Created in COMSOL Multiphysics 6.1

Large Eddy Simulation of a Sports Car

Introduction

The shape of a car body depends on several factors; besides being appealing, it must also provide comfort and good performance. In the early 20th century, the first two criteria dominated the design. Early versions of the model T resembled motor-driven horse carriages and sometimes even lacked windshield, side-windows, and doors. A later example is the London black cab, which had to be tall enough to comfortably accommodate a passenger wearing a bowler hat and have a turning circle of 25 ft. As the speed of travel increased, performance, both with regard to fuel consumption and stability, became an increasingly important design criterion, leading to the aerodynamically shaped vehicles we see today. The fuel consumption depends on the drag force, which can be obtained, with decent accuracy, from a stationary RANS (Reynolds-averaged Navier–Stokes) simulation. Other important design criteria are related to the fluctuating forces that may induce vibrations causing discomfort to the driver and passengers, and in extreme cases even leading to fatigue of engine and chassis parts. With LES (large eddy simulations), it is possible to resolve the spatially and temporally fluctuating forces acting on a car traveling at high speeds. The fluctuating forces exerted by the flow can be used as surface loads in a structural analysis.

In this model, we perform a LES of a sports car traveling at a speed of 180 km/h. The transient analysis is initialized from the results of a stationary RANS simulation, which in turn has been initialized from a potential-flow solution in order to accelerate the convergence of the RANS simulation.

Model Definition

The car model, which has length L = 4.46 m, width W = 1.92 m, and height H = 1.17 m, is placed inside a block, with length $13L$, width $7W$ and height 10 m, at two car lengths distance from the front face. The model geometry is shown in Figure 1.

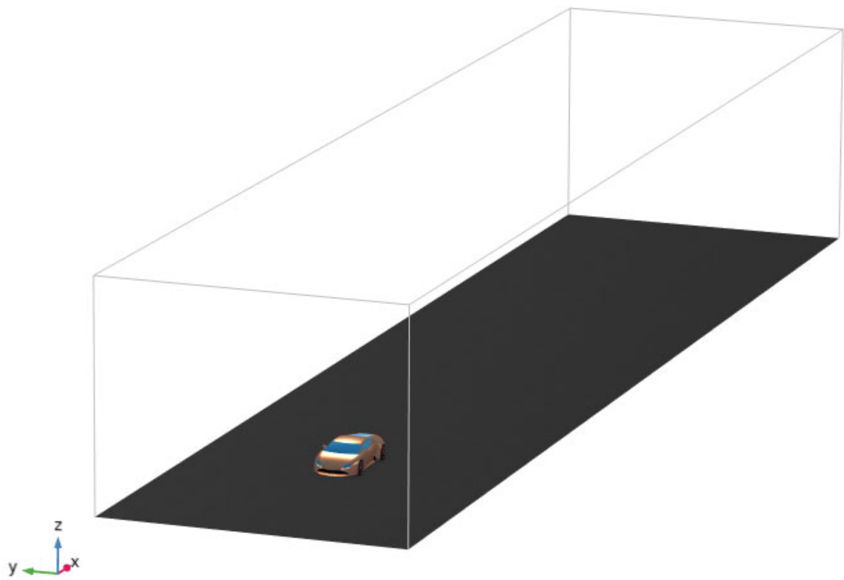


Figure 1: Geometry of the computational domain.

The inlet velocity at the front face of the domain, and the velocity of the road (bottom face) is set to U_0 = 50 m/s in the x direction, whereas a slip condition is applied at the side and top faces of the domain. At the rear face of the domain, the gauge pressure is set to zero. On the surfaces of the wheels, the velocity is set to

$$\mathbf{u}_w = -\omega\hat{\mathbf{y}} \times (\mathbf{r} - \mathbf{r}_c) \tag{1}$$

where $\hat{\mathbf{y}}$ is the unit vector in the y direction, $\mathbf{r} = (x, y, z)$ is a point on a wheel, $\mathbf{r}_c = (x_c, y_c, z_c)$ is the center of the corresponding wheel, and

$$\omega = \frac{U_0}{R_w} \qquad (2)$$

is the magnitude of the angular velocity of the wheels, where R_w is the wheel radius. To model the tire deformation, the road is placed 1 cm up from the lowest point on the tires. The velocity is smoothed exponentially, with a length scale of 2.5 mm, from the tires to the road. Figure 2 shows the magnitude of the applied boundary velocity on the road and front wheel.

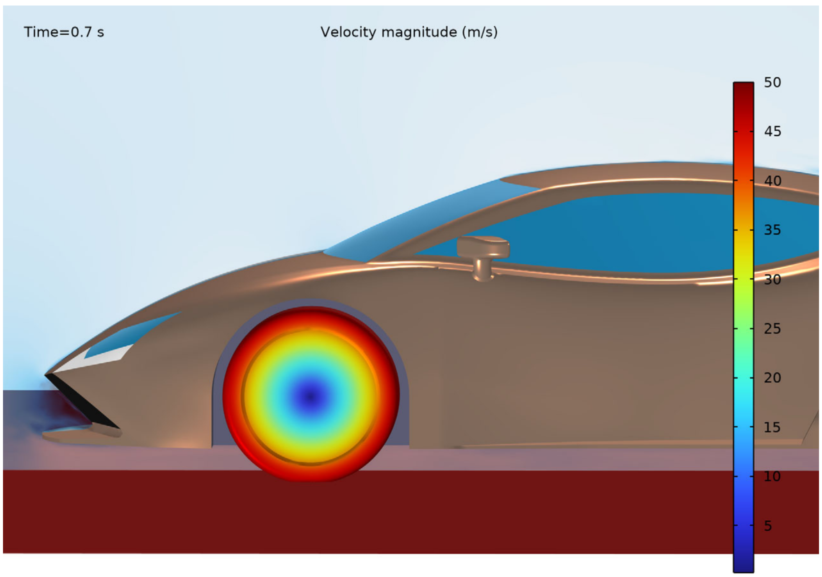


Figure 2: Magnitude of velocity in m/s on the road and front wheel boundaries.

