Cylinder in Cross Flow—Comparing CFD Simulations w/ Experiments

Theoretical Drag Coefficients

Examples of cylindrical objects in cross flow (i.e. with the freestream flow direction normal to the cylinder axis) include wind and water flow over offshore platform supports, flow across pipes or heat exchanger tubes, and wind flow over power and phone lines. The drag coefficient for such an object depends strongly on the behavior of the fluid around the cylinder (see Figure 1). For example, depending on the Reynolds number, \( Re = \frac{\rho U D}{\mu} \), the flow pattern near the cylinder can vary significantly, where \( \rho \) and \( \mu \) are the fluid density and viscosity, \( U \) is the upstream velocity, and \( D \) is the cylinder diameter. For low velocities (i.e. \( Re_D < 5 \)), the flow around the cylinder is unseparated; whereas for \( 5 < Re_D < 40 \), two stationary eddies form immediately downstream of the cylinder. For higher velocities (i.e. \( Re_D > 40 \)), an unsteady wake flow occurs, the width and nature of which depends on the Reynolds number.

Fig. 1 Different characteristic flow regimes manifested by a cylinder in cross flow [1]
In Figure 2, the theoretical drag coefficient associated with these different flow regimes is plotted for a smooth circular cylinder as a function of Reynolds number. For example, for the case of no separation (i.e. Case A), the drag coefficient is a little less than 50. For the case of a laminar boundary layer with a wide turbulent wake (i.e. Case D), the drag coefficient is approximately 1.6 whereas for the case of a turbulent boundary layer with a narrower turbulent wake, the drag coefficient is reduced. In this case, the drag coefficient is approximately 0.30. So why is this?

To answer this question, let’s consider how the flow behaves around the cylinder. Over the forward portion of the cylinder, the surface pressure decreases from the stagnation point toward the shoulder. In this region, the boundary layer (i.e. the thin region adjacent to the surface where viscous shear effects are important) develops under a favorable pressure gradient (i.e. $\frac{\partial P}{\partial \theta} < 0$). In this region the net pressure force on a fluid element in the direction of the flow is sufficient to overcome the resisting shear force. Thus, the motion of the element in the flow direction is maintained. However, the surface pressure eventually reaches a minimum and then begins increasing toward the rear of the cylinder. Thus, the boundary layer in this downstream region experiences an adverse pressure gradient (i.e. $\frac{\partial P}{\partial \theta} > 0$). Since the pressure increases in the flow direction, a fluid element in the boundary layer experiences a net pressure force opposite to its direction of motion. At some point, the momentum of the fluid element will be insufficient to carry it into regions of increasing pressure. Here, the fluid adjacent to the solid surface is brought to rest, and flow separation from the surface occurs. In the case of a turbulent boundary layer,
there is more momentum associated with the fluid. Thus, *separation* occurs farther back on the cylinder.

As a result, the wake region behind the cylinder is considerably narrower and there is a considerable drop in *pressure drag* (with only a slight increase in the *friction drag*).

This is actually the reason why a golf ball is dimpled. The surface roughness associated with the dimples facilitates an earlier transition from a laminar boundary layer to a turbulent boundary layer (i.e. at smaller Reynolds numbers). The resulting reduction in drag permits the golf ball to be hit greater distances.

**Comparing ANSYS Fluent Simulations to Experiments**

Now, let’s consider two of these cases—namely, Case A and Case D. In both cases, we will compare the theoretical value (from Fig. 1) with the experimental value and the computationally determined value using ANSYS Fluent.

**Case A—Creeping Flow \((Re_D \approx 0.15)\)**

As we indicated earlier, the theoretical \(C_D\) value for Case A is approximately 45 - 50. This compares quite favorably with the value attained using ANSYS Fluent which was 43.7 (shown in Fig. 4). A plot of the velocity magnitude is shown in Figure 3. Due to the low velocities, there is no flow separation, and as might be expected, the regions of highest velocity occur along the sides of the cylinder where the fluid accelerates to move around the object. Unfortunately, the speeds involved here are too low to be tested in the MME wind tunnel. Thus, no experimental \(C_D\) value is available for comparison for this case.

Fig. 3 Velocity Magnitude Contours for Case A
Case D— Wide Turbulent Wake ($Re_D \approx 20,000 – 40,000$)

As we indicated earlier, the theoretical $C_D$ value for Case D is approximately 1.6. This compares well with the value attained using ANSYS Fluent which was 1.524 (shown in Fig. 5). This value is simply the result of adding the pressure drag (i.e. 1.498) to the viscous (or, friction) drag component (i.e. 0.026). Clearly in this case, the pressure drag is more dominant. The experimentally determined drag coefficient ($C_D = 1.70–1.72$) is shown in Fig. 6 for a slightly higher Reynolds number (i.e. $Re_D = 48,200$). All three values compare quite favorably.
Fig. 5 Computationally Determined $C_d$ for Case D

<table>
<thead>
<tr>
<th>Object</th>
<th>$D$ (mm)</th>
<th>$T_1$ (m/s)</th>
<th>$P_{atm}$ (mmHg)</th>
<th>$T_1$ (Celsius)</th>
<th>$T_1$ (K)</th>
<th>$\mu_{air}$ (Pa-s)</th>
<th>$\rho_{air}$ (kg/m$^3$)</th>
<th>$Re$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylinder</td>
<td>88.9</td>
<td>23.33</td>
<td>8.55</td>
<td>746</td>
<td>23.33</td>
<td>296.48</td>
<td>1.84E-05</td>
<td>1.16791</td>
</tr>
</tbody>
</table>

Fig. 6 Experimentally Determined $C_d$ for Case D (slightly higher $Re_d$)
A plot of the velocity magnitude for Case D (as predicted from the CFD simulation) is shown in Figure 7. In this image, the stagnation region at the front of the cylinder, and the wide wake region behind the cylinder can be clearly observed with defined regions of recirculation. It should also be noted in Fig. 8 that (for a similar Reynolds number) the static pressure begins showing negative values at angles of approx. 30-40° (as measured from the front of the cylinder) which agrees well with the negative surface pressure difference measurements shown in Fig. 6. Here, negative surface pressure measurements were recorded in the wind tunnel at approx. 20-30°.

Fig. 7 Contours of Velocity Magnitude and Velocity Vectors for Case D
Fig. 8 Contours of Static Pressure at a Similar $Re_D$

References: