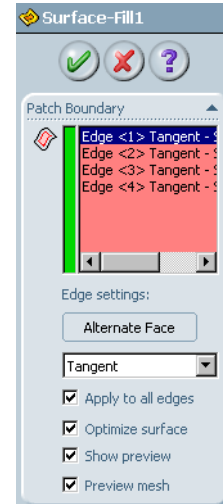
7 Edge settings.

Under **Edge settings**, set the **Curvature Control** to **Tangent**, which creates a surface within the selected boundary that is tangent to the patch edges.

Make sure the **Apply to all edges** and **Preview mesh** options are selected.

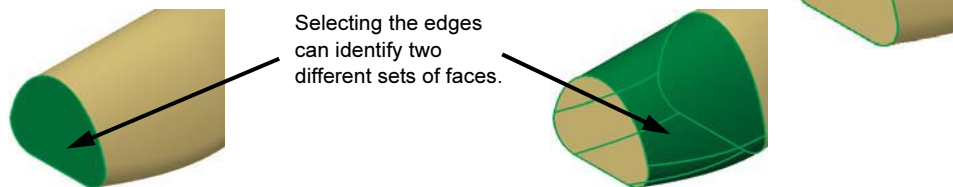
8 Patch boundary.

Define the **Patch Boundary** - the edges of the patch we will apply - by selecting the four edges of the planar face.



Tangent to What?

As indicated in step 7, we want the surface we are creating to be tangent to the body of the car. However, because any edge is the boundary between *two* faces, simply selecting the edges does not unambiguously identify the reference faces for the tangency.



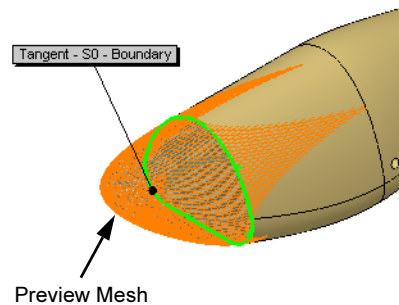
9 Alternate Face.

As you can see from the graphic in step 8, the system has selected the flat face for the tangency control. This is not what we want.

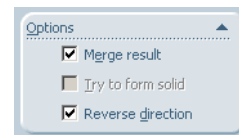
To make the patch tangent to the body of the car, click **Alternate Face**.

The preview mesh appears.

However, you aren't done yet.

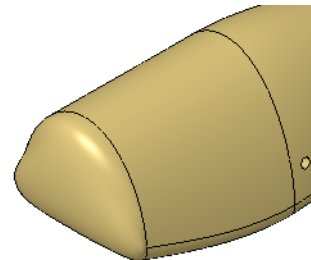
**10 Merge result.**

Select the **Merge result** option, and click **OK**. This ensures that the new fill surface feature becomes an integral part of the solid body.



When patching a solid with a filled surface there are often two possible results. If the fill surface cannot be merged into the solid click **Reverse direction** to correct.


Note: The illustration at the right shows the poor result we would have gotten if we had not changed the fillet radius to 9.5mm in step 5.

**11 Create a new sketch.**

To streamline the body further, we will make a cut at the rear of the solid.


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

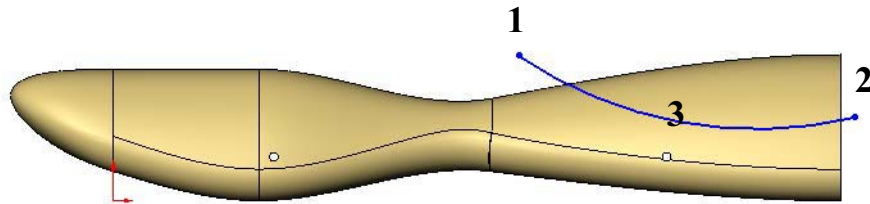
12 Right view orientation.

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

13 Create 3 Point Arc for the body cut.

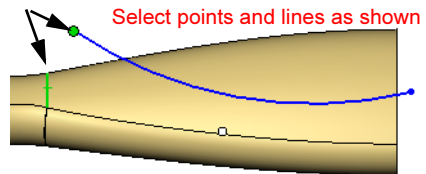
We can make the car even more streamlined by cutting a section from the back of the body.


Click the **3 Point Arc**  tool from the sketch toolbar or from the **Tools, Sketch Entities, 3 Point Arc**. Click in order of 1, 2, and 3 as shown. Turn off the **3 Point Arc** tool.



