Flow Past a Cylinder

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 Karman vortex street appears with a predictable frequency, and it 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. The cylinder is offset somewhat from the center of the flow to destabilize what otherwise would be steady-state symmetrical flow. 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 Karman 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 2nd-order elements are used for the velocity field. A far better approach is to use a pair of weak-constraint variables to enforce the no-slip condition; at the same time they collect 2nd-order accurate information on the viscous forces.

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

$$C_D = \frac{2F_D}{\rho U_{\mathrm{mean}}^2 D}$$

$$C_L = \frac{2F_L}{\rho U_{\mathrm{mean}}^2 D}$$

using the following parameters:

- F_D and F_L are the drag and lift forces
- ρ is the fluid’s density
- U_{mean} is the mean velocity
- D is the characteristic length, in this case the cylinder’s diameter

Results

Figure 6-1 shows the flow pattern resulting from the geometry.

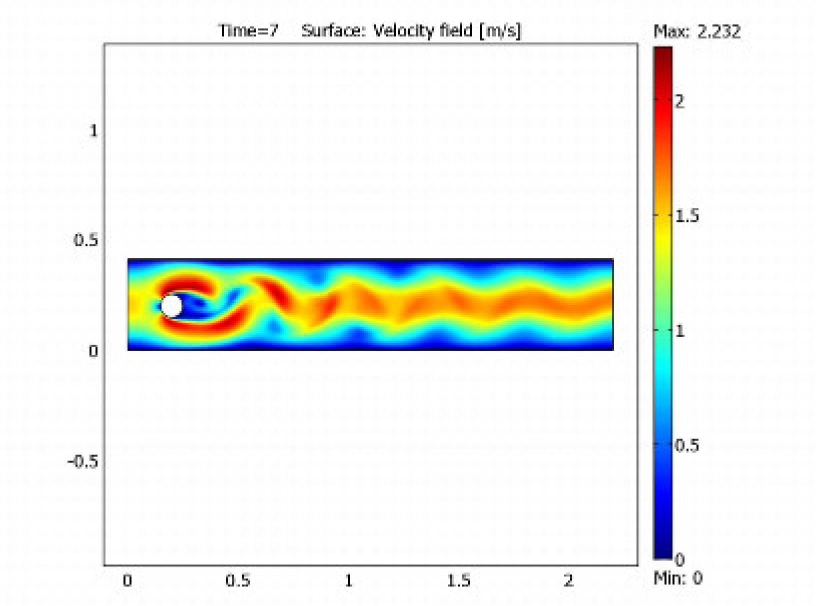


Figure 6-1: A plot of the last time step clearly shows the von Karman path.

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 use them to compute a probable value for the "real" answer.

Reference

1. M. Schäfer and S. Turek, "Benchmark computations of laminar flow around cylinder", E.H. Hirschel (editor), *Flow Simulation with High-Performance Computers II, Volume 52 of Notes on Numerical Fluid Mechanics*, Vieweg, 1996, pp. 547-566.

Model Library Path: `COMSOL_Multiphysics/Fluid_Dynamics/cylinder_flow`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator**, select **2D** from the **Space dimension** list.

2 In the list of application modes, select **COMSOL Multiphysics>Fluid Dynamics>Incompressible Navier-Stokes**, **Steady-state analysis**. Accept the default entry in the **Element** list, **Lagrange - P_2 P_1** elements.

3 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, select **Axes/Grid Settings**.

2 In the **Axes/Grid Settings** dialog box, specify the following settings. To enter the grid spacing, first click the **Grid** tab and clear the **Auto** check box.

AXIS		**GRID**	
x min	`-0.3`	x spacing	`0.2`
x max	`2.5`	Extra x	
y min	`-0.3`	y spacing	`0.05`

