Bernoulli, vorticity, and boundary layers

I. BASICS OF BERNOULLI

See the numbered schematics in Figure 1 which are associated with the following problems. In all problems here assume that the Reynolds number is high and the effect of viscosity is negligible. Apply Bernoulli's equation along a streamline to answer these questions;

$$P + \frac{1}{2}\rho \mathbf{v}^2 + \rho g z = \text{constant along a streamline.}$$

Recall that Bernoulli's equation assumes incompressible, steady, and inviscid flow. In some of the problems you will also need to apply conservation of mass; e.g. that the flow rate through the system is constant.

1. Consider air flow through a venturi meter at an unknown volumetric flow rate Q. The cross sectional area of the inlet and contraction are known. The height of water in the manometer is measured. Derive a relationship for the flow rate as a function of the measured height.

2. Air flows through a duct as shown. Determine the velocity of the air as a determined by the measured height of water in the manometer.

3. From the measurement of the height of water before and after a sluice gate, estimate the total flow rate through the open channel.

4. The inlet to a venturi is at atmospheric pressure. Air flows through the venturi at flow rate Q_a. A small tube is connected to a tank of liquid through a thin pipe. The liquid is drawn into the venturi by the low pressure at the throat. The liquid flow rate, Q_l, can be computed through Poiseueille's law ($\Delta P = Q128\mu L/(\pi D^4)$). Derive a formula for the flow rate of liquid in terms of the flow rate of air, the fluid properties of air and the liquid, the area contraction ratio of the venturi, and the length/diameter of the liquid pipe.

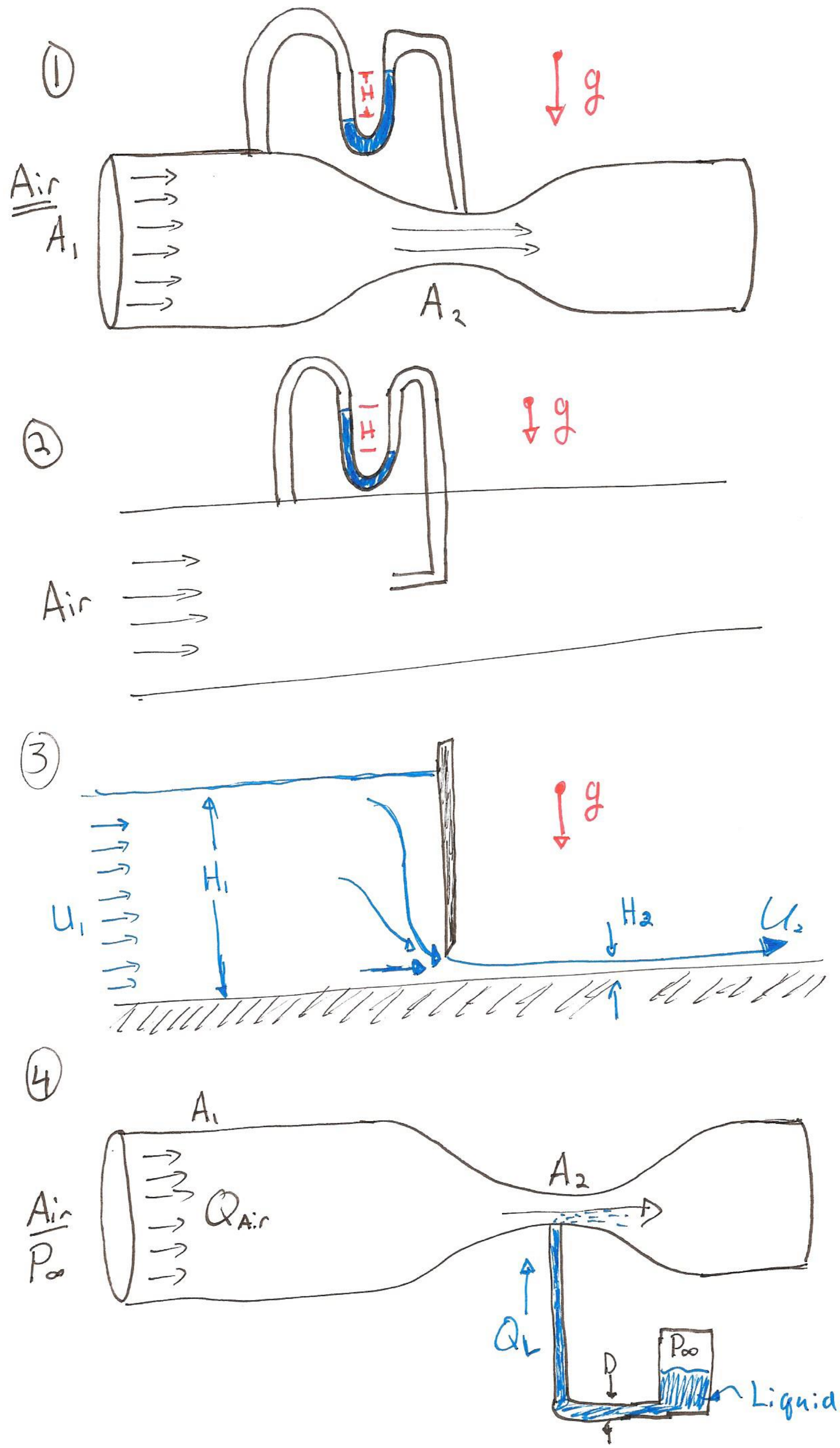

FIG. 1 Schematics for basic Bernoulli calculations.