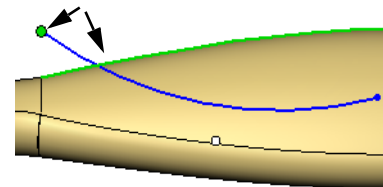
14 Add Relations.

Hold down **Ctrl** and select the first point and the edge between the two loft features as shown in the illustration.

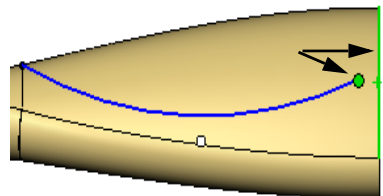


Select **Coincident**  from the **Add Relations** list in the PropertyManager.

Add a second **Coincident** relation for the first point and the upper silhouette edge of the body.

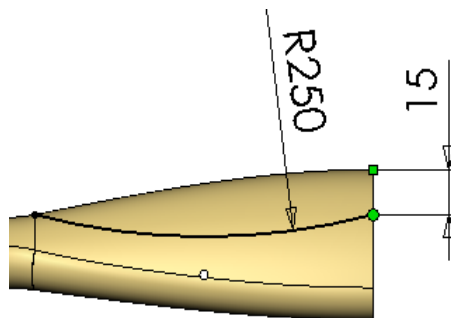


Add a final **Coincident** relation between the second point and the back edge.




15 Add dimensions.

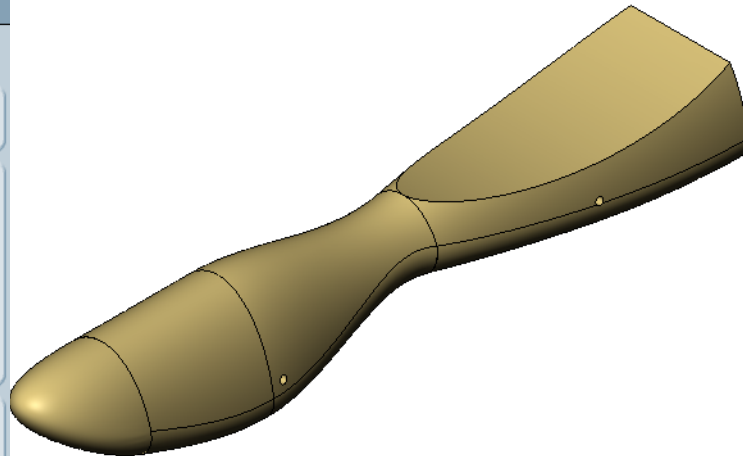
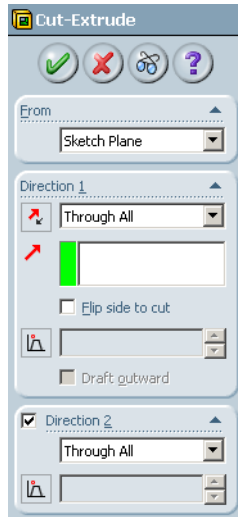
Using the **Dimension** tool, add dimensions to fully define the sketch as shown.




16 Cut.

Click **Insert**, **Cut**, **Extrude**, or click **Extruded Cut** tool  on the Feature toolbar. The **End Condition** type of both **Direction 1** field and **Direction 2** field is automatically set to **Through All**. This is because you cannot cut partway through with an open sketch contour.

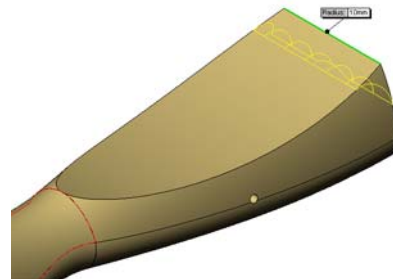
Click **OK**.


**17 Add fillet.**

Change the view back to the **Isometric** view.

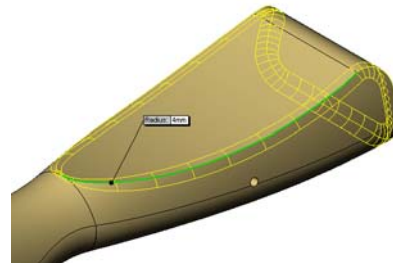
Click **Fillet**  and set the fillet radius to **10mm** and choose the edge as shown.

Click **OK**.

**18 Add another fillet.**

Click **Fillet**  again and set the fillet radius to **4mm** and choose the edge as shown.