The large eddy simulation is initialized from a RANS k-ε solution. In order to accelerate the convergence of the RANS simulation, the k-ε simulation is in turn initialized from a potential-flow solution. For irrotational flow, a velocity potential, ϕ, can be defined such that $\mathbf{u} = \nabla\phi$. Assuming that the flow is incompressible, ϕ can be obtained by solving Laplace's equation,

$$\Delta\phi = 0 \qquad (3)$$

with

$$
\begin{array}{ll}
\hat{\mathbf{n}} \cdot \nabla\phi = -U_0 & \text{at the front face} \\
\phi = 0 & \text{at the rear face} \\
\hat{\mathbf{n}} \cdot \nabla\phi = 0 & \text{at all other boundaries}
\end{array}
$$

The initial values for velocity and pressure, applied in the RANS k-ε simulation, are obtained from the potential-flow solution as

$$
\mathbf{u}_{\text{init}} = \nabla\phi
$$

$$
p_{\text{init}} = \frac{\rho}{2}(U_0^2 - \nabla\phi \cdot \nabla\phi)
$$

The large eddy simulation is first computed for 0.6 s (corresponding to roughly seven car-length passages) to obtain a fully developed turbulent flow field, and after that data is sampled every 2 ms during 0.1 s.

Results and Discussion

Figure 3, Figure 4, and Figure 5 show slice plots of the velocity magnitude for the potential-flow solution, the RANS k-ε solution, and the LES solution at $t = 0.7$ s respectively. The potential-flow solution quickly approaches its asymptotic state away from the car boundaries; the superposition of a uniform flow and a source doublet. Note that no drag force on the car body can be derived from the potential-flow solution, and thus it is only useful as an initial guess for a more accurate model.

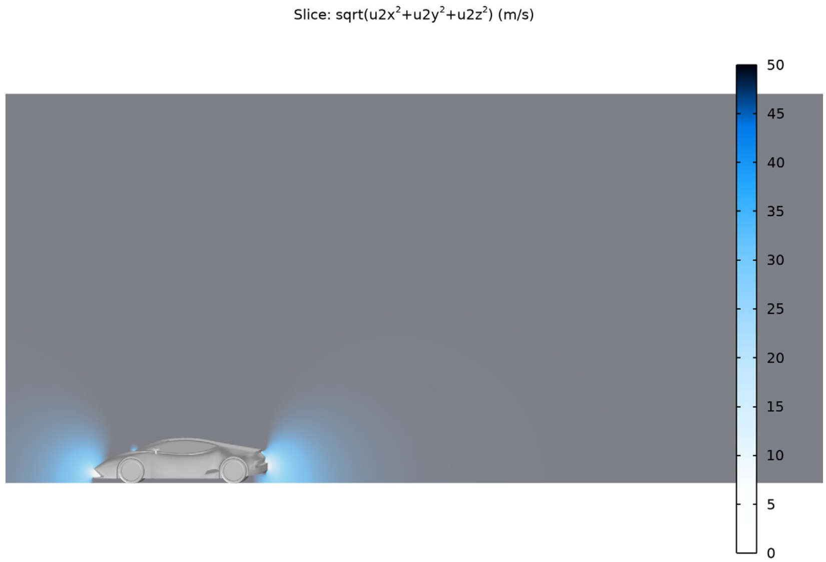


Figure 3: Velocity magnitude in m/s from the potential-flow solution.

The RANS solution has a wake extending all the way to the outlet boundary. This breaking of the fore-aft pressure symmetry (seen in the potential-flow solution) induces a form drag on the car body. The form drag is the major contribution to the total drag on the car.

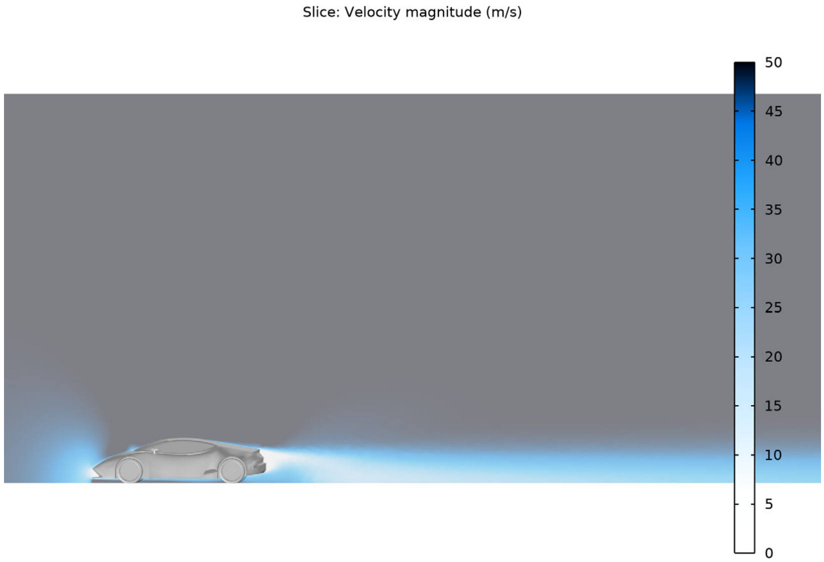


Figure 4: Velocity magnitude in m/s from the RANS k-ε solution.

Transient vortex detachment can be observed in the LES solution. The drag coefficient, C_d, is given by

$$C_d = \frac{F_x}{\frac{1}{2}\rho U_0^2 A} \tag{4}$$

where F_x is the x-component of the force on the car and A is its area cross section:

$$\begin{aligned} F_x &= \int_S \left(p n_x + \tau_w \frac{u_{tx}}{|\mathbf{u}_t|}\right) dS \\ A &= \int_S \max(n_x, 0)\, dS \end{aligned} \tag{5}$$

where $\tau_w = \rho u_\tau^2$ is the shear stress at the wall, $\mathbf{u}_t$ is the tangential velocity, and the integrations are performed over the surfaces of the car body. The form drag can be obtained by leaving out the shear-stress term in the force calculation.

