Model created in COMSOL Multiphysics 6.4

Unsteady 3D Flow Past a Cylinder

This model is licensed under the COMSOL Software License Agreement 6.4.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Fluid flow past a cylinder is a common test case in computational fluid dynamics. The flow pattern is characterized by the Reynolds number which is defined as

$$\mathrm{Re} = \frac{\rho U_{\mathrm{mean}} D}{\mu}$$

where ρ is the density, U_{mean} is the mean velocity of the free stream, D is the cylinder diameter, and μ is the dynamic viscosity.

The flow patterns around a cylinder in a free stream for different Reynolds numbers are shown in Ref. 1. At Re below 5, the flow remains attached to the cylinder. For Re between 5 and 15, steady wake vortices start forming on the downstream side of the cylinder. The wake becomes unsteady and forms a laminar vortex street for Re between 40 and 150.

Flow around a cylinder in a channel is even more complicated due to the effect of wall boundaries. Computer simulations of this problem at the intermediate Re regime (between 40 and 150) are challenging since they need to be 3D and time-dependent.

In this verification model, a benchmark problem of unsteady, incompressible 3D flow past a cylinder for Re = 100 during a period of 8 seconds is considered. The lift and drag coefficients are computed and are compared with those in Ref. 2.

Model Definition

The geometry is a cylinder of radius R with the axis parallel to the z-axis, and placed at $(x_c, y_c, 0)$, inside the box $[0, L] \times [0, H] \times [0, H]$. Figure 1 shows the geometry corresponding to $R = 0.05$ m , $L = 2.5$ m, $H = 0.5$ m, and $x_c = 0.5$ m, $y_c = 0.2$ m.

The fluid to be considered is incompressible and Newtonian with a kinematic viscosity of 10^{-3} m^2/s. The inflow velocity profile varies in time according to:

$$U(0, y, z, t) = 36\, U_{\mathrm{mean}}\, yz \frac{(H-y)(H-z)}{H^4} \sin\left(\frac{\pi t}{8}\right), \text{ V=W=0.} \tag{1}$$

The lift and drag coefficients C_D and C_L are computed as functions of time,

$$C_D(t) = \frac{2F_D(t)}{\rho U_{\mathrm{mean}}^2 A}, \qquad C_L(t) = \frac{2F_L(t)}{\rho U_{\mathrm{mean}}^2 A} \tag{2}$$

where F_D and F_L are the drag and lift forces, and A is the projected area, $A = 2RH$.

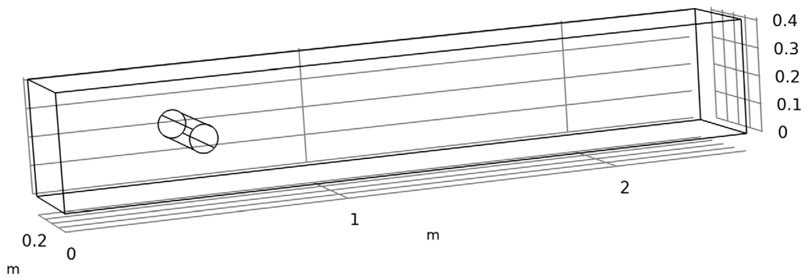


Figure 1: The geometry is a cylinder placed inside a box.

The simulation is performed with a relatively coarse mesh with 7200 hexahedral elements shown in Figure 2. P2+P2 shape functions are chosen for the velocity and pressure to allow for better conservation and higher accuracy compared to P2+P1 and P1+P1. The generalized alpha method with automatic time stepping is chosen since it has less damping than the BDF method.

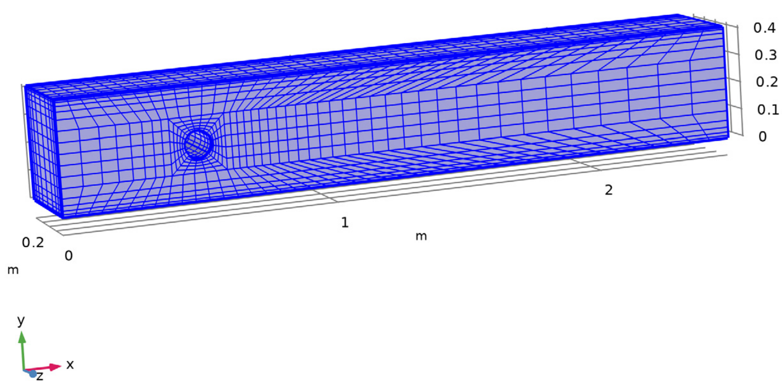


Figure 2: The relatively coarse mesh, with 7200 hexahedral elements, used in the simulation.

