Created in COMSOL Multiphysics 6.2

Large Eddy Simulation of a 3D Hill Geometry

Introduction

Flow prediction in the case of separating flows from blunt surfaces are computationally challenging. Such scenarios are frequently encountered in a variety of engineering simulations of external and internal flows, for instance, flow around highly-loaded swept wings and duct flows with surface deformations and curved geometries. They demonstrate a high sensitivity to upstream flow and turbulence conditions for detachment and reattachment phenomena, both spatially and temporally. Moreover, three-dimensional flows admit convoluted separation patterns and leaner recirculation regions.

The present model demonstrates the application of the Large Eddy Simulation (LES) method with automatic wall modeling to predict flow quantities for a well-studied laboratory experiment of flow around an axisymmetric 3D hill feature in a duct system; see Ref. 1 and Ref. 2. The Reynolds number of the flow, Re_H is equal to 130,000 based on the hill height, H. Synthetic turbulence inlet condition option is enabled to generate isotropic turbulence at the inlet boundary. The nondimensional quantity, coefficient of pressure, denoted as C_p, is validated with those reported in the experiments. Velocity fields at the leeward side and in the wake region of the hill are compared with those obtained in the experiments.

Model Definition

The present section describes the details of the geometry and the flow problem setup. Details required to build the geometry, mesh the domain of interest, and apply suitable solution postprocessing are presented.

GEOMETRY

The domain of interest is a block of dimensions $19.8H$, $3.2H$, and $11.7H$ in the x, y, and z directions. It represents a long rectangular duct with large aspect ratio. The base of the axisymmetric hill is located in the plane defined by $y = 0$. It is centered at a distance of $4.1H$ and $5.85H$ along the x and z directions. The hill has a radius of $2H$ at the base and the crest is located at $y = H$. The hill profile, h, as a function of the hill radius, r, is given by

$$h(r) = \frac{-H}{6.04844}\left[J_0(3.1962)I_0\left(\frac{3.1962\,\mathrm{r}}{2H}\right) - I_0(3.1962)J_0\left(\frac{3.1962\,\mathrm{r}}{2H}\right)\right]. \tag{1}$$

Here, J_0 is the Bessel function of the first kind and I_0 is the modified Bessel function of the first kind. The final geometry is shown in Figure 1.

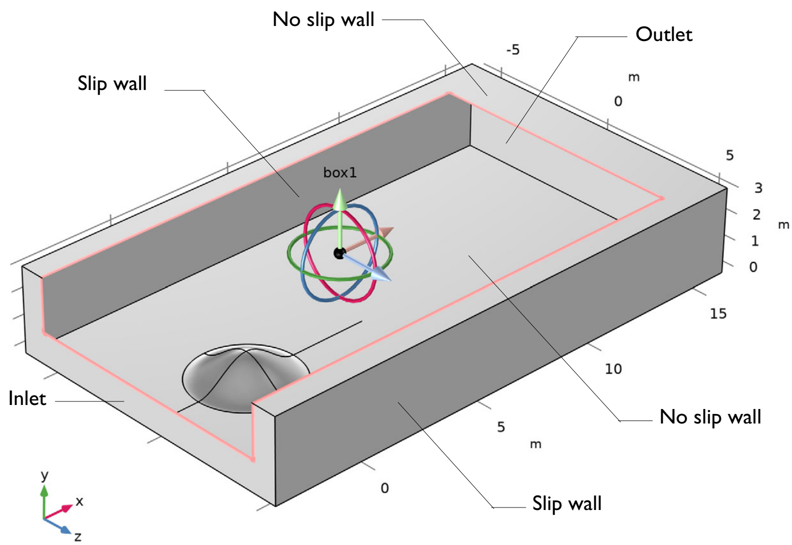

Figure 1: Geometry setup of the model is shown with a cut-box to reveal the hill profile. Boundary conditions for the LES problem are also annotated.

PHYSICS INTERFACE SETTINGS

In the model, $H = 1$ m, the density of the fluid, $\rho = 1$ kg/m^3, the velocity of the fluid, $U = 2$ m/s, and the dynamic viscosity of the fluid is calculated from the Reynolds number as $\mu = \rho UH/(\mathrm{Re}_H)$ kg/(m·s).

As a first step, the potential flow problem which describes an incompressible, irrotational flow is solved. The potential flow problem takes the form of the Laplace equation for the velocity potential, φ , as

$$\nabla \cdot \nabla \varphi = 0. \tag{2}$$

The boundary conditions are:

$$\mathbf{n} \cdot \nabla\varphi = -U \quad \text{at the inlet boundary,} \tag{3}$$

$$\varphi = 0 \quad \text{at the outlet boundary,} \tag{4}$$

$$\mathbf{n} \cdot \nabla\varphi = 0 \quad \text{at remaining boundaries.} \tag{5}$$

The LES problem uses the Residual Based Variational Multiscale (RBVM) LES model with automatic wall treatment. The initial values of the velocity and pressure fields for the LES problem are obtained from the potential flow solution as

$$\mathbf{u}_{\text{init}} = \nabla\varphi, \tag{6}$$

$$p_{\text{init}} = \frac{\rho}{2}(U^2 - (\nabla\varphi \cdot \nabla\varphi)). \tag{7}$$

Figure 1 shows the boundary conditions as applied to the problem. A normal inflow velocity condition is prescribed at the inlet. Since the inlet boundary is located close to the hill feature, it has the prescribed velocity profile of the $(1/7)$th power law for turbulent flows, given by

$$U_0 = \frac{U}{0.75644}\left|\frac{y}{3.2H}\right|^{\frac{1}{7}}\left|1 - \left(\frac{y}{3.2H}\right)\right|^{\frac{1}{7}} \tag{8}$$

The velocity profile is adjusted to ensure that the mass flow rate is equivalent to that produced by a uniform inlet velocity of value U. The synthetic turbulence option at the inlet is enabled and the turbulence length scale is specified as $0.1H$. The number of Fourier modes is chosen to be 600. The outlet condition has the “suppress backflow” option disabled. This permits backflow, which is a common feature in turbulent wake regions.