y max	0.7	Extra y	0.41

3 Click **OK**.

4 From the **Options** menu, select **Constants**.

5 Enter the following names and expressions:

NAME	EXPRESSION
rho0	1
eta0	1e-3
Umax	0.3

6 Click **OK**.

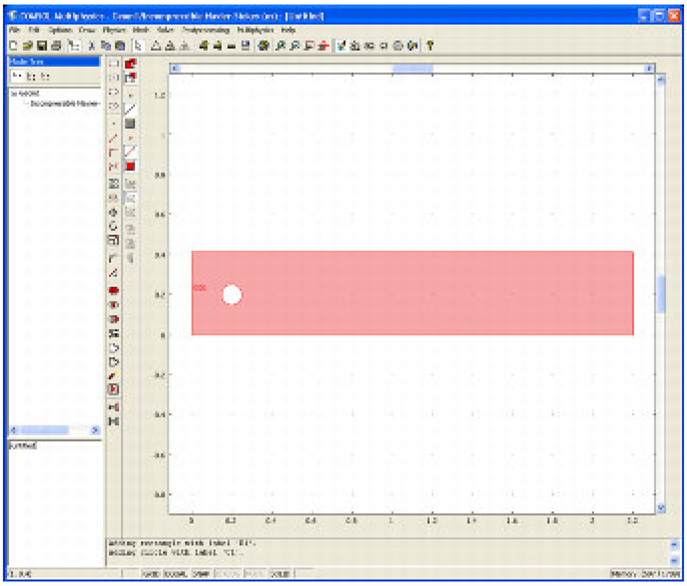
These values together with the cylinder having a diameter of 0.1 m lead to a Reynolds number of 20.

GEOMETRY MODELING

1 From the Draw menu, select the **Rectangle/Square** button. With the mouse, create a rectangle `R1` from (0, 0) to (2.2, 0.41).

2 From the Draw menu, select the **Ellipse/Circle (Centered)** button. Using the mouse, create a circle `C1` with center at (0.2, 0.2) and a radius of 0.05. Use the right mouse button to ensure that you are creating a circle, and use the tick mark at y = 0.15 to obtain the correct radius.

3 Create the composite object. From the Draw toolbar, click the **Create Composite Object** button. In the **Set formula** edit field enter `R1-C1`.



Click **OK**.

PHYSICS SETTINGS

Application Mode Properties

You must first enable the weak constraints in the **Application Mode Properties** dialog box.

1 From the **Physics** menu, choose **Properties**.

2 In the **Application Mode Properties** dialog box, go to the **Weak constraints** list and select **Non-ideal**.

3 Click **OK**.

Boundary Conditions

First set the boundary conditions for the Incompressible Navier-Stokes equations as if the weak constraints did not exist. When you couple a constraint variable to a dependent variable inside the domain, the weak constraint mode analyzes the constraints specified for the domain variable and replaces them with a weak term involving the constraint variable.

1 From the **Physics** menu, choose **Boundary Settings**.

2 In the **Boundary Settings** dialog box, enter the following:

BOUNDARY	**1**	**2, 3, 5–8**	**4**
Type	Inflow	No-slip	Outflow
u	`4*Umax*s*(1-s)`		
v	`0`		
p			`0`

3 Click the **Weak Constr.** tab.

4 Select boundaries 1–4 (the channel boundaries) and clear the **Use weak constraints** check box.

5 Click **OK**.

Subdomain Settings

1 From the **Physics** menu, select **Subdomain Settings**.

2 In the **Subdomain Settings** dialog box, enter the following material properties:

SUBDOMAIN	**1**
ρ	`rho0`
η	`eta0`

3 Click **OK**.

MESH GENERATION

1 From the **Mesh** menu, choose **Free Mesh Parameters**.

2 Click the **Custom mesh size** button, enter `0.03` in the **Maximum element size** edit field, enter `1.2` in the **Element growth rate** edit field, and enter `0.1` in the **Mesh curvature factor** edit field.

3 Click **OK**.

4 Click the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar to solve the model.

POSTPROCESSING AND VISUALIZATION

The quantities in the benchmark study are the drag and lift coefficients. These you can calculate easily by integrating the weak constraint variables (which are really Lagrange multipliers corresponding to the viscous forces) and the pressure over the surface of the cylinder. First calculate the drag coefficient:

1 From the **Postprocessing** menu, choose **Boundary Integration**.

2 Select boundaries 5 to 8, corresponding to the cylinder surface. Enter the drag force expression `-lm1*2/(rho0*(2*Umax/3)^2*0.1)` in the **Expression** edit field.

3 Click **OK**.

The result, 5.579384, appears in the message log and agrees well with the interval [5.57 5.59] given in Ref. 1. Now calculate the lift coefficient.

4 Change the integrand in the **Expression** edit field to `-lm2*2/(rho0*(2*Umax/3)^2*0.1)`. Also this value, 0.010568, is in perfect agreement with Schäfer and Turek (Ref. 1), who give the

interval [0.0104 0.0110] for acceptable solutions.

Time-Dependent Simulation

Now that you know that the machinery works for low Reynolds numbers and steady flow, you can do a time-dependent simulation at Re = 100.

OPTIONS AND SETTINGS

In the **Constants** dialog box change the maximum inflow velocity U_{max} to 1.5, corresponding to a Reynolds number of 100.

