ME 608 Fluid Dynamics — Fall 2018
Prof. M. Wosnik, TAs J. Cuevas Bautista, C. Boahmah-Mensah

Laboratory Experiment: **Drag forces on bluff and streamlined bodies**

Experiment description

The drag force experienced by various shapes in air or water flow is an important design consideration for both stationary structures (buildings, bridges, lattice towers, etc) and moving objects (automobiles, planes, human-powered vehicles).

In this laboratory experiment we will determine the drag coefficients of various shapes in a wind tunnel (cf. Figure 1). The following shapes (cf. Figure 2) can be mounted on a force balance:

1. Circular cylinder(s)
2. Square cylinder
3. Airfoil shape, NACA 0020 ("00"=no camber, symmetrical, "20"=maximum thickness is 20% of chord length c, c.f., https://en.wikipedia.org/wiki/NACA_airfoil)

Experiment Goal

Measure wind speed and drag forces for the various shapes, and calculate and compare drag coefficients $c_{D,i}$ as a function of Reynolds number.

Theory

A fluid flow will exert a resultant force, $\vec{F}_{res}$, on an immersed body, which can be calculated as the integral of normal and tangential stresses over the body's surface:

$$\vec{F}_{res} = \iint\limits_{CS} \vec{t}\, dS = \iint\limits_{CS} \hat{n} \cdot \overline{\overline{T}}\, dS \tag{1}$$

Here $\vec{t}$ is the stress vector, which is related to the stress tensor $\overline{\overline{T}}$ via $\vec{t} = \hat{n} \cdot \overline{\overline{T}}$, and consists of normal and tangential stresses (see lecture notes). The drag force can be calculated as the streamwise (x) component of this resultant force, which takes the following form:

$$F_D = F_{res,\,x} = \iint\limits_{CS} t_x dS = \iint\limits_{CS} \left(\hat{n} \cdot \overline{\overline{T}}\right)_x dS = \iint\limits_{CS} -p\hat{n}_x dS + \iint\limits_{CS} \left(\tau_{xx} + \tau_{yx} + \tau_{zx}\right) dS \tag{2}$$

The first integral is the *pressure drag* (also called: *form drag*) and the second integral is the *viscous drag* (also called: *friction drag*) due to the viscous stresses, τ. Evaluating pressure and velocity gradients (to calculate viscous stresses) over the surface of an entire body can be quite involved and difficult, and is often not practical.

To simplify quantifying this drag force, a *drag coefficient* can be introduced, defined as:

$$F_D = c_D * A * KE \; = c_D * A * \frac{1}{2}\rho U^2 \qquad \text{or :} \qquad c_D \equiv \frac{F_D}{A\frac{1}{2}\rho U^2} \tag{3}$$

(Note: Remembering this definition is a "piece of $c_D * A * KE$")

It can be shown from dimensional analysis (Buckingham Pi theorem, c.f. Ch.7/HW 12) that two *nondimensional groups* can be formed from the quantities measured in this experiment, which results in

$$\frac{F_D/L}{D\rho U_c^2} = \phi\left(\frac{\rho U_c L_c}{\mu}\right) \tag{4}$$

where F_D/L is the drag force per unit span, L $[N/m]$. Using previously defined nondimensional parameters, we can write

$$\boldsymbol{c_D = f\,(Re)}$$

In general, similar shapes will have the same drag coefficient, c_D, provided dynamic similarity is achieved by operating at the same Reynolds number, $Re_L = \rho U_c L_c/\mu$. In practice, the drag coefficient for many shapes can become independent of Reynolds number above a certain Reynolds number, for a certain range of Reynolds number[1]. This sometimes allows us to conduct model scale experiments at lower, but sufficiently high Reynolds numbers (="incomplete similarity", c.f. example problem in text book Chapter 7).

In the wind tunnel experiments we can measure:

- Drag force F_D using the one-component force balance mounted in the wind tunnel test section.
- Reference area A. For bluff bodies, this is the flow facing area, calculated as maximum height x transverse length, $A = D * L$. Note that for airfoils, the reference area for drag coefficients is typically the planform area, $A_{planforn} = c * L$, where c is the chord length – however, to facilitate comparison among the three shapes, we will be using the flow-facing area as well.
- Fluid properties: density, $\rho = \rho(T)$; viscosity, $\mu = \mu\,(T)$
- Fluid velocity: the air velocity can be measured with a Pitot-static tube and a manometer.

Experiments

1. Measure the relevant dimensions of (each) shape. How variable are diameter/height over the span of the shape?
2. Measure or evaluate fluid properties: $\rho = \rho(T)$; $\mu = \mu\,(T)$. Repeat at end of lab.
3. Mount a shape on the force balance. Make sure the angle is set to the desired value for the square shape and the airfoil. Make sure the shape does not touch the wind tunnel side walls.
4. Zero the force balance, zero the well manometer.
5. Turn on the wind tunnel, select a wind tunnel speed.