MESHING

Element sizes close to the hill are chosen so as to obtain a good spatial resolution for both potential flow and LES solutions.

In the case of the potential flow solution, a coarse tetrahedral mesh is generated in the volume, see Figure 2.

On the other hand, the LES simulation requires high resolution mesh elements to transport turbulence generated at the inlet, obtain a good flow representation in the boundary layers, and capture flow features in the wake of the hill. In this context, triangular elements for the hill boundary and mapped quadrilateral elements for the boundary at $y = 0$ are constructed. Further on, a swept mesh for the volume is generated with appropriate element spacing for the boundary layers, see Figure 3.

The first boundary layer element is of thickness, measured in wall units, equal to $y^+ \approx 4$. The distance from the wall in wall units, y^+, is computed using the friction velocity, u_τ, the kinematic viscosity, ν, the wall shear stress, τ_w, and the skin-friction coefficient, c_f, as

$$y^{+} = \frac{y u_{\tau}}{\nu}, \quad u_{\tau} = \sqrt{\frac{\tau_{\mathrm{w}}}{\rho}}, \quad \tau_{\mathrm{w}} = c_{\mathrm{f}}\left(\frac{1}{2}\rho U^{2}\right). \tag{9}$$

The skin-friction coefficient as a function of the Reynolds number for channel flows is taken from Ref. 5.

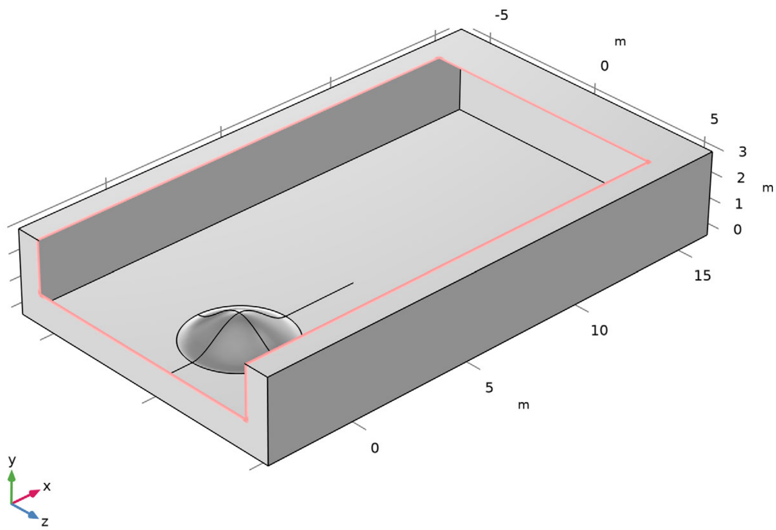

Figure 2: Mesh for the stationary potential flow solution is shown with a cut-box to reveal the surface mesh on the hill profile.

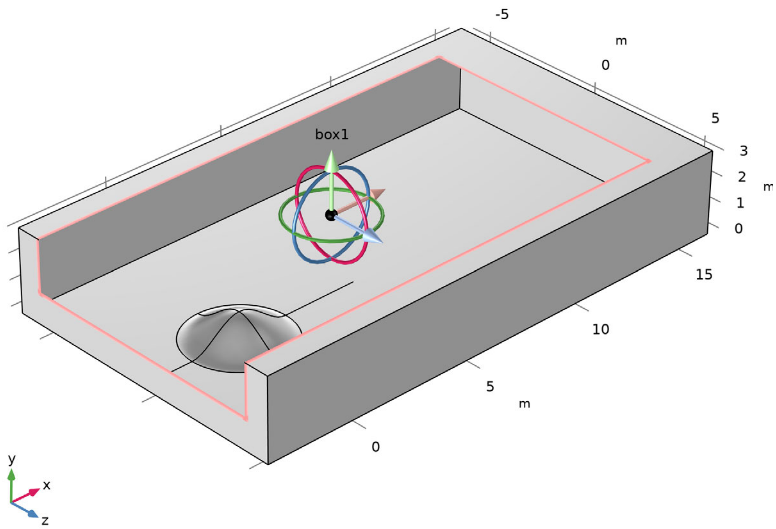

Figure 3: Mesh for the time dependent LES solution is shown with a cut-box to reveal the surface mesh on the hill profile.

STUDY

The potential flow problem is solved with a stationary study step. The LES problem is solved with a time-dependent study step with the end time equal to $40H/U$, which is the average time taken for a fluid parcel to traverse nearly twice the length of the domain. Four hundred solution time steps are recorded in the latter half of the time interval. These are time-averaged and used for comparison with experiments.

Results and Discussion

The first quantity of postprocess is the coefficient of pressure, given by

$$C_p = \frac{p}{\frac{1}{2}\rho U^2} \tag{10}$$

A contour plot of C_p on the hill surface is shown in Figure 4. The simulation correctly predicts the zone of high suction pressure at the crest of the hill and the zone of the low suction pressure at the base. These zones are slightly larger in the simulation when compared to experiments in Ref. 1. However, they show good correspondence to simulations reported in Ref. 4.

