

# Flow Past a Cylinder

This model is licensed under the COMSOL Software License Agreement 5.2a. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

# Introduction

The flow of fluid behind a blunt body such as an automobile is difficult to compute due to the unsteady flows. The wake behind such a body consists of unordered eddies of all sizes that create large drag on the body. In contrast, the turbulence in the thin boundary layers next to the streamlined bodies of aircraft and fish create only weak disturbances of flow.

An exception to this occurs when you place a slender body at right angles to a slow flow because the eddies organize. A von Kármán vortex street appears with a predictable frequency and involves the shedding of eddies from alternating sides. Everyday examples of this phenomenon include singing telephone wires and an automobile radio antenna vibrating in an air stream.

From an engineering standpoint, it is important to predict the frequency of vibrations at various fluid speeds and thereby avoid undesirable resonances between the vibrations of the solid structures and the vortex shedding. To help reduce such effects, plant engineers put a spiral on the upper part of high smokestacks; the resulting variation in shape prohibits the constructive interference of the vortex elements that the structure sheds from different positions.

## Model Definition

To illustrate how you can study such effects, the following model examines unsteady, incompressible flow past a long cylinder placed in a channel at right angle to the oncoming fluid. With a symmetric inlet velocity profile, the flow needs some kind of asymmetry to trigger the vortex production. This can be achieved by placing the cylinder with a small offset from the center of the flow. In this case, an unstructured mesh is used, and the small asymmetry in the mesh proves to be enough to trigger the vortex production.

The simulation time necessary for a periodic flow pattern to appear is difficult to predict. A key predictor is the Reynolds number, which is based on cylinder diameter. For low values (below 100) the flow is steady. In this simulation, the Reynolds number equals 100, which gives a developed von Kármán vortex street, but the flow still is not fully turbulent.

The frequency and amplitude of oscillations are stable features, but flow details are extremely sensitive to perturbations. To gain an appreciation for this sensitivity, you can compare flow images taken at the same time but with such minor differences as are created by different tolerances for the time stepping. It is important to note that this sensitivity is a physical reality and not simply a numerical artifact.

Before calculating the time-varying forces on the cylinder, you can validate the method of computation at a lower Reynolds number using the direct nonlinear solver. This saves time because you can find and correct simple errors and mistakes before the final time-dependent simulation, which requires considerable time.

The viscous forces on the cylinder are proportional to the gradient of the velocity field at the cylinder surface. Evaluating the velocity gradient on the boundary by directly differentiating the FEM solution is possible but not very accurate. The differentiation produces 1st-order polynomials when second-order elements are used for the velocity field. A far better approach is to use a pair of reaction force operators to compute the integrals of the viscous forces, comparable to second-order accurate integrals of the viscous forces. An alternative approach would be to use a pair of weak-constraint variables to enforce the no slip condition. Preferably use the reaction force operator instead of weak constraints when computing integrals of reaction forces or fluxes in postprocessing.

The drag and lift forces themselves are not as interesting as the dimensionless drag and lift coefficients. These depend only on the Reynolds number and an object's shape, not its size. The coefficients are defined as

$$C_{\rm D} = \frac{2F_{\rm D}}{\rho U_{\rm mean}^2 L}$$
$$C_{\rm L} = \frac{2F_{\rm L}}{\rho U_{\rm mean}^2 L}$$

using the following parameters:

- $F_{\rm D}$  and  $F_{\rm L}$  are the drag and lift forces
- $\rho$  is the fluid's density
- $U_{\text{mean}}$  is the mean velocity
- L is the characteristic length, in this case the cylinder's radius

# Results and Discussion



Figure 1 shows the flow pattern resulting from the geometry.

Figure 1: A plot of the last time step clearly shows the von Kármán path.

The flow around a cylinder is a common benchmark test for CFD algorithms. Various research teams have tried their strengths on this problem using different techniques. Results from some of these experiments have been collected by Schäfer and Turek (Ref. 1), who also used them to compute a probable value for the "real" answer.

Figure 2 shows how the lift coefficient develops a periodic variation as the von Kármán vortex structure is formed.



Figure 2: Lift coefficient,  $C_L$ , as a function of time.



Figure 3: Drag coefficient, C<sub>D</sub>, as a function of time.

# Reference

1. M. Schäfer and S. Turek, "Benchmark Computations of Laminar Flow Around Cylinder," E.H. Hirschel ed., *Flow Simulation with High-Performance Computers II*, *Volume 52 of Notes on Numerical Fluid Mechanics*, Vieweg, pp. 547–566, 1996.

#### Application Library path: COMSOL\_Multiphysics/Fluid\_Dynamics/cylinder\_flow

### Modeling Instructions

From the File menu, choose New.

#### NEW

In the New window, click Model Wizard.

#### MODEL WIZARD

- I In the Model Wizard window, click 2D.
- 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 Preset Studies>Time Dependent.
- 6 Click Done.

#### GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 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]	l m/s	Mean inflow velocity

Next, create a smoothed step function feature that you will use for ramping up the inflow velocity.

Step I (step I)

- I On the Home toolbar, click Functions and choose Global>Step.
- 2 In the Settings window for Step, locate the Parameters section.
- **3** In the **Location** text field, type **0.1**.

#### GEOMETRY I

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type **2.2**.
- 4 In the **Height** text field, type 0.4.