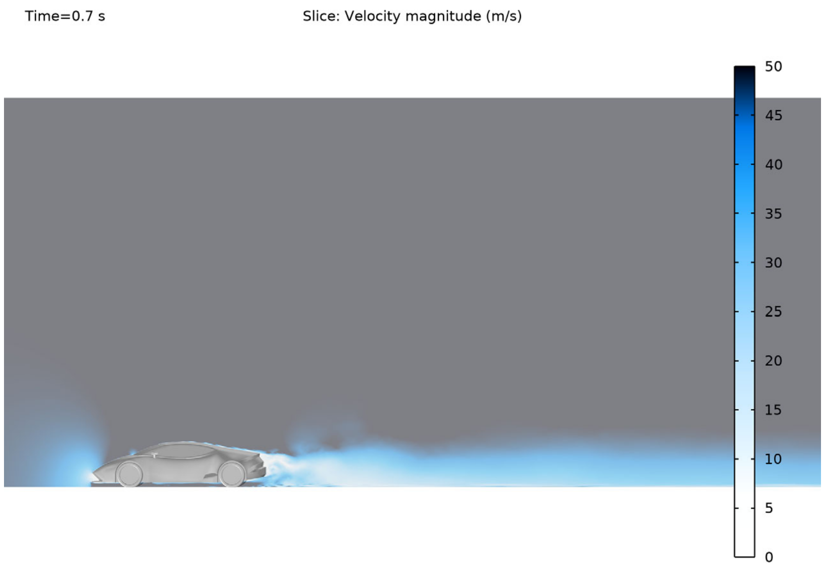


Figure 5: Velocity magnitude in m/s from the LES solution at $t = 0.7$ s.

Drag coefficients for the form drag and total drag are shown in Figure 6 as a function of time from t = 0.6 s to t = 0.7 s. The (total) drag coefficient fluctuates around the relatively low value of 0.2.

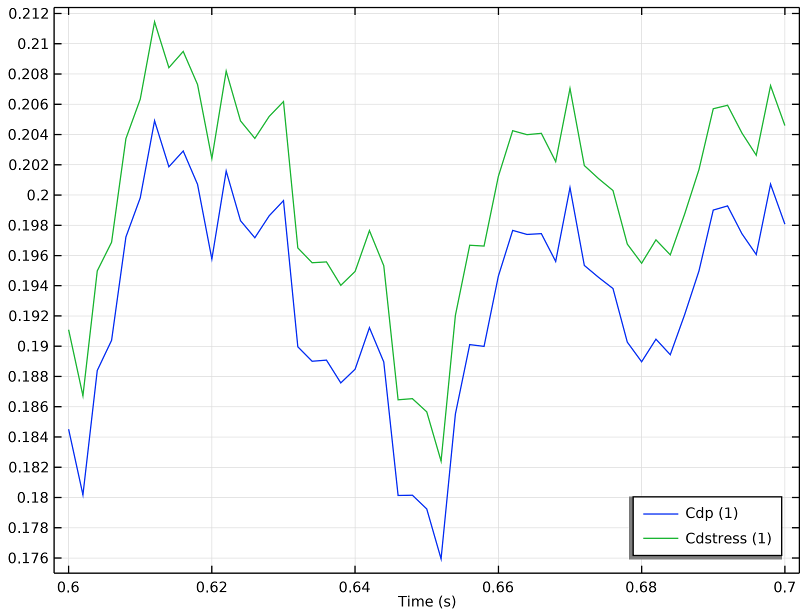


Figure 6: Drag coefficients based on form drag (blue) and total drag (green) from the LES solution between t = 0.6 s and t = 0.7 s.

Instantaneous streamlines and slice plots of the velocity magnitude at t = 0.7 s are shown in Figure 7 and Figure 8, respectively.

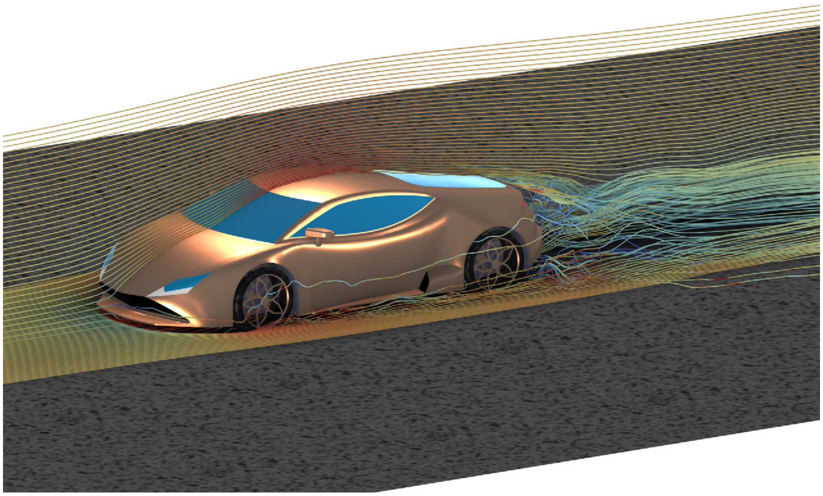


Figure 7: Instantaneous streamlines from the LES solution at t = 0.7 s.

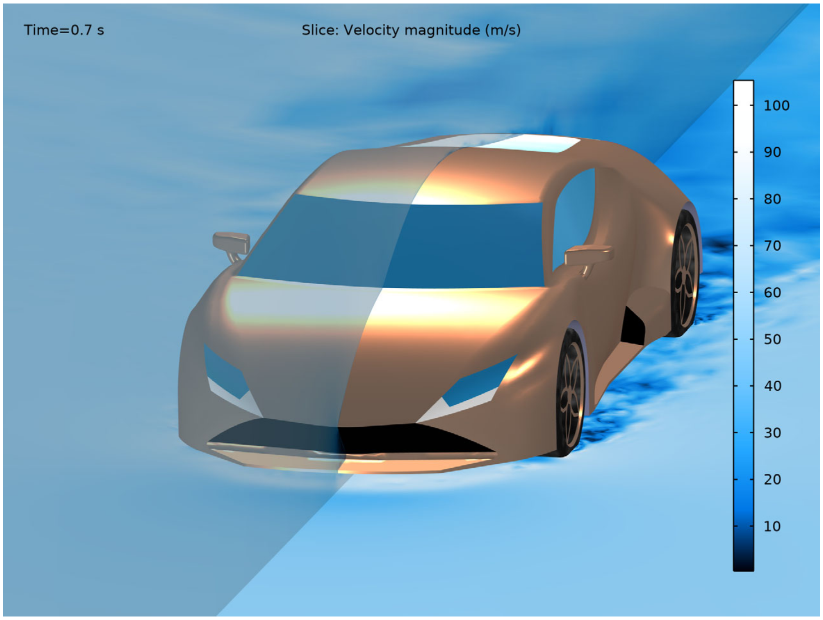


Figure 8: Slice plots of the velocity magnitude from the LES solution at $t = 0.7$ s.