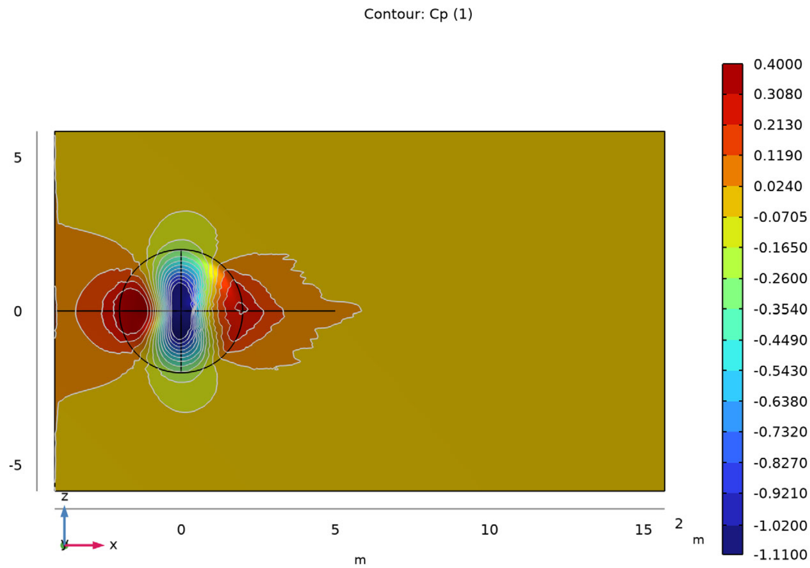

Figure 4: Contour plot of C_p on the hill surface.

The nondimensional vorticity flux vector, given by the quantity $-(\mathbf{n} \times \nabla C_p)$, is plotted in Figure 5 and shows good qualitative correlation to experimental measurements in Ref. 1. It demonstrates nonuniformity of vorticity on the hill which is observed in the experiments in the form of streamwise and spanwise variations in the separation pattern on the hill surface.

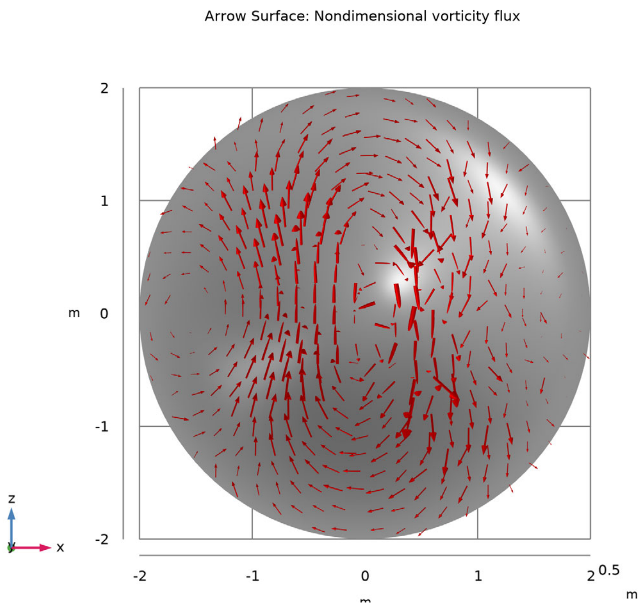

Figure 5: Vector plot of nondimensional vorticity flux on the hill surface.

The velocity field on the plane defined by $z = 0$ is visualized in Figure 6. The flow is shown to accelerate at the crest of the hill where the suction pressure is the highest. The adverse pressure gradient in the leeward side of the hill results in a back-flow region, shown here in magenta color. The simulation over-predicts the length and under-predicts the height of the separation zone at the hill base as reported in experiments (see Ref. 2).

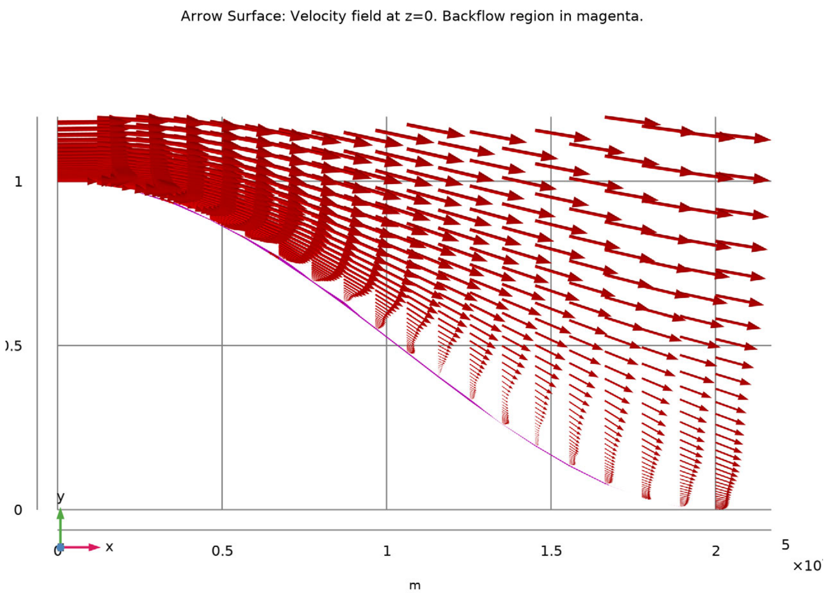

Figure 6: Vector plot of velocity on the plane defined by z = 0. The region of back-flow is shown in magenta.

Similarly, the velocity field in the wake region on the plane defined by $x = 3.69H$ is shown in Figure 7. The recirculation bubble is seen attached to the bottom surface as a result of fluid roll-up from the sides of the hill and the down-wash from the top of the hill. The position of the recirculation region is predicted correctly, however, the width is under-predicted when compared to Ref. 4.

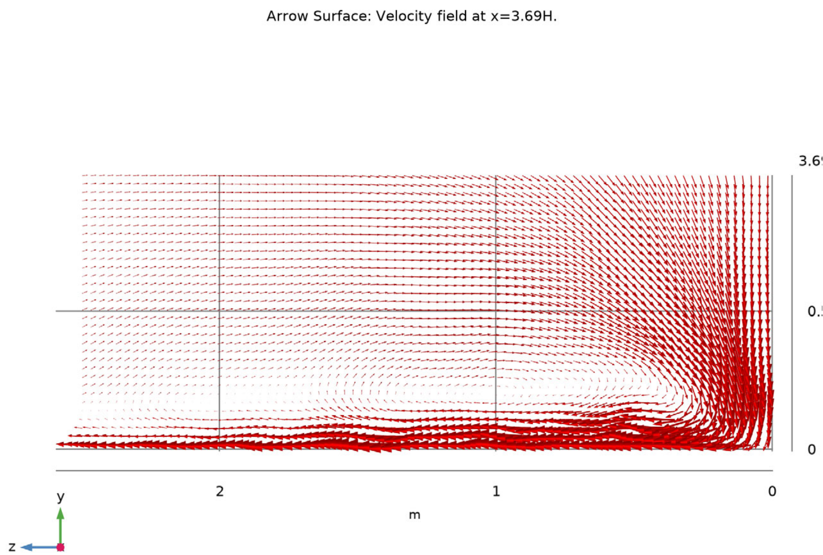

Figure 7: Vector plot of velocity on the plane defined by x = 3.69H.