Results and Discussion

Figure 3 shows the flow pattern at t = 7.95 s, the last saved time before the inflow velocity returns to zero.

The lift and drag coefficients versus time are shown in Figure 4 and Figure 5 respectively. They capture the general shape of those published Ref. 1 quite well. Note that the nonzero drag at t = 0 s is due to the initial acceleration at the inlet boundary.

In order to verify the results, the L2 norm is computed for the lift and drag coefficients over the entire solved time-span. The results are consistent with those published Ref. 2. The relative error in lift is around 12% and the relative error in drag is approximately 1.2%.

When the mesh size is reduced by a factor of 2, resulting in 57,600 elements, the computational time increases by a factor of 8 but the agreement with the current simulation is still excellent.

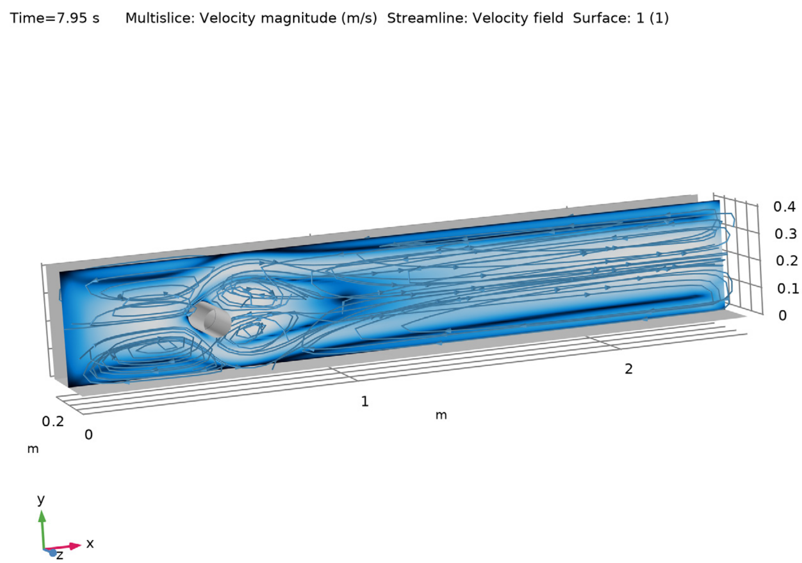


Figure 3: Computed velocity field at t = 7.95 s.

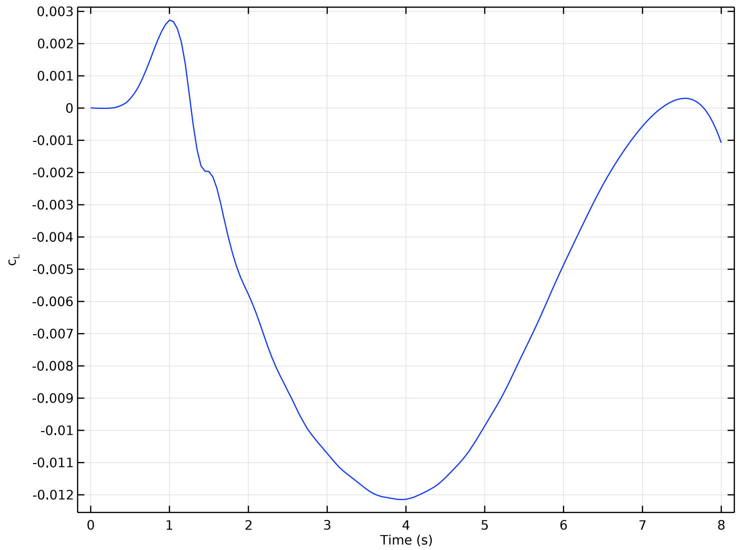


Figure 4: Lift coefficient versus time.