#### Circle I (c1)

- I Right-click Rectangle I (rI) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Circle.
- 3 In the Settings window for Circle, locate the Position section.
- 4 In the **x** text field, type 0.2.
- **5** In the **y** text field, type **0.2**.
- 6 Locate the Size and Shape section. In the Radius text field, type 0.05.

#### Difference I (dif1)

- I Right-click Circle I (cl) and choose Build Selected.
- 2 On the Geometry toolbar, click Booleans and Partitions and choose Difference.
- **3** Select the object **rI** only.
- 4 In the Settings window for Difference, locate the Difference section.
- 5 Find the Objects to subtract subsection. Select the Active toggle button.
- 6 Select the object **cl** only.
- 7 On the Geometry toolbar, click Build All.



#### 8 Click the Zoom Extents button on the Graphics toolbar.

#### MATERIALS

In the Model Builder window, under Component I (compl) right-click Materials and choose Blank Material.

Material I (mat1)

I In the Settings window for Material, locate the Material Contents section.

**2** In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Density	rho	1	kg/m³	Basic
Dynamic viscosity	mu	1e-3	Pa∙s	Basic

#### LAMINAR FLOW (SPF)

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- 2 Select Boundary 1 only.

To define a parabolic velocity profile use the predefined local curve parameter s. Ramp up the velocity using the previously defined step function. Append the inverse unit bracket [1/s] to the time variable t because the step function expects a dimensionless argument.

- 3 In the Settings window for Inlet, locate the Velocity section.
- **4** In the  $U_0$  text field, type U\_mean\*6\*s\*(1-s)\*step1(t[1/s]).

#### Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 4 only.

#### MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Mesh Settings section.
- 3 From the Element size list, choose Finer.
- 4 Click Build All.

If you zoom in on the inlet and the cylinder you can see the boundary layers that the physics-controlled mesh gives.



#### STUDY I

Step 1: Time Dependent

- I In the Model Builder window, expand the Study I node, then click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- **3** In the **Times** text field, type range(0,0.2,3.4) range(3.5,0.02,7).

#### Solution 1 (soll)

- I On the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Study I>Solver Configurations node.
- 3 In the Model Builder window, expand the Solution I (soll) node, then click Time-Dependent Solver I.
- **4** In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.
- **5** Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.
- 6 On the Study toolbar, click Compute.

#### RESULTS

#### Velocity (spf)

Add a **Particle Tracing with Mass** node to the first default plot group to reproduce the plot in Figure 1.

I In the Model Builder window, click Velocity (spf).

#### Particle Tracing with Mass I

On the Velocity (spf) toolbar, click More Plots and choose Particle Tracing with Mass.

#### Velocity (spf)

- I In the Settings window for Particle Tracing with Mass, click to expand the Mass and velocity section.
- 2 Locate the Mass and Velocity section. In the Mass text field, type 4\*pi/3\*1e-9.
- **3** Find the **Initial velocity** subsection. In the **x component** text field, type **u**.
- 4 In the y component text field, type v.
- 5 Locate the Particle Positioning section. In the y text field, type range(0.1,0.05,0.3).
- **6** In the **x** text field, type 0.
- 7 Click to expand the Release section. From the Release particles list, choose At intervals.
- 8 Select the Start time check box.
- 9 In the associated text field, type 3.6.
- **IO** In the **Time between releases** text field, type **0.4**.
- II Click to expand the Coloring and style section. Locate the Coloring and Style section. Find the Line style subsection. From the Type list, choose None.
- 12 Find the Point style subsection. From the Type list, choose Point.

# **13** Find the **Point motion** subsection. From the **When particle leaves domain** list, choose **Disappear**.

I4 On the Velocity (spf) toolbar, click Plot.

To reproduce Figure 2 and Figure 3 of the lift and drag coefficients, first add an Integral data set for computing the total reaction force on the cylinder.

#### Integral I

On the Results toolbar, click More Data Sets and choose Evaluation>Integral.

#### Selection

- I On the **Results** toolbar, click **Selection**.
- 2 In the Settings window for Selection, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Boundary.
- **4** Select Boundaries 5–8 only.

#### I D Plot Group 3

- I On the **Results** toolbar, click **ID Plot Group**.
- 2 In the Settings window for 1D Plot Group, locate the Data section.
- 3 From the Data set list, choose Integral I.

#### Point Graph I

On the ID Plot Group 3 toolbar, click Point Graph.

#### I D Plot Group 3

- I In the Settings window for Point Graph, locate the y-Axis Data section.
- 2 In the Expression text field, type -reacf(v)\*2/(spf.rho\*U\_mean^2\*0.1).
- **3** Select the **Description** check box.
- **4** In the associated text field, type Lift coefficient.
- 5 On the ID Plot Group 3 toolbar, click Plot.

Compare the graph with that shown in Figure 2.

Finally, visualize the drag coefficient using the following steps:

#### I D Plot Group 4

- I On the Home toolbar, click Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for 1D Plot Group, locate the Data section.
- 3 From the Data set list, choose Integral I.

# Point Graph 1

On the ID Plot Group 4 toolbar, click Point Graph.

I D Plot Group 4

- I In the Settings window for Point Graph, locate the y-Axis Data section.
- 2 In the Expression text field, type -reacf(u)\*2/(spf.rho\*U\_mean^2\*0.1).
- **3** Select the **Description** check box.
- 4 In the associated text field, type Drag coefficient.
- 5 On the ID Plot Group 4 toolbar, click Plot.

Compare with Figure 3.