Click **OK**.



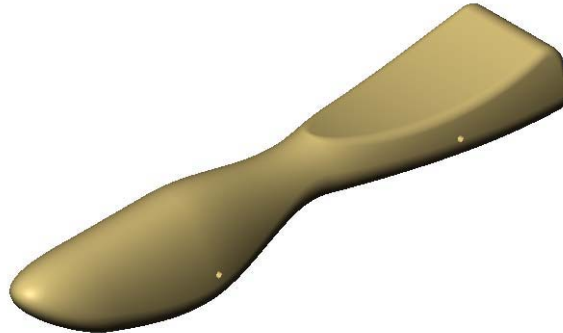
19 Complete.

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

20 Save and close the part file.

21 Reopen the assembly.

When you reopen the assembly, the latest version of the car will be referenced; all of the changes you just made will be present.



Note: If you did *not* close the assembly at the end of Lesson 4, you will get a message when the assembly window become visible. The changes to the car are detected and SolidWorks asks if you want to rebuild the assembly.


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

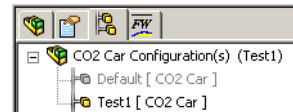
Click **Yes**.

Assembly Configurations

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

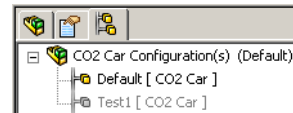
1 Switch to ConfigurationManager.


Click the ConfigurationManager tab  to change from the COSMOSFloWorks analysis tree to the ConfigurationManager.



2 Switch back to the default configuration.

Position the cursor over the Default configuration and double-click it to make it active.



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

SolidWorks

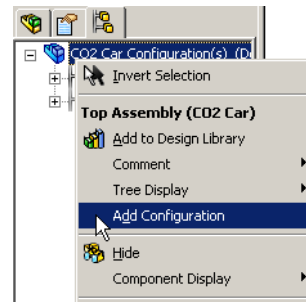
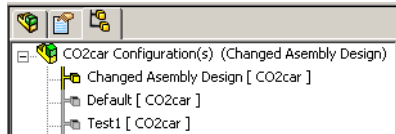
Making Design Changes

3 Add a new configuration.

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

Type the configuration name Changed Assembly Design in the **Add Configuration** window and click **OK**.

A new configuration is added to the assembly.



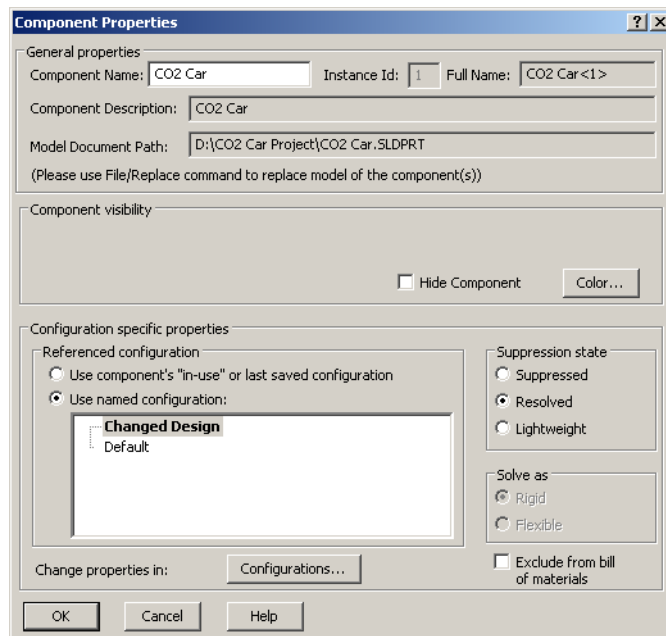
However, the new configuration is identical to the Default configuration. We need to change it so that this configuration shows the modified car body.

4 Referencing a different configuration.

Switch back to the FeatureManager design tree.