[1] The drag coefficient can become independent of Reynolds number above a certain Reynolds number as long as the fundamental character of the flow over the shape does not change, e.g., the boundary layer on the smooth circular cylinder changing from a laminar to a turbulent boundary layer at what is known as the "drag crisis" at $Re_D \approx 2 \times 10^5$. See plot and discussion in your fluids book

6. The flow is measured with a pitot-static tube. When flow is steady, read and record the water column heights in the two legs (on left) of the manometer: $h_1 = (p_{atm} - p_{static})/\rho_{man} g$ and $h_2 = (p_{atm} - p_{stagnation})/\rho_{man} g$. Make sure you read at the meniscus of the water surface. The manometer is divided into 0.1 inch lines. However, to get an accurate velocity measurement, you need to (visually) further subdivide, e.g. to within 0.025 inch, or 0.020 inch. The manometer height difference Δh is the difference between these two readings. Use Bernoullis equation to calculate free stream air velocity in the wind tunnel.
7. Read and record the drag force from the drag balance monitor. (If it fluctuates a lot, do not try to "average", but rather take several random readings, including outliers, and average those random readings. This is proper "human sampling" of a random variable.)
8. Change the wind tunnel speed and repeat steps 5 through 7 four times, for a total of five different Reynolds numbers per shape. (caution: larger shapes, or the airfoil at higher angles of attack at high wind speeds can exceed the force balance capacity!)
9. Change to another shape, and repeat step 1 through 8. In addition to zero angle of attack, also test the airfoil at two non-zero angles of attack.

Calculate results "on the fly" right away to see whether you are getting reasonable results!!!

Experiment Discussion

- How repeatable are the measurements?
- What are sources of uncertainty?
  - Have different students read the instruments.
  - To what accuracy can you read an instruments, e.g., the manometer height? "Propagate" this uncertainty into the velocity measurement calculation.
- How does flow blockage in the test section affect the results?
- How dependent on Reynolds number are your results (over the range covered)?
- Were there any surprising results?

Deliverables

- Write a Laboratory Report (can be co-authored by up to 4 students), following the report template that was posted. (Follow the template, it will guide you and make the report writing easier!)
- In the report
  - Plot your results for the various shapes
    - drag force vs. Reynolds number
    - drag coefficient vs. Reynolds number.
  - Compare your results to values found in literature (include this in plots and discussion!)
- Discussion of the experiment and your findings, including answering the questions under "Experiment Discussion" above.

Grading

The Laboratory Experiment will count for 9 percentage points of your final grade in this course.

- Attendance/participation grade: 4 points
  given for both the lecture where lab will be discussed (12/3) and laboratory experiment.

- Laboratory Report: 5 points
  graded based on the Deliverables discussed above.

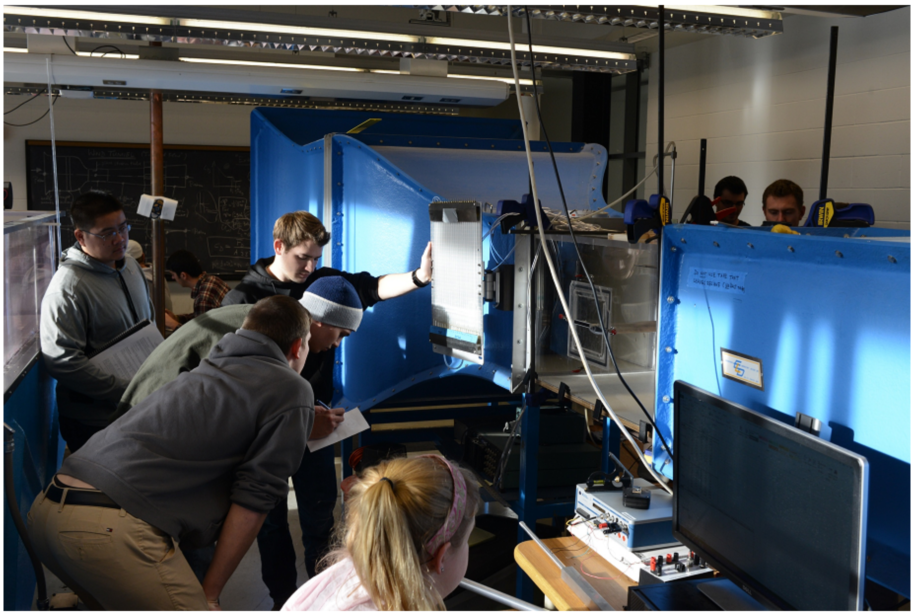

Figure 1: Open-return (Eiffel-type) wind tunnel in S125 Kingsbury, model ELD-404b, `http://www.eldinc.com/wind-tunnels/`

Figure 2: Examples of shapes to be tested in wind tunnel.