The fluctuating surface stresses, obtained from the LES simulation, can be applied as loads in structural analyses of various body parts to look at the excitation of eigenmodes, which can eventually lead to fatigue.

Notes About the COMSOL Implementation

The car body has several simplifications. For example, the cowl panel is neglected, and the different parts of the body are assumed to be assembled perfectly, with no gaps nor misalignments between different body panels. In reality, the body of a real supercar is full of small gaps between body panels and doors, of the order of magnitude of a millimeter. These gaps may induce additional turbulence resulting in a slightly higher drag coefficient. The mesh resolution is purposely made coarse in order to reduce the computation time.

Application Library path: `CFD_Module/Single-Phase_Flow/sports_car`

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Large Eddy Simulation>LES RBVM (spf)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Mathematics>Classical PDEs>Laplace Equation (lpeq)**.
5 Click **Add**.

Set the units for the Laplace equation interface so that the dependent variable corresponds to the velocity potential.

6 Click **Define Dependent Variable Unit**.
7 In the **Dependent variable quantity** table, enter the following settings:

Dependent variable quantity	Unit
Custom unit	m^2/s

8 Click **Define Source Term Unit**.
9 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	1/s

10 Click **Study**.
11 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Stationary**.
12 Click **Done**.

Define parameters for the moving wall boundary conditions.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
U0	50[m/s]	50 m/s	Car velocity
R_w	0.33[m]	0.33 m	Wheel radius
L_wb	103.1[in]	2.6187 m	Length, wheel base
x_w	40.4[in]	1.0262 m	Position, front wheel

DEFINITIONS

Variables 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Variables**.

Define variables for the moving wall conditions (the velocity of the tires).

3 In the **Settings** window for **Variables**, locate the **Variables** section.

4 In the table, enter the following settings:

Name	Expression	Unit	Description
Uwx	-U0*(z/R_w-1)*(1-exp(-(z-0.01[m])/0.0025[m]))+U0*exp(-(z-0.01[m])/0.0025[m])	m/s	Tire velocity, x-component
Uwz_front	U0*(x-x_w)/R_w*(1-exp(-(z-0.01[m])/0.0025[m]))	m/s	Front tire velocity,z-component
Uwz_rear	U0*(x-x_w-L_wb)/R_w*(1-exp(-(z-0.01[m])/0.0025[m]))	m/s	Rear tire velocity,z-component

Set the geometry representation to use the CAD Kernel

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Advanced** section.

3 From the **Geometry representation** list, choose **CAD kernel**.

Insert a geometry sequence that contains the geometry and the selections needed for the boundary selections of the individual surfaces in the geometry.

4 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

5 Browse to the model's Application Libraries folder and double-click the file `sports_car_geom_sequence.mph`.

6 In the **Geometry** toolbar, click **Build All**.

7 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Browse Materials**.

MATERIAL BROWSER

1 In the **Material Browser** window, select **Built-in>Air** in the tree.

2 Click **Add to Component**.

3 Click **Done**.

Set the wall treatment to automatic so that wall functions are used where the viscous sublayer is not resolved by the boundary layer mesh.

LES RBVM (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **LES RBVM (spf)**.

2 In the **Settings** window for **LES RBVM**, locate the **Turbulence** section.

3 From the **Wall treatment** list, choose **Automatic**.

4 Locate the **Domain Selection** section. Click **Clear Selection**.

5 Select Domain 1 only.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `U0`.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 285 only.

Wall 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundaries 2, 4, and 5 only.

3 In the **Settings** window for **Wall**, locate the **Boundary Condition** section.

4 From the **Wall condition** list, choose **Slip**.

Wall 3

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Wall**, click to expand the **Wall Movement** section.

4 Select the **Sliding wall** check box.

5 Specify the $\mathbf{u}_w$ vector as

U0	x
0	y
0	z

Wall 4

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 In the **Settings** window for **Wall**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Front wheel boundaries**.

4 Locate the **Wall Movement** section. From the **Translational velocity** list, choose **Manual**.

5 Specify the $\mathbf{u}_{tr}$ vector as

Uwx	x
0	y
Uwz_front	z

Wall 5

1 Right-click **Wall 4** and choose **Duplicate**.

2 In the **Settings** window for **Wall**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Rear wheel boundaries**.

4 Locate the **Wall Movement** section. Specify the $\mathbf{u}_{tr}$ vector as

Uwx	x
0	y
Uwz_rear	z

Set the boundary conditions for the potential flow equation in the Laplace Equation interface.

LAPLACE EQUATION (LPEQ)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laplace Equation (lpeq)**.

2 In the **Settings** window for **Laplace Equation**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

Flux/Source 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Flux/Source**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Flux/Source**, locate the **Boundary Flux/Source** section.

4 In the *g* text field, type `-U0`.

Dirichlet Boundary Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Dirichlet Boundary Condition**.

2 Select Boundary 285 only.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Calibrate for** list, choose **Fluid dynamics**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.5`.

6 In the **Minimum element size** text field, type `0.05`.

7 In the **Maximum element growth rate** text field, type `1.11`.

Size 1

1 In the **Model Builder** window, click **Size 1**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Finer**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `0.5`.

7 Select the **Minimum element size** check box.

8 In the associated text field, type `0.05`.

9 Select the **Maximum element growth rate** check box.

Size 2

1 In the **Model Builder** window, click **Size 2**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Finer**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `0.3`.

7 Select the **Minimum element size** check box.

8 In the associated text field, type `0.05`.

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Mesh 1>Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

3 In the **Number of layers** text field, type `5`.

4 In the **Stretching factor** text field, type `1.2`.

5 From the **Thickness specification** list, choose **Automatic**.

6 In the **Thickness adjustment factor** text field, type `2.5`.

STUDY 1

Step 1: Stationary

Solve the potential flow equation first by deselecting the LES RBVM (spf) interface in the study settings.