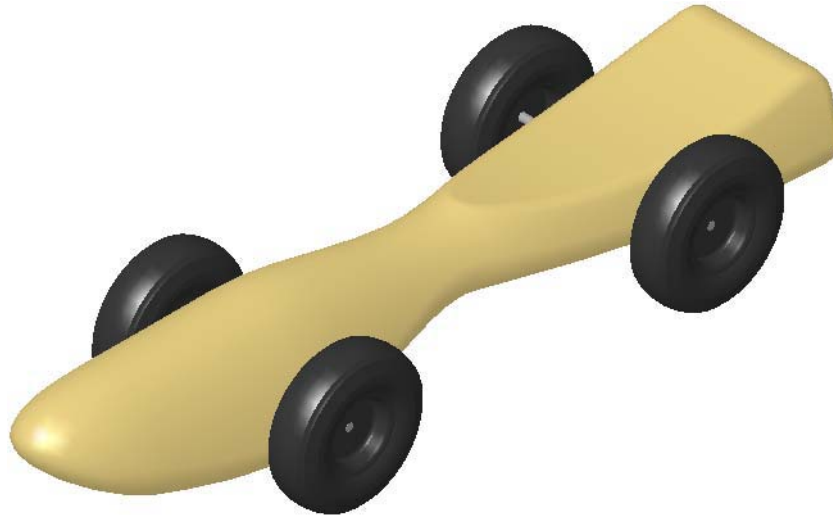
Right-click on the CO2 Car component in the FeatureManager design tree and select **Component Properties...**

Select **Use named configuration** and Changed Design in the **Referenced configuration** area. Click **OK**.



5 Complete.

You can change any aspect of the component to a different configuration.



Analyze the Modified Design with COSMOSFloWorks


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

1 Activate the analysis configuration Test1.

In the ConfigurationManager, double-click the configuration named Test1. This is the fastest and easiest way to switch between configurations.

The COSMOSFloWorks analysis tree tab  reappears.

2 Switch to the analysis tree.

Click the COSMOSFloWorks analysis tree tab  to access the analysis features.

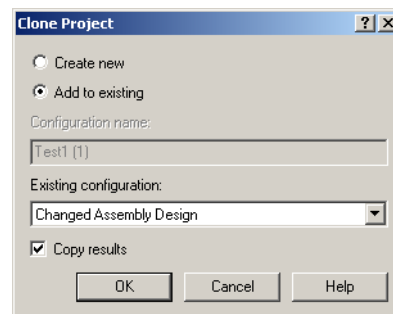
3 Clone project.

Right-click the uppermost feature, Test1, and select **Clone Project** from the shortcut menu.

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

From the **Existing configurations** list, select Changed Assembly Design.

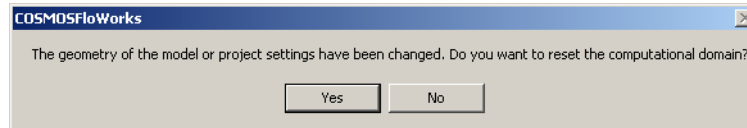
Select the **Copy results** option and click **OK**.



4 Messages.

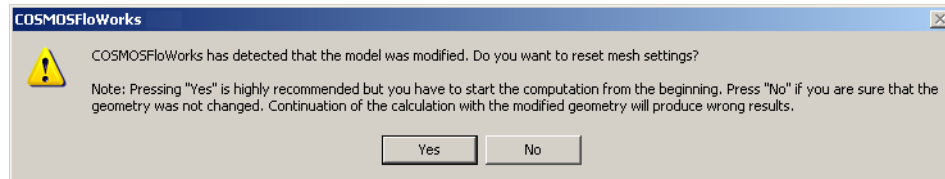
When you click **OK**, you will get a couple of messages:

Computational Domain



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

Mesh Settings



The geometry of the Changed Assembly Design configuration has changed. We've rounded the nose and made other changes. The mesh should be reset.

Click **Yes**.

5 Run the solver.

In the COSMOSFloWorks analysis tree, right-click the uppermost feature, Changed Assembly Design, and select **Run** from the shortcut menu.

Examine the Results

1 Load the results.

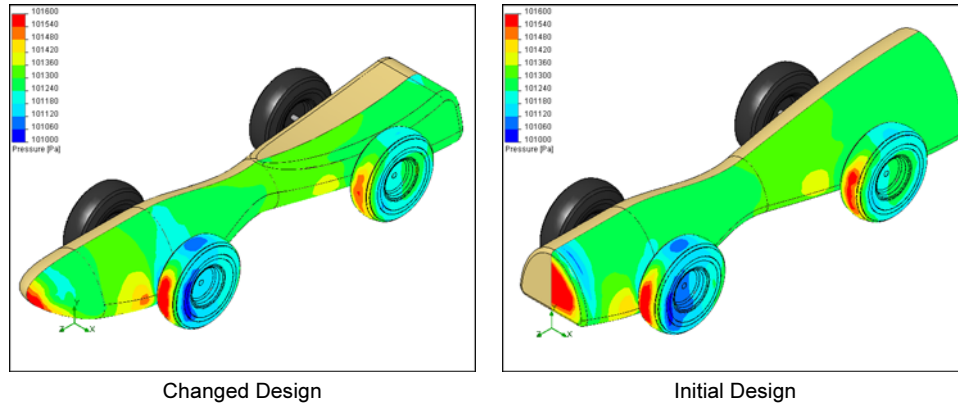
In the COSMOSFloWorks analysis tree, right-click Results and select **Load Results** from the shortcut menu.