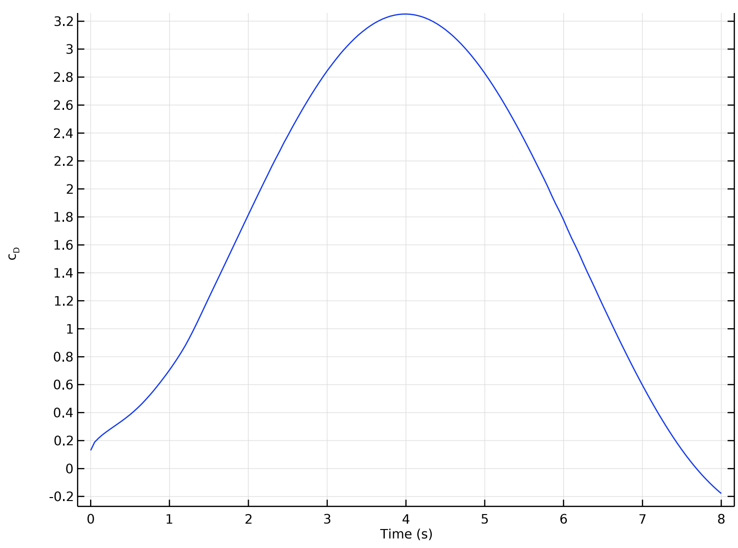


Figure 5: Drag coefficient versus time.

Notes About the COMSOL Implementation

The space discretization P2+P2 coupled with the generalized alpha time discretization works efficiently for this application. P2+P2 elements allow for a coarser mesh, better conservation, and more accuracy compared to P2+P1 and P1+P1 elements. The generalized alpha method has less damping than the BDF method. Automatic time-stepping works well and relatively large time steps can be used, and thus less computational time is needed compared to Ref. 2.

References

1. M. Van Dyke, *An album of fluid motion*, the Parabolic Press, ISBN 0-915760-03-7, 1982.

2. E. Bayraktar, O. Mierka, and S. Turek, “Benchmark Computation of 3D Laminar Flow Around a Cylinder with CFX, OpenFOAM and FeatFlow,” *IJCSE*, vol. 7, no. 3, pp. 253–266, 2012.

Application Library path: `CFD_Module/Verification_Examples/cylinder_flow_3d_periodic`

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** > **Laminar Flow (spf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Time Dependent**.

6 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
U_mean	1[m/s]	1 m/s	Mean inflow velocity
rho	1[kg/m^3]	1 kg/m³	Density
mu	0.001[Pa*s]	0.001 Pa·s	Dynamic viscosity
H	0.41[m]	0.41 m	Height and Width
L	2.5[m]	2.5 m	Length
xc	0.5[m]	0.5 m	Cylinder x-pos
yc	0.2[m]	0.2 m	Cylinder y-pos
R	0.05[m]	0.05 m	Cylinder radius
dt	0.05	0.05	
N	160	160	
final	N*dt	8	

GEOMETRY 1

First, create the box $[0,L]\times[0,H]\times[0,H]$.

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 `L`.

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

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

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	2*R
Layer 2	4*R

7 Find the **Layer position** subsection. Select the **Front** checkbox.

8 Clear the **Bottom** checkbox.

The extra layers are used later to set up the mesh.

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `R`.

4 In the **Height** text field, type `H`.

5 Locate the **Position** section. In the **x** text field, type `xc`.

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

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `45`.

Next, create a smaller box around the cylinder. This box will be used later on in the meshing sequence.

Block 2 (blk2)

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 `4*R`.

4 In the **Depth** text field, type `4*R`.

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

6 Locate the **Position** section. In the **x** text field, type `xc-2*R`.

7 In the **y** text field, type `yc-2*R`.

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

The following operations divide the flow domain into a number of subdomains. This way, a coarser mesh can be used far from the cylinder.

Line Segment 1 (ls1)

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

2 On the object **cyl1**, select Point 2 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **blk2**, select Point 5 only.