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Equation form
LES RBVM (spf)		Automatic (Time dependent)

Generate the solver sequence and modify the solver so that it uses GMRES with smoothed aggregate algebraic multigrid (SAAMG) as preconditioner. This is an efficient and memory lean alternative for this type of problems.

Solution 1 (sol1)

1. In the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
3. In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node.
4. Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** and choose **Iterative**.
5. In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Iterative 1** node.
6. Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Iterative 1** and choose **Multigrid**.
7. In the **Settings** window for **Multigrid**, locate the **General** section.
8. From the **Solver** list, choose **Smoothed aggregation AMG**.

Step 1: Stationary

In the **Study** toolbar, click **Compute**.

RESULTS

Potential flow

1. In the **Model Builder** window, expand the **Results** node, then click **3D Plot Group 1**.
2. In the **Settings** window for **3D Plot Group**, type `Potential flow` in the **Label** text field.
3. Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Slice 1

1. In the **Model Builder** window, expand the **Potential flow** node, then click **Slice 1**.
2. In the **Settings** window for **Slice**, locate the **Expression** section.
3. In the **Expression** text field, type `sqrt(u2x^2+u2y^2+u2z^2)`.
4. Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

5 In the **Planes** text field, type 1.

Surface 1

1. In the **Model Builder** window, expand the **Results>Potential flow** node.
2. Right-click **Potential flow** and choose **Surface**.
3. In the **Settings** window for **Surface**, locate the **Expression** section.
4. In the **Expression** text field, type 1.
5. Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
6. From the **Color** list, choose **Gray**.
7. Click to expand the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1. Right-click **Surface 1** and choose **Selection**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. From the **Selection** list, choose **Car surfaces**.

Slice 1

1. In the **Model Builder** window, under **Results>Potential flow** click **Slice 1**.
2. In the **Settings** window for **Slice**, locate the **Coloring and Style** section.
3. From the **Color table** list, choose **JupiterAuroraBorealis**.
4. From the **Color table transformation** list, choose **Reverse**.
5. Click to expand the **Range** section. Select the **Manual color range** check box.
6. In the **Minimum** text field, type 0.
7. In the **Maximum** text field, type 50.

Transparency 1

1. Right-click **Slice 1** and choose **Transparency**.
2. In the **Potential flow** toolbar, click **Plot**.

   To get a better set of initial conditions for the velocity and pressure in the large eddy simulation, add a k-epsilon turbulent interface and solve it on the coarse mesh.

ADD PHYSICS

1. In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2. Go to the **Add Physics** window.
3. In the tree, select **Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, k-ε (spf)**.
4. Click **Add to Component 1** in the window toolbar.

5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

TURBULENT FLOW, K-ε 2 (SPF2)

1 In the **Settings** window for **Turbulent Flow, k-ε**, locate the **Domain Selection** section.

2 Click **Clear Selection**.

3 Select Domain 1 only.

Copy the boundary conditions in the LES RBVM-interface and paste them into the k-epsilon interface.

LES RBVM (SPF)

Inlet 1, Outlet 1, Wall 2, Wall 3, Wall 4, Wall 5

1 In the **Model Builder** window, under **Component 1 (comp1)>LES RBVM (spf)**, Ctrl-click to select **Inlet 1**, **Outlet 1**, **Wall 2**, **Wall 3**, **Wall 4**, and **Wall 5**.

2 Right-click and choose **Copy**.

TURBULENT FLOW, K-ε 2 (SPF2)

Inlet 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Turbulent Flow, k-ε 2 (spf2)** and choose **Paste Multiple Items**.

Set the initial values for the velocity and the pressure to be computed from the results of the potential flow simulation.

Initial Values 1

1 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

2 Specify the **u** vector as

u2x	x
u2y	y
u2z	z

3 In the p text field, type `spf.rho/2*((U0)^2-u2x^2-u2y^2-u2z^2)`.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **LES RBVM (spf)** and **Laplace Equation (lpeq)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

5 Click **Add Study** in the window toolbar.

6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

1 In the **Settings** window for **Stationary**, click to expand the **Values of Dependent Variables** section.

2 Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

3 From the **Study** list, choose **Study 1, Stationary**.

4 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1, Stationary**.

7 In the **Home** toolbar, click **Compute**.

RESULTS

Slice

1 In the **Model Builder** window, expand the **Results>Velocity (spf2)** node, then click **Slice**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 From the **Plane** list, choose **zx-planes**.

4 In the **Planes** text field, type `1`.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **JupiterAuroraBorealis**.

6 Click to expand the **Range** section. Select the **Manual color range** check box.

7 In the **Maximum** text field, type `50`.

8 Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Reverse**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Potential flow** node.

2 Right-click **Surface 1** and choose **Copy**.

Velocity (spf2)

1 In the **Model Builder** window, under **Results** click **Velocity (spf2)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** check box.

Surface 1

Right-click **Velocity (spf2)** and choose **Paste Surface**.

Transparency 1

1 In the **Model Builder** window, right-click **Slice** and choose **Transparency**.

2 In the **Velocity (spf2)** toolbar, click **Plot**.

Create a finer mesh, to be used in the LES computation, by duplicating Mesh 1 and modifying the settings.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Duplicate**.

MESH 2

Size 2

Select only the road surface.

1 In the **Model Builder** window, expand the **Mesh 2** node, then click **Size 2**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Size**, click **Build Selected**.

Size 3

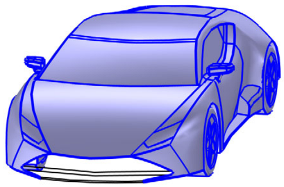
1 In the **Model Builder** window, right-click **Mesh 2** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Click the **Select Box** button in the **Graphics** toolbar.