2 Show the surface plot.

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

3 Surface plot results.

For comparison purposes in this book, we have also shown the surface plot for the initial design.



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

Flow Trajectories

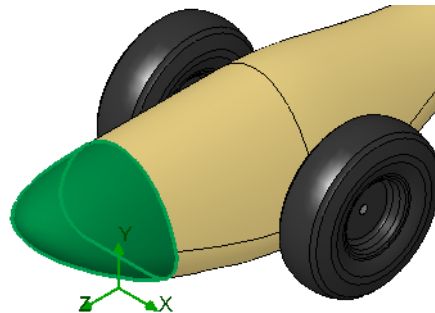
Now let's look at the flow trajectories.

4 Hide the surface plot.

Right-click *Surface Plot 1* and select **Hide**.

5 Missing face for the first set of flow trajectories.

The first set of flow trajectories used the flat face on the front of the car body as the reference. That face doesn't exist in this configuration. It was absorbed when the fill surface was merged to the body of the car. Therefore, we must redefine the reference before we can display the plot.

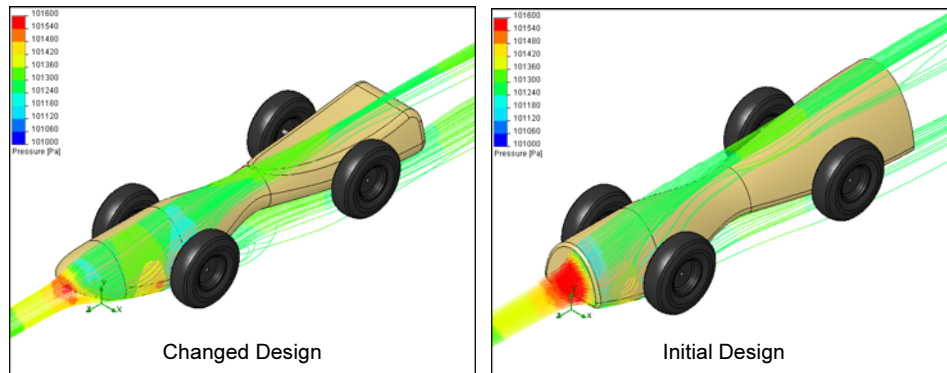


6 Edit definition.

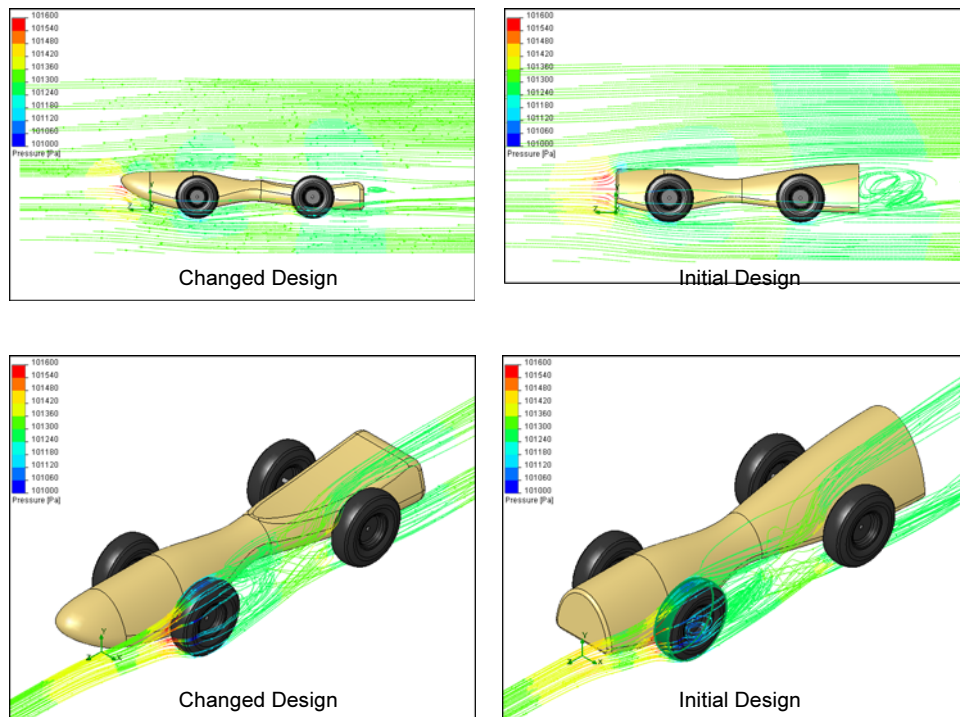
Right-click Flow Trajectories 1 and select **Edit Definition**.