Wind tunnel: components and pressure profile

The open return wind tunnel consists of, in flow direction:

- a bell mouth inlet, followed by a short straight section (do NOT sit in it!)
- turbulence management section with honeycomb and screens
- contraction (flow area decrease), which accelerates the flow into the
- test section (18" x 18" x 36")
- diffuser (flow area increase), which decelerates the flow and increases pressure. The diffuser also serves as a square-to-round transition, leading into the
- fan – provides pressure increase to compensate for losses in wind tunnel
- another short diffuser, with acoustic treatment

The wind tunnel components are sketched in Figure 3. The pressure profile throughout the wind tunnel is sketched in the same figure, aligned with the corresponding wind tunnel components. Since inlet and outlet are at atmospheric (room) pressure, the entire wind tunnel operates below atmospheric pressure. You can think of the fan as simply providing a pressure increase to compensate for viscous losses ("head loss") along the tunnel. [2]

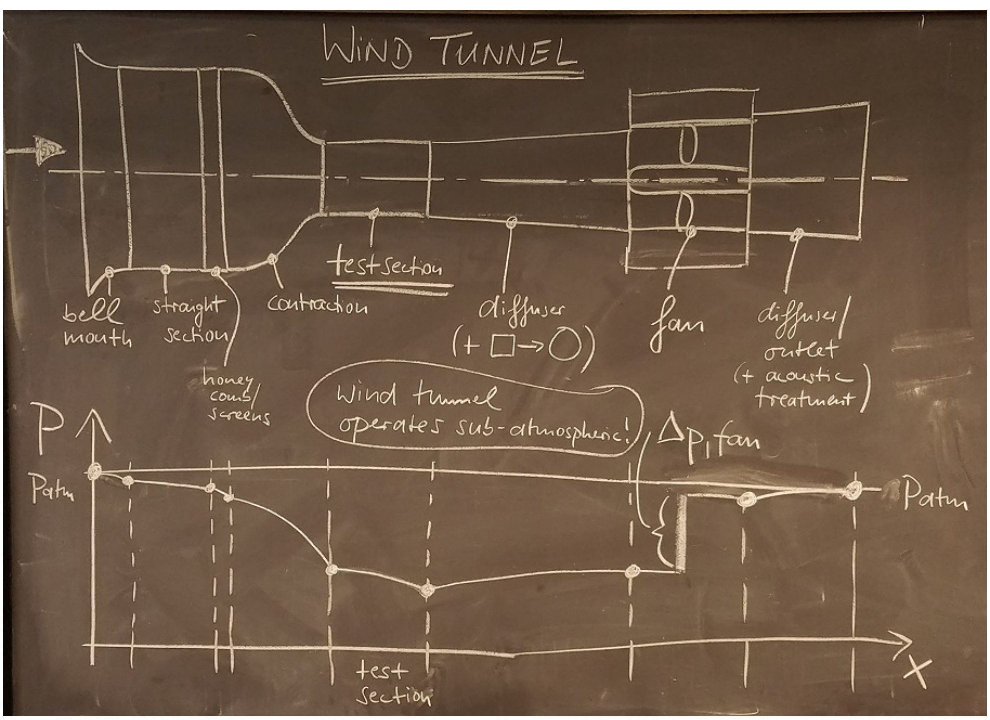

Figure 3: Wind tunnel: sketch of components and pressure profile. Tunnel operates sub-atmospheric, with lowest pressure in test section.

[2] If there were no viscous losses in the wind tunnel, i.e. inviscid (ideal) flow, we wouldn't need a fan at all. This is another example of a paradox often encountered when using inviscid (ideal) flow analysis.

Last, but not least, we hope that you will have fun while gaining some experience with fluid mechanics experiments…

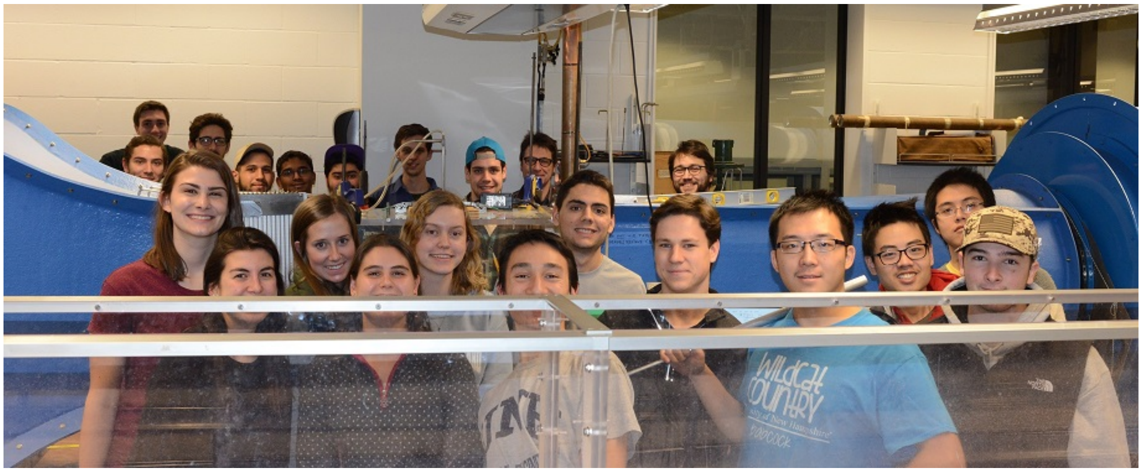

Figure 4: Wind tunnel experiments, ME 608 in Fall 2016.