5 Select Boundaries 10–14 and 16–284 only.



6 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

7 From the **Predefined** list, choose **Finer**.

8 Click the **Custom** button.

9 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

10 In the associated text field, type `0.15`.

11 Select the **Minimum element size** check box.

12 In the associated text field, type `0.02`.

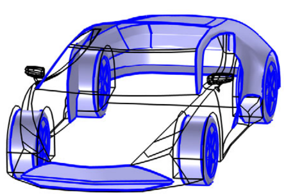
13 Click **Build Selected**.

Size 4

1 Right-click **Size 3** and choose **Duplicate**.

2 Click the **Clear Selection** button in the **Graphics** toolbar.

3 Select Boundaries 8, 9, 15, 28–90, 92–95, 98, 99, 197, 198, 205–260, and 262–284 only.



4 In the **Settings** window for **Size**, locate the **Element Size Parameters** section.

5 In the **Maximum element size** text field, type `0.2`.

6 Clear the **Minimum element size** check box.

7 In the **Maximum element size** text field, type `0.1`.

8 Click **Build Selected**.

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

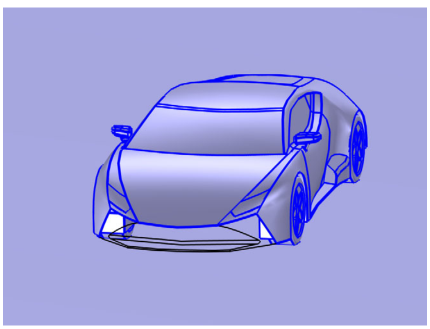
3 In the **Number of layers** text field, type `12`.

4 In the **Stretching factor** text field, type `1.12`.

5 From the **Thickness specification** list, choose **First layer**.

6 In the **Thickness** text field, type `0.0032`.

7 Select Boundaries 3, 10, 11, 13, 16–60, 69–80, 83–239, 248–259, 261, and 263–284 only.

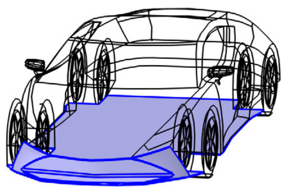


Boundary Layer Properties 2

1 In the **Mesh** toolbar, click **More Attributes** and choose **Boundary Layer Properties**.

2 Click the **Mesh Rendering** button in the **Graphics** toolbar.

3 Select Boundaries 6–9, 12, 14, and 15 only.



4 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

5 In the **Number of layers** text field, type `5`.

6 In the **Stretching factor** text field, type `1.15`.

7 From the **Thickness specification** list, choose **First layer**.

8 In the **Thickness** text field, type `0.0025`.

9 Click **Build Selected**.

Set the initial values in the LES RBVM (spf) interface to correspond to the velocity and pressure calculated with the k-epsilon turbulence model in the previous step.

LES RBVM (SPF)

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>LES RBVM (spf)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Specify the **u** vector as

u3	x
v3	y
w3	z

4 In the *p* text field, type `p2`.

ROOT

In the **Home** toolbar, click **Windows** and choose **Add Study**.

ADD STUDY

1. Go to the **Add Study** window.
2. Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Time Dependent**.
3. Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Laplace Equation (lpeq)** and **Turbulent Flow, k-ε 2 (spf2)**.
4. Click **Add Study** in the window toolbar.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Time Dependent

1. In the **Settings** window for **Time Dependent**, click to expand the **Values of Dependent Variables** section.
2. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
3. From the **Study** list, choose **Study 2, Stationary**.
4. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
5. From the **Method** list, choose **Solution**.
6. From the **Study** list, choose **Study 2, Stationary**.
7. Locate the **Study Settings** section. In the **Output times** text field, type `range(0,0.1,0.6), range(0.6,0.002,0.7)`.

Solution 3 (sol3)

1. In the **Study** toolbar, click **Show Default Solver**.

   Modify the time stepping, so that the relevant eddies are properly resolved in time, by restricting the maximum time step.
2. In the **Model Builder** window, expand the **Solution 3 (sol3)** node.
3. In the **Model Builder** window, under **Study 3>Solver Configurations>Solution 3 (sol3)** click **Time-Dependent Solver 1**.
4. In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
5. From the **Steps taken by solver** list, choose **Free**.
6. Select the **Initial step** check box.

7 In the associated text field, type `0.00001`.

8 From the **Maximum step constraint** list, choose **Constant**.

9 In the **Maximum step** text field, type `.001`.

10 In the **Study** toolbar, click **= Compute**.

RESULTS

Velocity (spf)

Change the settings so that the plots are only updated when requested. The plotting takes a while for such large data sets, so when building the plot sequence it is a good idea to turn off the automatic updating of the plots.

1 In the **Model Builder** window, click **Results**.

2 In the **Settings** window for **Results**, locate the **Update of Results** section.

3 Select the **Only plot when requested** check box.

Slice

1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Slice**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 From the **Plane** list, choose **zx-planes**.

4 In the **Planes** text field, type `1`.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **JupiterAuroraBorealis**.

6 From the **Color table transformation** list, choose **Reverse**.

7 Click to expand the **Range** section. Select the **Manual color range** check box.

8 In the **Maximum** text field, type `50`.

Transparency 1

Right-click **Slice** and choose **Transparency**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Velocity (spf2)** node.

2 Right-click **Surface 1** and choose **Copy**.

Velocity (spf)

1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** check box.

Surface 1

1 Right-click **Velocity (spf)** and choose **Paste Surface**.

Create another plot group and make a plot like in Figure 8

Car with velocity slices

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 3/Solution 3 (sol3)**.
4 In the **Label** text field, type `Car with velocity slices`.

Slice 1

1 Right-click **Car with velocity slices** and choose **Slice**.
2 In the **Settings** window for **Slice**, locate the **Plane Data** section.
3 From the **Plane** list, choose **zx-planes**.
4 In the **Planes** text field, type `1`.
5 Locate the **Coloring and Style** section. From the **Color table** list, choose **JupiterAuroraBorealis**.

Slice 2

1 Right-click **Slice 1** and choose **Duplicate**.
2 In the **Settings** window for **Slice**, locate the **Plane Data** section.
3 From the **Plane** list, choose **xy-planes**.
4 From the **Entry method** list, choose **Coordinates**.
5 In the **z-coordinates** text field, type `.05`.
6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.
7 In the **Car with velocity slices** toolbar, click **Plot**.