Select the rounded face at the nose of the car.

Click **Apply** and **OK**.

**7 Display the other flow trajectories.**

One at a time, display the other flow trajectory plots for the changed design.



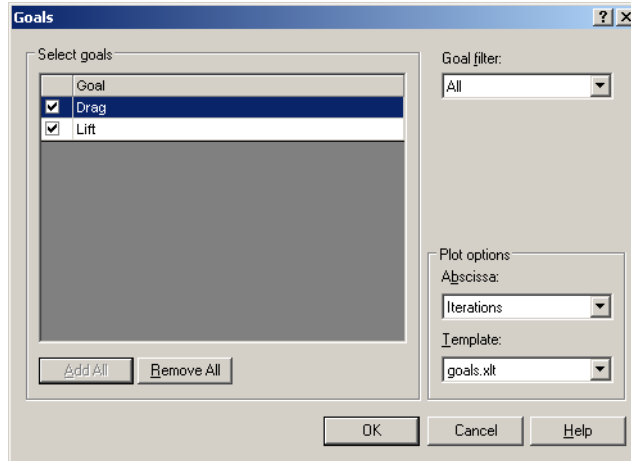
Quantitative Results

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

1 Create a goals plot.

In the COSMOSFloWorks analysis tree, expand the Results listing and right-click Goals. Select **Insert** from the shortcut menu.

Click **Add All** to add **Drag** and **Lift** to the goals. Click **OK**.



2 Excel spreadsheet.

Microsoft® Excel is launched and a spreadsheet opens.

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

The drag value for the new design is 0.330514367 newtons. The drag value for the original design is 0.410999779 newtons.

CO2 Car.SLDASM [Changed Assembly Design]		
Goal Name	Unit	Value
Drag	[N]	-0.330514367
Lift	[N]	0.020325048

Iterations: 88
Analysis interval: 39

CO2 Car.SLDASM [Test1]		
Goal Name	Unit	Value
Drag	[N]	-0.410999779
Lift	[N]	-0.025862278

Iterations: 88
Analysis interval: 39

To find the percentage of improvement use this formula:

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

For simplicity we will round to 3 decimal places. Substituting we get:

$$\frac{(0.411 - 0.331)}{0.411} \times 100 = 19.465$$

The changes yielded about a 19.5% improvement.

What About Lift?

It is interesting to note that the original design had a *downward* lift force of