Finally, the unsteady nature of the flow-field is demonstrated in the plot of instantaneous streamlines, corresponding to the end of the time interval, given in Figure 8. Also shown in the plot is the velocity magnitude on the plane defined by $z = 0$.

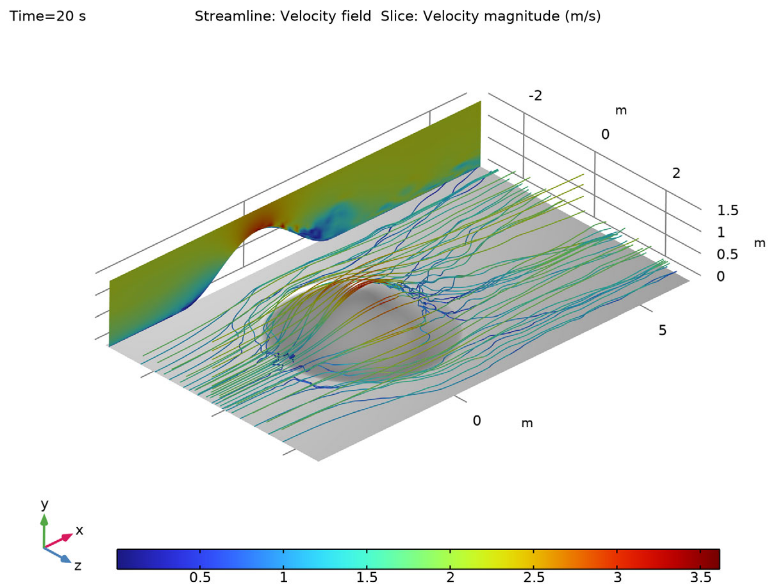

Figure 8: Instantaneous streamlines corresponding to the solution at the end of the simulation. Also shown is the velocity magnitude at the domain mid-plane defined by z = 0.

References

1. R.L. Simpson, C.H. Long, and G. Byun, "Study of vortical separation from an axisymmetric hill," *Int. J. Heat Fluid Flow*, vol. 23, pp. 582–591, 2002.

2. G. Byun and R.L. Simpson, "Structure of Three-Dimensional Separated Flow on an Axisymmetric Bump," *AIAA Journal,* vol. 44, no. 5, 2006.

3. L. Davidson, "Using isotropic synthetic fluctuations as inlet boundary conditions for unsteady simulations," *Adv. Appl. Fluid Mech.*, vol. 1, no. 1, pp. 1–35, 2007.

4. T. Persson, M. Liefvendahl, R.E. Bensow, and C. Fureby, "Numerical investigation of the flow over an axisymmetric hill using LES, DES, and RANS," *J. Turbul.,* vol. 7, no. 4, 2006.

5. S. Pope, "Turbulent Flows", *Cambridge University Press,* 2000.

Application Library path: `CFD_Module/Single-Phase_Flow/les_3d_hill`

Modeling Instructions

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 **Fluid Flow>Single-Phase Flow>Potential Flow>Incompressible Potential Flow (ipf)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Stationary**.

8 Click **Done**.

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
H	1[m]	1 m	Hill height
ReH	1.3e5	1.3E5	Reynolds number based on H
rho	1[kg/m^3]	1 kg/m³	Density of the fluid
U	2[m/s]	2 m/s	Velocity of the fluid

Name	Expression	Value	Description
mu	rho*U*H/ReH	1.5385E-5 kg/(m·s)	Viscosity of the fluid
LTin	0.1*H	0.1 m	Turbulence length scale at the inlet

DEFINITIONS

Create an analytic function that defines the hill profile.

Axisymmetric Hill Profile

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Analytic**.

2 In the **Settings** window for **Analytic**, type `Axisymmetric Hill Profile` in the **Label** text field.

3 In the **Function name** text field, type `hill`.

4 Locate the **Definition** section. In the **Expression** text field, type `-(besselj(0,3.1962)*besseli(0, 3.1962*x/(2*H))-besseli(0,3.1962)*besselj(0, 3.1962*x/(2*H)))*H/6.04844`.

5 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	x	0	2*H	0	

GEOMETRY 1

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `19.8*H`.

4 In the **Depth** text field, type `3.2*H`.

5 In the **Height** text field, type `11.7*H`.

6 Locate the **Position** section. In the **x** text field, type `-4.1*H`.

7 In the **z** text field, type `-11.7*H/2`.

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

4 Click **Go to Plane Geometry**.

Work Plane 1 (wp1)>Square 1 (sq1)

1 In the **Work Plane** toolbar, click **Square**.
2 In the **Settings** window for **Square**, locate the **Size** section.
3 In the **Side length** text field, type `4.5*H`.
4 Locate the **Position** section. From the **Base** list, choose **Center**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `19.8*H`.
4 In the **Height** text field, type `4.5*H`.
5 Locate the **Position** section. In the **yw** text field, type `-2.25*H`.
6 In the **xw** text field, type `-4.1*H`.

Line Segment 1 (ls1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **More Primitives> Line Segment**.
2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.
3 Click to clear the **Activate Selection** toggle button for **Start vertex**.
4 From the **Specify** list, choose **Coordinates**.
5 In the **x** text field, type `-4.1*H`.
6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.
7 In the **x** text field, type `5*H`.
8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Work Plane 2 (wp2)

1 In the **Geometry** toolbar, click **Work Plane**.
2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3 From the **Plane** list, choose **zy-plane**.
4 Click **Go to Plane Geometry**.

Work Plane 2 (wp2)>Parametric Curve 1 (pc1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Parametric Curve**.
2 In the **Settings** window for **Parametric Curve**, locate the **Parameter** section.
3 In the **Maximum** text field, type `2*H`.

4 Locate the **Expressions** section. In the **xw** text field, type `s`.

5 In the **yw** text field, type `hill(s)`.

Work Plane 2 (wp2)>Line Segment 1 (ls1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 In the **xw** text field, type `2*H`.

Work Plane 2 (wp2)>Line Segment 2 (ls2)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 In the **yw** text field, type `H`.

Work Plane 2 (wp2)>Convert to Solid 1 (csol1)

1 In the **Work Plane** toolbar, click **Conversions** and choose **Convert to Solid**.

2 In the **Settings** window for **Convert to Solid**, locate the **Input** section.

3 Click the **Paste Selection** button for **Input objects**.

4 In the **Paste Selection** dialog box, type `ls1 ls2 pc1` in the **Selection** text field.

5 Click **OK**.

Revolve 1 (rev1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 2 (wp2)** and choose **Revolve**.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

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

3 Click the **Paste Selection** button for **Objects to add**.

4 In the **Paste Selection** dialog box, type `blk1 wp1 ls1` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Difference**, locate the **Difference** section.

7 Click the **Paste Selection** button for **Objects to subtract**.

8 In the **Paste Selection** dialog box, type `rev1` in the **Selection** text field.

9 Click **OK**.

Mesh Control Edges 1 (mce1)

Designate as mesh control entities those edges that are only used in mesh creation.

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Edges**.

2 In the **Settings** window for **Mesh Control Edges**, locate the **Input** section.

3 Click the **Paste Selection** button for **Edges to include**.

4 In the **Paste Selection** dialog box, type `5, 9, 15-17, 19, 29-31, 33` in the **Selection** text field.

5 Click **OK**.

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

DEFINITIONS

Create an edge selection for plotting C_p values.

Edge for Cp plot

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Edge for Cp plot` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.

4 Select Edges 5, 13, 18, and 19 only.

INCOMPRESSIBLE POTENTIAL FLOW (IPF)

Set up the Incompressible Potential Flow interface to solve for the velocity potential and to get an initial approximation for the velocity and pressure. Use a linear approximation to reduce the memory requirements.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Incompressible Potential Flow (ipf)**.

2 In the **Settings** window for **Incompressible Potential Flow**, locate the **Pressure** section.

3 In the U_{scale} text field, type `U`.

4 Click to expand the **Discretization** section. From the **Velocity potential** list, choose **Linear**.

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Incompressible Potential Flow (ipf)** click **Fluid Properties 1**.

2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.

3 From the ρ list, choose **User defined**. In the associated text field, type `rho`.

Velocity 1

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

2 Select Boundary 1 only.

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

4 In the U_{in} text field, type `U`.

Open Boundary 1

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

2 Select Boundary 10 only.

LES RBVM (SPF)

Set up the Large Eddy Simulation problem.

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**.

Fluid Properties 1

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

2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.

3 From the ρ list, choose **User defined**. In the associated text field, type `rho`.

4 From the μ list, choose **User defined**. In the associated text field, type `mu`.

Initial Values 1

Use the stationary solution of the potential-flow problem to compute the initial values.

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

Initial Values 2

1 In the **Model Builder** window, click **Initial Values 2**.

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

3 Specify the **u** vector as

ipf.u	x
ipf.v	y
ipf.w	z

4 In the p text field, type `ipf.p`.

Wall 2

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

Use a slip wall condition on the lateral sides of the channel.

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

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

4 Select Boundaries 3 and 4 only.

Inlet 1

Specify the inlet velocity profile based on the (1/7)th power law for turbulent flows. Include synthetic turbulence effects.

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 `U*abs(y/(3.2*H))^(1/7)*abs(1-(y/(3.2*H)))^(1/7)/0.75644`.

5 Select the **Include synthetic turbulence** check box.

6 Locate the **Turbulence Conditions** section. From the I_T list, choose **Low (0.01)**.

7 From the L_T list, choose **User defined**.

8 In the text field, type `LTin`.

9 In the N text field, type `600`.

Outlet 1

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

2 Select Boundary 10 only.

3 In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.

4 Clear the **Suppress backflow** check box.

MESH 1

Create a mesh suitable for the LES simulation.

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 In the table, clear the **Use** check boxes for **LES RBVM (spf)** and **Incompressible Potential Flow (ipf)**.

4 Locate the **Sequence Type** section. 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 `0.1222*H`.

6 In the **Minimum element size** text field, type `0.0390*H`.

Mapped 1

Create a boundary mesh with Mapped and Triangular meshes.

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

2 Select Boundaries 2, 11–13, and 16 only.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 4, 24, 36, and 38 only.

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

4 In the **Number of elements** text field, type `25`.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Edge 1 only.

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

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `25`.

6 In the **Element ratio** text field, type `2.5`.

7 From the **Growth rate** list, choose **Exponential**.

8 Select the **Reverse direction** check box.

9 Right-click **Distribution 2** and choose **Duplicate**.

Distribution 3

1 In the **Model Builder** window, click **Distribution 3**.

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

3 Click **Clear Selection**.

4 Select Edge 25 only.

5 Locate the **Distribution** section. Clear the **Reverse direction** check box.

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

2 Select Boundaries 7, 9, and 14 only.

Copy Face 1

1 In the **Mesh** toolbar, click **Copy** and choose **Copy Face**.

2 Select Boundaries 7, 9, and 14 only.

3 In the **Settings** window for **Copy Face**, locate the **Destination Boundaries** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Boundaries 6, 8, and 15 only.

Swept 1

Sweep the boundary mesh to obtain the volume mesh.

In the **Mesh** toolbar, click **Swept**.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

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

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type 82.

5 In the **Element ratio** text field, type 82.

6 From the **Growth rate** list, choose **Exponential**.

7 Select the **Symmetric distribution** check box.

Free Tetrahedral 1

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

2 Right-click **Mesh 1** and choose **Build All**.

STUDY 1: STATIONARY POTENTIAL-FLOW SOLUTION

Solve for a stationary solution to the potential-flow problem. This solution is used in the initial value for the LES problem.

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

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

3 Clear the **Generate default plots** check box.

4 In the **Label** text field, type `Study 1: Stationary Potential-Flow Solution`.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1: Stationary Potential-Flow Solution** click **Step 1: Stationary**.

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

3 In the table, clear the **Solve for** check box for **LES RBVM (spf)**.

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

Disable the **Initial Values 2** node of the LES RBVM interface in the potential-flow study.

5 Select the **Modify model configuration for study step** check box.

6 In the tree, select **Component 1 (comp1)>LES RBVM (spf)>Initial Values 2**.

7 Click **Disable**.

ADD STUDY

Solve the time-dependent study for the LES problem.

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 box for **Incompressible Potential Flow (ipf)**.

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

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

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

STUDY 2: TIME-DEPENDENT LES SOLUTION

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

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

3 Clear the **Generate default plots** check box.

4 In the **Label** text field, type `Study 2: Time-Dependent LES Solution`.

5 Locate the **Study Settings** section. Select the **Store solution for all intermediate study steps** check box.

Step 1: Time Dependent

Set the end time of the time-dependent study to $40H/U = 20$ seconds. Store solutions in the range 10–20 s

1 In the **Model Builder** window, under **Study 2: Time-Dependent LES Solution** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,20*H/U,20*H/U) range(20*H/U,(20*H/U)/400,40*H/U)`.

4 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Study** list, choose **Study 1: Stationary Potential-Flow Solution, Stationary**.

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

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

8 From the **Study** list, choose **Study 1: Stationary Potential-Flow Solution, Stationary**.

Step 2: Combine Solutions

1 In the **Study** toolbar, click **Combine Solutions**.

2 In the **Settings** window for **Combine Solutions**, locate the **Combine Solutions Settings** section.

3 From the **Solution operation** list, choose **Remove solutions**.

4 From the **Selection** list, choose **Manual**.

5 In the **Index** text field, type `1 2`.

6 Select the **Clear source solution** check box.

Solution 2 (sol2)

Use automatic time-stepping to speed up the simulation.

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Maximum step constraint** list, choose **Automatic**.

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

ADD STUDY

Compute the average of the stored solutions from the previous step.

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.

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

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

STUDY 3: TIME-AVERAGED LES SOLUTION

1 In the **Settings** window for **Study**, type `Study 3: Time-Averaged LES Solution` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate convergence plots** check box.

3 Clear the **Generate default plots** check box.

Step 1: Combine Solutions

1 In the **Study** toolbar, click **Combine Solutions**.

2 In the **Settings** window for **Combine Solutions**, locate the **Combine Solutions Settings** section.

3 From the **Solution operation** list, choose **Weighted summation**.

4 From the **Solution** list, choose **Study 2: Time-Dependent LES Solution/Solution 2 (sol2)**.

5 In the **Expression** text field, type `1.0/401`.

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

STUDY 2: TIME-DEPENDENT LES SOLUTION

Remove the solution except for the last time step to reduce file size.

Step 3: Combine Solutions 2

1 In the **Study** toolbar, click **Combine Solutions**.

2 In the **Settings** window for **Combine Solutions**, locate the **Combine Solutions Settings** section.

3 From the **Solution operation** list, choose **Remove solutions**.

4 From the **Exclude or include** list, choose **Include**.

5 From the **Time (s)** list, choose **Manual**.

6 In the **Index** text field, type `401`.

7 Select the **Clear source solution** check box.

8 Right-click **Step 3: Combine Solutions 2** and choose **Reset Solver to Default for Selected Step**.

9 Right-click **Step 3: Combine Solutions 2** and choose **Compute Selected Step**.

RESULTS

Create 1D and 2D datasets from the solution dataset for use in plotting.

Exterior Walls

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

2 Right-click **Results>Datasets** and choose **Surface**.

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

4 Locate the **Data** section. From the **Dataset** list, choose **Study 3: Time-Averaged LES Solution/Solution 4 (sol4)**.

5 Locate the **Selection** section. From the **Selection** list, choose **All boundaries**.

Edge at z=0

1 In the **Results** toolbar, click **More Datasets** and choose **Edge 3D**.

2 In the **Settings** window for **Edge 3D**, type `Edge at z=0` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3: Time-Averaged LES Solution/Solution 4 (sol4)**.

4 Locate the **Selection** section. Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `5 13 18 19` in the **Selection** text field.

6 Click **OK**.

Cut Plane at z=0

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, type `Cut Plane at z=0` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3: Time-Averaged LES Solution/Solution 4 (sol4)**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

*Cut Plane at x=3.69*H*

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, type `Cut Plane at x=3.69*H` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3: Time-Averaged LES Solution/Solution 4 (sol4)**.

4 Locate the **Plane Data** section. In the **x-coordinate** text field, type `3.69*H`.

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

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

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

Cp Contours on the hill

Create a 2D contour plot of the coefficient of pressure on the 3D hill.

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **3D Plot Group 1** toolbar, click **Plot**.

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

4 From the **View** list, choose **New view**.

5 In the **3D Plot Group 1** toolbar, click **Plot**.

6 Click **Go to Source**.

7 In the **Model Builder** window, click **3D Plot Group 1**.

8 In the **Label** text field, type `Cp Contours on the hill`.

9 Locate the **Data** section. From the **Dataset** list, choose **Study 3: Time-Averaged LES Solution/Solution 4 (sol4)**.