Line Segment 2 (ls2)

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

2 On the object **cyl1**, select Point 6 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **blk2**, select Point 7 only.

Line Segment 3 (ls3)

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

2 On the object **cyl1**, select Point 8 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **blk2**, select Point 8 only.

Line Segment 4 (ls4)

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

2 On the object **cyl1**, select Point 4 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **blk2**, select Point 6 only.

6 Click **Build All Objects**.

7 Click the **Wireframe Rendering** button in the **Graphics** toolbar, and rotate the geometry to get a better view.

Now, create the final computational domain with a hollow cylinder.

Difference 1 (dif1)

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

2 Select the objects **blk1** and **blk2** only.

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

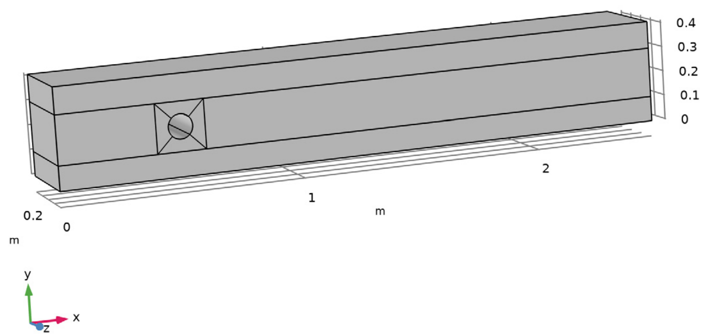
4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **cyl1** only.

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

7 Click the **Wireframe Rendering** button in the **Graphics** toolbar. The geometry looks as follows.

8 In the **Model Builder** window, click **Geometry 1**.



The extra entities are only there to help control the mesh. They are not used in any other part of the simulation. So, assign them as mesh control entities. That way, they will only show up under the **Mesh** node.

Mesh Control Domains 1 (mcd1)

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

2 On the object **fin**, select Domains 2, 4, and 5 only.

Mesh Control Edges 1 (mce1)

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

2 On the object **mcd1**, select Edges 9, 10, 22, and 24 only.

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

Create a selection for the cylinder. This will be useful later for evaluating the drag and lift coefficients.

DEFINITIONS

Cylinder

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

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select the **Group by continuous tangent** checkbox.

5 Select Boundaries 6–9 only.

6 In the **Label** text field, type `Cylinder`.

LAMINAR FLOW (SPF)

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Stabilization**.

3 Click **OK**.

4 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

5 In the **Settings** window for **Laminar Flow**, click to expand the **Consistent Stabilization** section.

6 Find the **Navier–Stokes equations** subsection. Clear the **Crosswind diffusion** checkbox.

7 Click to expand the **Discretization** section. From the **Discretization of fluids** list, choose **P2+P2**. **`P2+P2`** is used because it is more conservative and more accurate than **`P2+P1`** and **`P1+P1`**.

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Laminar Flow (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`.

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 `36*U_mean*z*y*(H-y)*(H-z)/H^4*sin(pi*t/8[s])`, which corresponds to Equation 1.

Outlet 1

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

2 Select Boundary 10 only.

DEFINITIONS

Before continuing with the mesh, add variables to compute the drag and lift coefficient according to Equation 2. Start by defining an integration operator over the cylinder.

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

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

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

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

Now, use this operator for the definition of the drag and lift coefficients.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
c_L	intop1(-spf.T_stressy/(0.5*spf.rho*(U_mean)^2*(2*R*H)))		Lift coefficient
c_D	intop1(-spf.T_stressx/(0.5*spf.rho*(U_mean)^2*(2*R*H)))		Drag coefficient

The reference values are available in a text file.

Interpolation: Reference Lift Coefficient

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

2 In the **Settings** window for **Interpolation**, type `Interpolation: Reference Lift Coefficient` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `c_L_ref`.

4 From the **Data source** list, choose **File**.

5 Click **Browse**.

6 Browse to the model's Application Libraries folder and double-click the file `cylinder_flow_3d_periodic_LiftRef.txt`.

7 Locate the **Data Column Settings** section. In the table, click to select the cell at row number 1 and column number 1.

8 In the **Unit** text field, type `s`.

Interpolation: Reference Drag Coefficient

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

2 In the **Settings** window for **Interpolation**, type `Interpolation: Reference Drag Coefficient` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `c_D_ref`.

4 From the **Data source** list, choose **File**.

5 Click **Browse**.

6 Browse to the model's Application Libraries folder and double-click the file `cylinder_flow_3d_periodic_DragRef.txt`.

7 Locate the **Data Column Settings** section. In the table, click to select the cell at row number 1 and column number 1.

8 In the **Unit** text field, type `s`.

MESH 1

Use advanced operations such as Map and Sweep to create a hexahedral mesh.

Mapped 1

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

Size

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

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

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

4 From the **Predefined** list, choose **Coarse**.

Mapped 1

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

2 Select Boundaries 19–21 and 23 only.

Distribution 1

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

2 Select Edges 12, 13, 16, and 19 only.

Distribution 2

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

2 Select Edges 57 and 58 only.

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

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

5 In the **Element ratio** text field, type 2.

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

Distribution 3

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

2 Select Edges 59 and 60 only.

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

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

5 In the **Element ratio** text field, type 2.

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

7 Click **Build Selected**.

Mapped 2

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

2 Select Boundaries 17 and 22 only.

Distribution 1

1 Right-click **Mapped 2** and choose **Distribution**.

2 Select Edges 27 and 31 only.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 2** and choose **Distribution**.

2 Select Edges 35 and 38 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 8.

6 In the **Element ratio** text field, type 3.

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

8 Select the **Reverse direction** checkbox.

Distribution 3

1 Right-click **Mapped 2** and choose **Distribution**.

2 Select Edges 51 and 54 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 30.

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

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

8 Click **Build Selected**.

Mapped 3

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

2 Select Boundaries 4 and 18 only.

Distribution 1

1 Right-click **Mapped 3** and choose **Distribution**.

2 Select Edges 4 and 32 only.

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

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

5 In the **Element ratio** text field, type 4.

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

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 3** and choose **Distribution**.

2 Select Edges 23 and 28 only.

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

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

5 In the **Element ratio** text field, type 4.

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

Distribution 3

1 Right-click **Mapped 3** and choose **Distribution**.

2 Select Edges 5 and 8 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 43.

6 In the **Element ratio** text field, type 1.6.

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

8 Click **Build Selected**.

Swept 1

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 `10`.

5 In the **Element ratio** text field, type `4`.

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

7 Select the **Symmetric distribution** checkbox.

8 Click **Build All**.

The mesh in Figure 2 has now been generated.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** 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(0,0.05,8)`.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `0.001`.

Solution 1 (sol1)

Choose the generalized alpha method for the time stepping. It has less damping than the BDF.

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, under **Study 1 > Solver Configurations > Solution 1 (sol1)** click **Time-Dependent Solver 1**.

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

5 From the **Method** list, choose **Generalized alpha**.

6 Select the **Initial step** checkbox. In the associated text field, type `0.01`.

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

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

First, evaluate the drag and lift coefficients.

RESULTS

Global Evaluation: Lift and Drag Coefficients

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

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation: Lift and Drag Coefficients` in the **Label** text field.

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

Expression	Unit	Description
c_L	1	Lift coefficient
c_D	1	Drag coefficient

4 Click **Evaluate**.

Table 1: Lift and Drag Coefficients

1 In the **Model Builder** window, expand the **Results** > **Tables** node, then click **Table 1**.

2 In the **Settings** window for **Table**, type `Table 1: Lift and Drag Coefficients` in the **Label** text field.

TABLE 1: LIFT AND DRAG COEFFICIENTS

1 Go to the **Table 1: Lift and Drag Coefficients** window.

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

RESULTS

Table Graph 1

1 In the **Settings** window for **Table Graph**, locate the **Data** section.

2 From the **Plot columns** list, choose **Manual**.

3 In the **Columns** list box, select **Lift coefficient (1)**.

Lift coefficient

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

2 In the **Settings** window for **1D Plot Group**, type `Lift coefficient` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** checkbox. In the associated text field, type c_L.

5 In the **Lift coefficient** toolbar, click **Plot**. Compare with Figure 4.

Drag coefficient

1 Right-click **Lift coefficient** and choose **Duplicate**.

2 In the **Model Builder** window, click **Lift coefficient 1**.

3 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

4 In the **y-axis label** text field, type c_D.

5 In the **Label** text field, type Drag coefficient.

Table Graph 1

1 In the **Model Builder** window, click **Table Graph 1**.

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

3 In the **Columns** list box, select **Drag coefficient (1)**.

4 In the **Drag coefficient** toolbar, click **Plot**. Compare with Figure 5.

Continue with computing the maximum and minimum values.

Global Evaluation: Maximum Lift and Drag Coefficients

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

2 In the **Settings** window for **Global Evaluation**, type Global Evaluation: Maximum Lift and Drag Coefficients in the **Label** text field.

3 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Maximum**.

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

The maximum lift and drag coefficients are about $c_{L,max}$= 0.00273 and $c_{D,max}$= 3.2511.

Global Evaluation: Minimum Lift and Drag Coefficients

1 Right-click **Global Evaluation: Maximum Lift and Drag Coefficients** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, type Global Evaluation: Minimum Lift and Drag Coefficients in the **Label** text field.

3 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Minimum**.

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

The minimum lift and drag coefficients are about $c_{L,min}$= -0.0121 and $c_{D,min}$= -0.1784.

Table 2: Maximum Lift and Drag Coefficients

1 In the **Model Builder** window, under **Results** > **Tables** click **Table 2**.

2 In the **Settings** window for **Table**, type `Table 2: Maximum Lift and Drag Coefficients` in the **Label** text field.

Table 3: Minimum Lift and Drag Coefficients

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

2 In the **Settings** window for **Table**, type `Table 3: Minimum Lift and Drag Coefficients` in the **Label** text field.

Calculate the relative L2 error of the Lift and Drag Coefficients over time using the ratio of the L2 norms of the error and reference values.

Global Evaluation: Relative error (L2)

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
`sqrt(sum((at(k*dt,c_L)-int1(k*dt))^2,k,0,160))/sqrt(sum((int1(k*dt))^2,k,0,160))`	`1`	`Relative eror (L2) of lift coefficient`
`sqrt(sum((at(k*dt,c_D)-int2(k*dt))^2,k,0,160))/sqrt(sum((int2(k*dt))^2,k,0,160))`	`1`	`Relative eror (L2) of drag coefficient`

4 Locate the **Data** section. From the **Time selection** list, choose **Last**.

5 Click ▾ next to **Evaluate**, then choose **New Table**.

6 In the **Label** text field, type `Global Evaluation: Relative error (L2)`.

Table 4: Relative Error (L2)

1 In the **Model Builder** window, under **Results** > **Tables** click **Table 4**.

2 In the **Settings** window for **Table**, type `Table 4: Relative Error (L2)` in the **Label** text field.

The values are consistent with the values in the paper.

Now, visualize the velocity field as shown in Figure 3.

Velocity (spf)

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

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

3 Clear the **Show legends** checkbox.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

5 Locate the **Data** section. From the **Time (s)** list, choose **7.95**. Since the inlet velocity vanishes at the final time step, the solution at t=7.95s is chosen for a better visualization of the streamlines.

Multislice 1

To see the flow pattern clearly, choose to plot the velocity parallel to the xy – plane.

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

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

3 Find the **x-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **y-planes** subsection. In the **Planes** text field, type `0`.

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

Streamline 1

Add streamlines with arrows.

1 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Streamline**.

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

3 From the **Positioning** list, choose **Uniform density**.

4 In the **Density level** text field, type `8.7`.

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

6 Select the **Radius scale factor** checkbox. In the associated text field, type `0.0025`.

7 Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

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

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

10 Click **Define custom colors**.

11 Set the RGB values to 71, 145, and 199, respectively.

12 Click **Add to custom colors**.

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

Surface 1

1 Right-click **Velocity (spf)** 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 **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

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 **Cylinder**, and add the bottom and back boundaries. This corresponds to:

4 Select Boundaries 2, 3, and 6–9 only.

5 In the **Velocity (spf)** toolbar, click **Plot**.