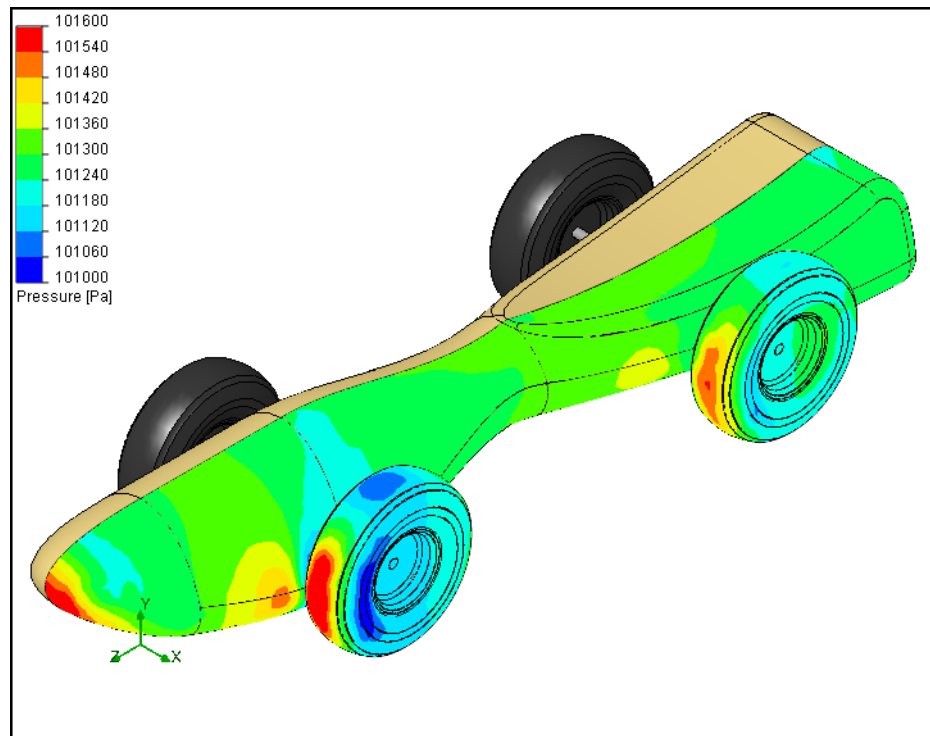
approximately 0.026 newtons. The modified design has an *upward* lift force of 0.020 newtons. As long as this force is less than the weight of the car, it could be beneficial. This type of car does not rely on traction for acceleration and an upward lift force may reduce rolling friction between the wheels and the track.

CO2 Car.SLDASM [Test1]			CO2 Car.SLDASM [Changed Assembly Design]		
Goal Name	Unit	Value	Goal Name	Unit	Value
Drag	[N]	-0.410999779	Drag	[N]	-0.330514367
Lift	[N]	-0.025862278	Lift	[N]	0.020325048
Iterations: 88			Iterations: 88		
Analysis interval: 39			Analysis interval: 39		

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

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

Looking at the wheels in the surface plot gives us a clue. The red color in the surface plot indicates that there are areas of high pressure on the leading portions of the front and rear wheels. Let's look at this data quantitatively.



Surface Parameters

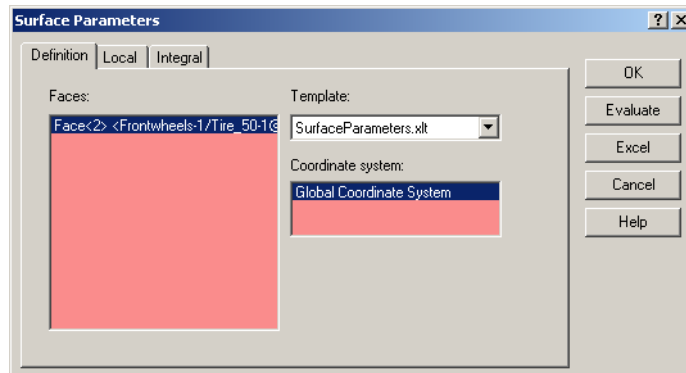
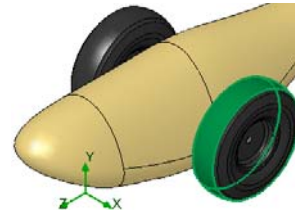
Surface Parameters are results that allow you to examine the forces on selected faces in the model. We will create a report for the front and rear wheels of the assembly.

3 Insert surface parameters.

In the COSMOSFloWorks analysis tree, expand the Results listing and right-click Surface Parameters.

Select **Insert** from the shortcut menu.

Select the face that represents the tread of the wheel and click **OK**.



4 Excel spreadsheet.

An Excel spreadsheet opens.

Surface Parameters 1[Changed Assembly Design]				
Face<2> <Frontwheels-1/Tire_50-1@Revolve1>				
Global Coordinate System				
Iteration [] 88				
Local parameters				
Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	100998	102271	101276	
Temperature (fluid) [K]	293.557	293.563	293.561	
Density [kg/m ³]	1.19941	1.21478	1.20259	
Total Pressure [Pa]	100998	102271	101276	
Dynamic Pressure [Pa]	0	0	0	
Velocity [m/s]	0	0	0	
Heat Flux [W/m ²]	0	0	0	
Air Mass Fraction []	1	1	1	
Air Volume Fraction []	1	1	1	
Integral parameters				
Parameter	Value	x-component	y-component	z-component
Force [N]	0.0972543	0.00812772	-0.00116344	-0.0969071
Shear force [N]	0.00267457	0.000350107	8.39E-05	-0.00265023
Torque [N*m]	0.00326494	-0.00152175	0.00288278	-0.000183459
Heat transfer rate [W]	0			
Mass flow rate [kg/s]	0			
Volume flow rate [m ³ /s]	0			
Normal force [N]	0.0945854	0.00777761	-0.0012473	-0.0942569
Mass flow rate of Air [kg/s]	0			
Volume flow rate of Air [m ³ /s]	0			
Surface area [m ²]	0.00174792	1.84E-05	9.22E-06	-1.42E-07

The information we are interested in is in the section labeled *Integral Parameters*. Specifically we want the z-component of the Force [N]. For this example it is 0.0969 newtons.

5 Repeat for rear wheel.

Display the surface parameters for the rear wheel.

Surface Parameters 2[Changed Assembly Design]				
Face<3> <Rearwheels-1\Fire_50-1@Revolve1>				
Global Coordinate System				
Iteration [] 88				
Local parameters				
Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	101029	101838	101285	
Temperature (fluid) [K]	293.557	293.565	293.561	
Density [kg/m ³]	1.19976	1.2087	1.20244	
Total Pressure [Pa]	101029	101838	101285	
Dynamic Pressure [Pa]	0	0	0	
Velocity [m/s]	0	0	0	
Heat Flux [W/m ²]	0	0	0	
Air Mass Fraction []	1	1	1	
Air Volume Fraction []	1	1	1	
Integral parameters				
Parameter	Value	x-component	y-component	z-component
Force [N]	0.0876732	-0.000330503	-0.00447483	-0.0875583
Shear force [N]	0.00196027	-0.000455753	3.58E-05	-0.00190622
Torque [N*m]	0.00372427	-0.00216457	0.00302712	-0.000146248
Heat transfer rate [W]	0			
Mass flow rate [kg/s]	0			
Volume flow rate [m ³ /s]	0			
Normal force [N]	0.0857708	0.00012525	-0.00451052	-0.0856521
Mass flow rate of Air [kg/s]	0			
Volume flow rate of Air [m ³ /s]	0			
Surface area [m ²]	0.00174792	4.11E-05	1.69E-06	-3.47E-06

In this case we see the force on the rear wheel is 0.0876 newtons.

What Does This Tell Us?

Adding the drag forces on the front and rear wheels (0.0969 + 0.0876) gives us a total of 0.1845 newtons of drag force that is attributable to the wheels. Comparing that to the total drag force of 0.331 newtons, we see that the wheels are a major contributor to drag. In fact, they represent over 55% of the drag!

$$(0.1845 \div 0.331) \times 100 = 55.74$$

Note: In the note on page 68, we said that to find the total drag on the car you would have to double the drag force values because we used symmetry during the calculations. Why didn't we double the values for this comparison? The answer is we displayed the surface parameters for only one front wheel and one rear wheel. That's only half of the wheels. Since the total drag force data is based on analyzing half of the car (symmetry) and we are calculating the relative contribution of half of the wheels, the proportions are correct.

Did Changing the Body Really Help?

We have already seen that redesigning the body gave us nearly a 20% reduction in drag. Since the wheels represent such a significant portion of the total drag, we must have made a major improvement in the body. Without running an analysis on just the body in its original and redesigned configurations, we can make some approximations using the data we already have.

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

Drag Source	Initial Design	Changed Design
Entire Model	0.411	0.331
Front Wheel	0.092	0.097
Rear Wheel	0.102	0.088
Total for Wheels	0.194	0.185
Approx. amount attributable to Body (Entire Model - Front & Rear Wheels)	0.217	0.146

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

$$\frac{(0.217 - 0.146)}{0.217} \times 100 = 32.719 \text{ or about } 32.7\%$$

Note: This is only an approximation because there are a number of factors we did not take into account. For example, the drag force on the wheels was estimated using only the large surface that represents the tread. We did not include the other faces on the wheel (shown in green in the illustration at the right), or the exposed portions of the axles.



So yes, changing the body had a major impact on the aerodynamics. It resulted in approximately a 33% reduction in drag on the body. And even with the wheels contributing so much drag, it resulted in nearly a 20% reduction in drag on the car as a whole.

More To Explore

Using what you have learned, explore some additional design modifications. Consider modifying the car body even further to enclose the wheels. Or consider using narrower wheels to reduce drag. With Solidworks and COSMOSFloWorks together you can easily explore many design variations.