10 Click to expand the **Number Format** section. Select the **Manual color legend settings** check box.

11 Select the **Show trailing zeros** check box.

12 In the **Precision** text field, type `5`.

Contour 1

1 Right-click **Cp Contours on the hill** and choose **Contour**.

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

3 In the **Expression** text field, type `p/(0.5*rho*U^2)`.

4 Select the **Description** check box. In the associated text field, type `Cp`.

5 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

6 In the **Levels** text field, type `3.08E-01 2.13E-01 1.19E-01 2.40E-02 -7.05E-02 -1.65E-01 -2.60E-01 -3.54E-01 -4.49E-01 -5.43E-01 -6.38E-01 -7.32E-01 -8.27E-01 -9.21E-01 -1.02E+00`.

7 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

8 From the **Legend type** list, choose **Filled**.

9 Click to expand the **Quality** section.

Contour 2

1 In the **Model Builder** window, right-click **Cp Contours on the hill** and choose **Contour**.

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

3 In the **Expression** text field, type `p/(0.5*rho*U^2)`.

4 Select the **Description** check box. In the associated text field, type `Cp`.

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

6 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

7 In the **Levels** text field, type `3.08E-01 2.13E-01 1.19E-01 2.40E-02 -7.05E-02 -1.65E-01 -2.60E-01 -3.54E-01 -4.49E-01 -5.43E-01 -6.38E-01 -7.32E-01 -8.27E-01 -9.21E-01 -1.02E+00`.

8 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

9 From the **Color** list, choose **Gray**.

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

Cp Contours on the hill

1 In the **Model Builder** window, click **Cp Contours on the hill**.

2 In the **Cp Contours on the hill** toolbar, click **Plot**.

View 3D 5

Modify the view to obtain Figure 4.

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

2 Click the **Orthographic Projection** button in the **Graphics** toolbar.

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

Nondimensional Vorticity Flux

Create a 2D contour plot of the nondimensional vorticity flux.

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **3D Plot Group 2** toolbar, click **Plot**.

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

4 From the **View** list, choose **New view**.

5 In the **3D Plot Group 2** toolbar, click **Plot**.

6 Click **Go to Source**.

7 In the **Model Builder** window, click **3D Plot Group 2**.

8 In the **Label** text field, type `Nondimensional Vorticity Flux`.

9 Locate the **Data** section. From the **Dataset** list, choose **Study 3: Time-Averaged LES Solution/Solution 4 (sol4)**.

10 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

11 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Boundary**.

12 Click **Paste Selection**.

13 In the **Paste Selection** dialog box, type `6 7 8 9` in the **Selection** text field.

14 Click **OK**.

Surface 1

1 Right-click **Nondimensional Vorticity Flux** and choose **Surface**.

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

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

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

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Gray**.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Nondimensional Vorticity Flux** and choose **Arrow Surface**.

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

3 In the **x-component** text field, type `(nz*dtang(p, y)-ny*dtang(p, z))/(0.5*rho*U^2)`.

4 In the **y-component** text field, type `(nx*dtang(p, z)-nz*dtang(p, x))/(0.5*rho*U^2)`.

5 In the **z-component** text field, type `(ny*dtang(p, x)-nx*dtang(p, y))/(0.5*rho*U^2)`.

6 From the **Components to plot** list, choose **Tangential**.

7 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

8 In the **Title** text area, type `Arrow Surface: Nondimensional vorticity flux`.

9 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `400`.

10 Locate the **Coloring and Style** section.

11 Select the **Scale factor** check box. In the associated text field, type `0.25`.

Nondimensional Vorticity Flux

1 In the **Model Builder** window, click **Nondimensional Vorticity Flux**.

2 In the **Nondimensional Vorticity Flux** toolbar, click **Plot**.

View 3D 6

Modify the view to obtain Figure 5.

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

2 Click the **Orthographic Projection** button in the **Graphics** toolbar.

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

Velocity Vectors at z=0

Create a 2D plot showing the velocity vectors on the leeward side of the hill and the region of backward flow on the plane defined by $z = 0$.

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **3D Plot Group 3** toolbar, click **Plot**.

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

4 From the **View** list, choose **New view**.

5 In the **3D Plot Group 3** toolbar, click **Plot**.

6 Click **Go to Source**.

7 In the **Model Builder** window, click **3D Plot Group 3**.

8 In the **Label** text field, type `Velocity Vectors at z=0`.

9 Locate the **Data** section. From the **Dataset** list, choose **None**.

Arrow Surface 1

1 Right-click **Velocity Vectors at z=0** and choose **Arrow Surface**.

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

3 From the **Dataset** list, choose **Cut Plane at z=0**.

4 Locate the **Expression** section. In the **x-component** text field, type `u`.

5 In the **y-component** text field, type `v`.

6 In the **z-component** text field, type `w`.

7 From the **Components to plot** list, choose **Tangential**.

8 Locate the **Title** section. From the **Title type** list, choose **Manual**.

9 In the **Title** text area, type `Arrow Surface: Velocity field at z=0.`.

10 Locate the **Arrow Positioning** section. From the **Placement** list, choose **Mesh nodes**.

11 Locate the **Coloring and Style** section.

12 Select the **Scale factor** check box. In the associated text field, type `0.08`.

Filter 1

1 Right-click **Arrow Surface 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `(x>=0) && (x<=2*H) && (y<= 1.2*H)`.

Surface 1

1 In the **Model Builder** window, right-click **Velocity Vectors at z=0** and choose **Surface**.

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

3 From the **Dataset** list, choose **Cut Plane at z=0**.

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

5 Locate the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Backflow region in magenta.`.

7 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

8 From the **Color** list, choose **Magenta**.

Filter 1

1 Right-click **Surface 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `(u<=0) && (x>=0) && (x<=2*H) && (y<= 1.2*H)`.