Transparency 1

1 In the **Model Builder** window, expand the **Car with velocity slices** node.
2 Right-click **Slice 1** and choose **Transparency**.

Slice 2

1 In the **Settings** window for **Slice**, click to expand the **Title** section.
2 From the **Title type** list, choose **None**.

Tires

1 In the **Model Builder** window, right-click **Car with velocity slices** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Tires` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `1`.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 Right-click **Tires** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Tires**.

Material Appearance 1

1 In the **Model Builder** window, right-click **Tires** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Rubber**.

5 From the **Color** list, choose **Black**.

Car with velocity slices

1 In the **Model Builder** window, under **Results** click **Car with velocity slices**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** check box.

Wheel Bays

1 Right-click **Car with velocity slices** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Wheel Bays` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 Right-click **Wheel Bays** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Wheel bays**.

Wheel Bays

1 In the **Model Builder** window, click **Wheel Bays**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Custom**.

5 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

6 Click **Define custom colors**.

7 Set the RGB values to 83, 84, and 107, respectively.

8 Click **Add to custom colors**.

9 Click **Show color palette only** or **OK** on the cross-platform desktop.

Windows

1 In the **Model Builder** window, right-click **Tires** and choose **Duplicate**.

2 In the **Model Builder** window, click **Tires 1**.

3 In the **Settings** window for **Surface**, type `Windows` in the **Label** text field.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 In the **Model Builder** window, click **Selection 1**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Windows**.

Material Appearance 1

1 In the **Model Builder** window, click **Material Appearance 1**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Material type** list, choose **Water**.

Headlights

1 In the **Model Builder** window, right-click **Windows** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Headlights` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 In the **Model Builder** window, expand the **Headlights** node, then click **Selection 1**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Headlights**.

Material Appearance 1

1 In the **Model Builder** window, click **Material Appearance 1**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Material type** list, choose **Plastic (shiny)**.

4 From the **Color** list, choose **White**.

Rims

1 In the **Model Builder** window, right-click **Headlights** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Rims` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 In the **Model Builder** window, expand the **Rims** node, then click **Selection 1**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Rims**.

Material Appearance 1

1 In the **Model Builder** window, click **Material Appearance 1**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Material type** list, choose **Rose gold**.

Body

1 In the **Model Builder** window, right-click **Rims** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Body` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 In the **Model Builder** window, expand the **Body** node, then click **Selection 1**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Body**.

Material Appearance 1

1 In the **Model Builder** window, click **Material Appearance 1**.

2 In the **Settings** window for **Material Appearance**, in the **Graphics** window toolbar, click ▾ next to **Scene Light**, then choose **Ambient Occlusion**.

3 In the **Graphics** window toolbar, click ▾ next to **Scene Light**, then choose **Outdoor Environment**.

Rear Lights

1 In the **Model Builder** window, right-click **Body** and choose **Duplicate**.

2 In the **Model Builder** window, click **Body 1**.

3 In the **Settings** window for **Surface**, type `Rear Lights` in the **Label** text field.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 In the **Model Builder** window, click **Selection 1**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Rear lights**.

Material Appearance 1

1 In the **Model Builder** window, click **Material Appearance 1**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Material type** list, choose **Plastic (shiny)**.

4 From the **Color** list, choose **Red**.

5 In the **Car with velocity slices** toolbar, click **Plot**.

Air vents

1 In the **Model Builder** window, right-click **Wheel Bays** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Air vents` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 In the **Model Builder** window, expand the **Air vents** node, then click **Selection 1**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Air vents**.

Air vents

1 In the **Model Builder** window, click **Air vents**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Click **Define custom colors**.

4 Set the RGB values to 0, 0, and 0, respectively.

5 Click **Add to custom colors**.

6 Click **Show color palette only** or **OK** on the cross-platform desktop.

Wheel Bays 1

In the **Model Builder** window, right-click **Wheel Bays** and choose **Duplicate**.

Selection 1

1 In the **Model Builder** window, expand the **Air vents** node, then click **Selection 1**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Air vents**.

Air vents

1 In the **Model Builder** window, click **Air vents**.

2 In the **Settings** window for **Surface**, type `Air vents` in the **Label** text field.

3 In the **Car with velocity slices** toolbar, click **Plot**.

4 Locate the **Coloring and Style** section. Click **Define custom colors**.

5 Set the RGB values to 0, 0, and 0, respectively.

6 Click **Add to custom colors**.

7 Click **Show color palette only** or **OK** on the cross-platform desktop.

8 In the **Car with velocity slices** toolbar, click **Plot**.

Streamlines

1 In the **Model Builder** window, right-click **Car with velocity slices** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Streamlines` in the **Label** text field.

3 In the **Model Builder** window, expand the **Streamlines** node.

Slice 1, Slice 2

1 In the **Model Builder** window, under **Results>Streamlines**, Ctrl-click to select **Slice 1** and **Slice 2**.

2 Right-click and choose **Disable**.

Streamline 1

1 In the **Model Builder** window, right-click **Streamlines** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 From the **Positioning** list, choose **Starting-point controlled**.

4 From the **Entry method** list, choose **Coordinates**.

5 In the **x** text field, type `-8`.

6 In the **y** text field, type `0`.

7 In the **z** text field, type `range(0,2/39,2)`.

8 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

9 In the **Tube radius expression** text field, type `.005`.

10 Select the **Radius scale factor** check box.

11 Click to expand the **Advanced** section. Clear the **Allow backward time integration** check box.

Color Expression 1

1. Right-click **Streamline 1** and choose **Color Expression**.
2. In the **Settings** window for **Color Expression**, click to expand the **Range** section.
3. Select the **Manual color range** check box.
4. In the **Maximum** text field, type `70`.
5. Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Streamline 2

1. In the **Model Builder** window, under **Results>Streamlines** right-click **Streamline 1** and choose **Duplicate**.
2. In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
3. In the **x** text field, type `4.31`.
4. In the **y** text field, type `range(-0.8,0.041025641025641026,0.8)`.
5. In the **z** text field, type `0.88`.
6. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Streamline 1**.

Streamline 3

1. Right-click **Streamline 2** and choose **Duplicate**.
2. In the **Model Builder** window, click **Streamline 3**.
3. In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
4. In the **z** text field, type `0.1`.