COMPUTING THE SOLUTION

During the first seconds of the simulation, before the system reaches a state of steady periodic motion, the output is not really interesting. Therefore you can save memory by saving only the value five times per second up to 3.5 seconds, and then fifty times per second for another second and a half. Perform the following steps for the time-dependent simulation of drag and lift:

1. From the **Solve** menu, choose **Solver Parameters**.
2. In the **Solver** list, select **Time dependent**.
3. Enter `[0:0.2:3.5 3.52:0.02:7]` in the **Times** edit field.
4. Enter `1e-4` in the **Absolute tolerance** edit field.
5. Click **OK**.
6. Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The difference from the former case can be seen immediately. Downstream of the cylinder, the Karman path is clearly visible (see Figure 6-1 earlier in this discussion).

To see the evolution of the vortex trail from zero velocity until the flow is fully developed, click the **Animation** button on the Plot toolbar on the left-hand side of the user interface. Notice that the time scale of the movie changes after 3.5 seconds.

You can also investigate the forces on the tube as a function of time by assigning the surface integral to a so-called integration-coupling variable:

1. Select the menu item **Options>Integration Coupling Variables>Boundary Variables**.
2. Ctrl-click boundaries 5 to 8 to select them as a group. Type `Lift` as the first **Name** item and `-lm2*2/(rho0*(2*Umax/3)^2*0.1)` in the **Expression** edit field.
3. Clear the **Global destination** check box. This saves memory because you define the integral value to be stored only in one node instead of all the nodes in the model.
4. Click the **Destination** tab and select point 1. Click **OK.**
5. Select the menu item **Solve>Update Model**. This calculates the integral without re-solving the problem.
6. From the **Postprocessing** menu, select **Domain Plot Parameters**. Select all the time steps, then click **Plot type: Point plot**. Go to the **Point** tab, select point **1**, and type `Lift` in the **Expression** edit field. The plot clearly shows the oscillations in the lift force.

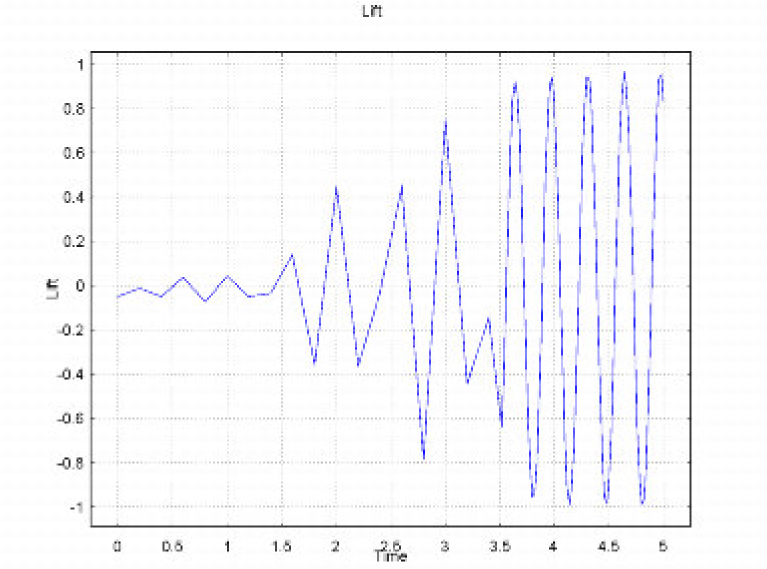


Figure 6-2: Time evolution of the total lift force on the cylinder.

Finally, you can investigate how suspended particles behave in the flow stream. Consider, for example, small water droplets entering with the air flow.

1 In the **Plot Parameters** dialog box, go to the **Particle Tracing** page.

2 Select the **Particle tracing plot** check box.

3 In the **Plot type** list, select **Points**.

4 In the **Predefined forces** list, select the **Khan and Richardson force (ns)**.

5 On the **Start Points** tab, enter `0` in the **x** edit field and then enter `0.1:0.05:0.3` in the **y** edit field.

6 Go to the **Initial Values** page. Clear the **Auto** check box and enter `3.6` in the **Start time** edit field to study the particles only for fully developed flow.

7 In the **Initial velocity** edit fields, enter `u` and `v` to give the particles the same initial velocity as the inflow.

8 Go to the **Point Settings** page, find the **When particles leaves the domain** list, and select **Disappear**.

9 On the **Release** page, select **Time between releases** and specify the time as `0.4`.

10 Click the **Advanced** button.

11 Select the **Manual tuning of step size** check box and enter `0.02` in the **Initial time step** edit field and `0.01` for **Maximum time step**.

12 Click **OK** to close the **Advanced Parameters** window.

13 Go to the **Animate** tab and click **Start Animation**. Remember that the time scale of the movie changes after 3.5 seconds.