Velocity Vectors at z=0

1 In the **Model Builder** window, under **Results** click **Velocity Vectors at z=0**.

2 In the **Velocity Vectors at z=0** toolbar, click **Plot**.

View 3D 7

Modify the view to obtain Figure 6.

1 Click the **Go to XY View** button in the **Graphics** toolbar.

2 Click the **Orthographic Projection** button in the **Graphics** toolbar.

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

Velocity Vectors at x=3.69H

Create a 2D plot showing the velocity vectors in the wake region of the hill on the plane defined by $x = 3.69H$.

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **3D Plot Group 4** toolbar, click **Plot**.
3 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.
4 From the **View** list, choose **New view**.
5 In the **3D Plot Group 4** toolbar, click **Plot**.
6 Click **Go to Source**.
7 In the **Model Builder** window, click **3D Plot Group 4**.
8 In the **Label** text field, type `Velocity Vectors at x=3.69H`.
9 Locate the **Data** section. From the **Dataset** list, choose **None**.

Arrow Surface 1

1 Right-click **Velocity Vectors at x=3.69H** and choose **Arrow Surface**.
2 In the **Settings** window for **Arrow Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Cut Plane at x=3.69*H**.
4 Locate the **Expression** section. In the **x-component** text field, type `u`.
5 In the **y-component** text field, type `v`.
6 In the **z-component** text field, type `w`.
7 From the **Components to plot** list, choose **Tangential**.
8 Locate the **Title** section. From the **Title type** list, choose **Manual**.
9 In the **Title** text area, type `Arrow Surface: Velocity field at x=3.69H.`.
10 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `4e4`.
11 Locate the **Coloring and Style** section.
12 Select the **Scale factor** check box. In the associated text field, type `0.6`.

Filter 1

1 Right-click **Arrow Surface 1** and choose **Filter**.
2 In the **Settings** window for **Filter**, locate the **Element Selection** section.
3 In the **Logical expression for inclusion** text field, type `(y>=0) && (y<=H) && (z>=0) && (z<=2.5*H)`.

Velocity Vectors at x=3.69H

1 In the **Model Builder** window, under **Results** click **Velocity Vectors at x=3.69H**.

2 In the **Velocity Vectors at x=3.69H** toolbar, click **Plot**.

View 3D 8

Modify the view to obtain Figure 7.

1 Click the **Go to YZ View** button in the **Graphics** toolbar.

2 Click the **Rotate Left 90°** button in the **Graphics** toolbar.

3 Click the **Orthographic Projection** button in the **Graphics** toolbar.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.

Instantaneous Streamlines

Create a 3D plot showing instantaneous streamlines.

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **3D Plot Group 5** toolbar, click **Plot**.

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

4 From the **View** list, choose **New view**.

5 In the **3D Plot Group 5** toolbar, click **Plot**.

6 Click **Go to Source**.

7 In the **Model Builder** window, click **3D Plot Group 5**.

8 In the **Label** text field, type `Instantaneous Streamlines`.

9 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Time-Dependent LES Solution/Solution 2 (sol2)**.

10 Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.

Surface 1

1 Right-click **Instantaneous Streamlines** and choose **Surface**.

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

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

4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

Filter 1

1 Right-click **Surface 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `(x>=-4*H) && (x<=6*H) && (z>=-3*H) && (z<=3*H) && (y>=0) && (y<=1.5*H)`.

Instantaneous Streamlines

1 In the **Model Builder** window, under **Results** click **Instantaneous Streamlines**.

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

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

Streamline 1

1 Right-click **Instantaneous 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 `-4*H`.

6 In the **y** text field, type `0.01*H`.

7 In the **z** text field, type `range(-2*H,4*H/10,2*H)`.

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 `0.01*H`.

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

Color Expression 1

Right-click **Streamline 1** and choose **Color Expression**.

Filter 1

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

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

3 In the **Logical expression for inclusion** text field, type `(x>=-4*H) && (x<=6*H) && (z>=-3*H) && (z<=3*H) && (y>=0) && (y<=1.5*H)`.

Streamline 1

Right-click **Streamline 1** and choose **Duplicate**.

Streamline 2

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

2 In the **Settings** window for **Streamline**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Streamline Positioning** section. In the **y** text field, type `range(0,0.2*H,1.6*H)`.

5 In the **z** text field, type `0`.

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Streamline 1**.

7 Right-click **Streamline 2** and choose **Duplicate**.

Streamline 3

1 In the **Model Builder** window, click **Streamline 3**.

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

3 In the **x** text field, type `2*H`.

4 In the **y** text field, type `0.01*H`.

5 In the **z** text field, type `range(-2*H,4*H/10,2*H)`.

6 Right-click **Streamline 3** and choose **Duplicate**.

Streamline 4

1 In the **Model Builder** window, click **Streamline 4**.

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

3 In the **y** text field, type `0.1*H`.

4 Right-click **Streamline 4** and choose **Duplicate**.

Streamline 5

1 In the **Model Builder** window, click **Streamline 5**.

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

3 In the **y** text field, type `0.3*H`.

Slice 1

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

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

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

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

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Streamline 1**.

Translation 1

1 Right-click **Slice 1** and choose **Translation**.

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

3 In the **z** text field, type `-2.9*H`.

Filter 1

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

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

3 In the **Logical expression for inclusion** text field, type `(x>=-4*H) && (x<=6*H) && (z>=-3*H) && (z<=3*H) && (y>=0) && (y<=1.5*H)`.

Instantaneous Streamlines

1 In the **Model Builder** window, under **Results** click **Instantaneous Streamlines**.

2 In the **Instantaneous Streamlines** toolbar, click **Plot**.

View 3D 9

Modify the view to obtain Figure 8.

1 Click the **Orthographic Projection** button in the **Graphics** toolbar.

2 Click the **Zoom Extents** button in the **Graphics** toolbar.

3 In the **Model Builder** window, under **Results>Views** click **View 3D 9**.