Streamline 4

1. Right-click **Streamline 3** and choose **Duplicate**.
2. In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
3. In the **x** text field, type `-8`.
4. In the **y** text field, type `range(-1.5,0.043478260869565216,1.5)`.
5. In the **Streamlines** toolbar, click **Plot**.

Surface 10

In the **Model Builder** window, right-click **Streamlines** and choose **Surface**.

Selection 1

1. In the **Model Builder** window, right-click **Surface 10** and choose **Selection**.
2. Select Boundary 3 only.

Material Appearance 1

1 In the **Model Builder** window, right-click **Surface 10** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Rubber**.

5 From the **Color** list, choose **Black**.

6 In the **Streamlines** toolbar, click **Plot**.

Visualize the wheel boundary velocity by duplicating the Car with velocity slice plot group, and edit the plot group sequence.

Wheel Velocity

1 In the **Model Builder** window, right-click **Car with velocity slices** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Wheel Velocity` in the **Label** text field.

Slice 1

1 In the **Model Builder** window, expand the **Wheel Velocity** node, then click **Slice 1**.

2 In the **Settings** window for **Slice**, locate the **Coloring and Style** section.

3 Clear the **Color legend** check box.

Slice 2

In the **Model Builder** window, right-click **Slice 2** and choose **Disable**.

Surface 10

1 In the **Model Builder** window, right-click **Wheel Velocity** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `Uwx`.

Selection 1

1 Right-click **Surface 10** and choose **Selection**.

2 Select Boundary 3 only.

3 Click the **Zoom Out** button in the **Graphics** toolbar.

4 Click the **Go to XZ View** button in the **Graphics** toolbar.

Rims, Tires

1 In the **Model Builder** window, under **Results>Wheel Velocity**, Ctrl-click to select **Tires** and **Rims**.

2 Right-click and choose **Disable**.

Surface 11

In the **Model Builder** window, right-click **Wheel Velocity** and choose **Surface**.

Selection 1

1. In the **Model Builder** window, right-click **Surface 11** and choose **Selection**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. From the **Selection** list, choose **Front wheel boundaries**.

Surface 11

1. In the **Model Builder** window, click **Surface 11**.
2. In the **Settings** window for **Surface**, click **Insert Expression (Ctrl+Space)** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>Uwx - Tire velocity, x-component - m/s**.
3. Locate the **Expression** section. In the **Expression** text field, type `sqrt(Uwx^2+Uwz_front^2)`.
4. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 10**.
5. In the **Wheel Velocity** toolbar, click **Plot**.

Surface 12

1. In the **Model Builder** window, right-click **Surface 11** and choose **Duplicate**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `sqrt(Uwx^2+Uwz_rear^2)`.
4. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 11**.

Selection 1

1. In the **Model Builder** window, expand the **Surface 12** node, then click **Selection 1**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. From the **Selection** list, choose **Rear wheel boundaries**.

Wheel Velocity

1. In the **Model Builder** window, under **Results** click **Wheel Velocity**.
2. In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.
3. From the **Title type** list, choose **Manual**.
4. In the **Title** text area, type `Velocity magnitude (m/s)`.
5. In the **Wheel Velocity** toolbar, click **Plot**.

   To calculate the total drag coefficient, define an integration operator over the surfaces of the car and the wheels, and define variables to calculate the effective exposed area and the drag coefficient, Cd, by following the instructions below. The drag coefficient computed by the pressure only is defined as Cdp. However, we also define a parameter, Cdstress, that includes the viscous stress contributions that can be calculated from the automatic wall functions that are applied wherever the boundary layer is not fully resolved.

DEFINITIONS

Integration 1 (intop1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** node.

2 Right-click **Component 1 (comp1)>Definitions** and choose **Nonlocal Couplings> Integration**.

3 In the **Settings** window for **Integration**, locate the **Source Selection** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Car surfaces**.

Variables 1

1 In the **Model Builder** window, click **Variables 1**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
A	intop1(max(0,spf.nxmesh))	m^2	
Cdp	2/(A*U0^2*1.2[kg/m^3])*intop1(p* spf.nxmesh)		
Cdstress	2/(A*U0^2*1.2[kg/m^3])*intop1(p* spf.nxmesh+spf.rho*spf.u_tau^2* spf.Utx/nojac(sqrt(spf.Utx^2+ spf.Uty^2+spf.Utz^2+eps)))		

STUDY 3

In the **Study** toolbar, click **Update Solution**.

RESULTS

Global Evaluation 1

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 3/Solution 3 (sol3)**.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (s)** list, choose **0.6 (2)**, **0.602**, **0.604**, **0.606**, **0.608**, **0.61**, **0.612**, **0.614**, **0.616**, **0.618**, **0.62**, **0.622**, **0.624**, **0.626**, **0.628**, **0.63**, **0.632**, **0.634**, **0.636**, **0.638**, **0.64**, **0.642**, **0.644**, **0.646**, **0.648**, **0.65**, **0.652**, **0.654**, **0.656**, **0.658**, **0.66**, **0.662**, **0.664**, **0.666**, **0.668**, **0.67**, **0.672**, **0.674**, **0.676**, **0.678**, **0.68**, **0.682**, **0.684**, **0.686**, **0.688**, **0.69**, **0.692**, **0.694**, **0.696**, **0.698**, and **0.7**.

6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`Cdp`	1	
`Cdstress`	1	

7 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Average**.

8 Click **Evaluate**.

Global Evaluation 2

1 In the **Model Builder** window, right-click **Global Evaluation 1** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data Series Operation** section.

3 From the **Transformation** list, choose **None**.

4 Click next to **Evaluate**, then choose **New Table**.

TABLE

1 Go to the **Table** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Drag Coefficient

1 In the **Model Builder** window, under **Results>1D Plot Group 11** click **Table Graph 1**.

2 In the **1D Plot Group 11** toolbar, click **Plot**.

3 In the **Settings** window for **Table Graph**, type `Drag Coefficient` in the **Label** text field.

4 Click to expand the **Legends** section. Select the **Show legends** check box.

5 In the **1D Plot Group 11** toolbar, click **Plot**.