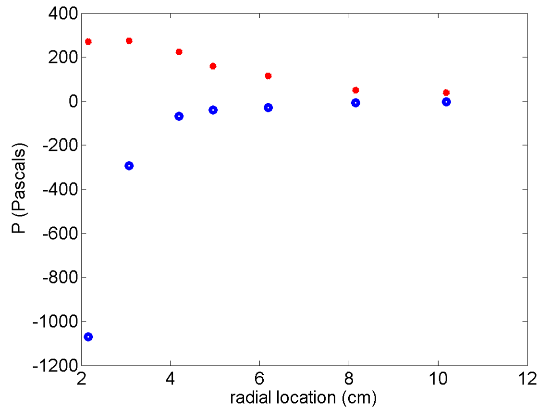

FIG. 2 Pressure data for the levitating plate. The blue data with open circles the device is oriented downward such that the plate is held up with respect to gravity. If the air flow is turned off the plate falls to the ground. For this case the air flow was measured to be 0.0138 m^3/s and the gap height was measured to be 2 mm. The red data with closed dots the device is inverted and the plate is held up with respect to gravity. If the air flow is turned off the plate collapses to the blower. In this case the flow rate was measured to be 0.0023 m^3/s and the gap height was determined to be 0.7 mm.

II. LEVITATING PLATE

In class I will demonstrate a simple experimental setup. Essentially air blows out of an opening in a flat plate. Another plate is brought close to the opening and this plate is pulled upward and levitates like an upside down air hockey puck, even though the air is blowing outward. The behavior seems counterintuitive, but is easily explained with basic fluid dynamics. We can explain the effect qualitatively quite easily and quantitatively with a little more work.

- To make progress, we will simplify the Navier Stokes equations significantly. Start with looking up the full Navier-Stokes equations for a Newtonian incompressible flow in cylindrical coordinates. Assume flow is only radial, v_r, such that the theta and z components of the velocity are zero. Also assume the problem is symmetric and all theta derivatives drop out. Leave viscosity in the formulation. Demonstrate that the radial momentum equation and conservation of mass with these simplifications are,

  $$\frac{\partial (r v_r)}{\partial r} = 0,$$

  $$\rho v_r \frac{\partial v_r}{\partial r} = -\frac{\partial P}{\partial r} + \mu \frac{\partial^2 v_r}{\partial z^2}.$$

  For each term in the original Navier Stokes, comment why that term is zero and what approximation you are making.

- For the case where a constant volumetric flow rate, Q, is forced into the inner hole, solve for the velocity as a function of radial coordinate. Here, you are assuming that v_r is only a function of r or is interpreted as the average velocity across the gap.

- Assume viscosity is zero and remove those terms from your equation. Convert your equation to a form that looks like Bernoulli's equation along a streamline from the center to the outer edge.

- Using the momentum equation without viscosity (Bernoulli's equation) to solve for pressure as a function of radial coordinate. This analysis will combine Bernoulli's equation with conservation of mass. When you integrate for the pressure, you will take the pressure at the outer radius to be atmospheric. Compare your result to the experimental measurements seen in Figure 2.

- From the pressure with no viscosity in the formulation, compute the total lift force for the plate.

- The results with no viscosity can explain the basic levitation phenomena, but they are inconsistent with the data where the disk is floating like an air hockey puck. Lets leave in viscosity and integrate the equation over the gap height (and assuming there is symmetry between $z = 0$ and $z = H$),

$$\int_0^H \left(\frac{\partial}{\partial r} \left(\frac{1}{2} \rho v_r^2 + P \right) \right) dz = -2\mu \left. \frac{\partial v_r}{\partial z} \right|_{z=0}$$

If we define $\bar{v}_r$ as the appropriately averaged velocity then we have

$$\frac{\partial}{\partial r} \left(\frac{1}{2} \rho \bar{v}_r^2 + P \right) = -\frac{2\mu}{H} \frac{\partial v_r}{\partial z}$$

Note that $\tau = \mu \frac{\partial v_r}{\partial z}$ is the shear stress at the solid surface,

$$\frac{\partial}{\partial r} \left(\frac{1}{2} \rho \bar{v}_r^2 + P \right) = -\frac{2\tau}{H}.$$

If the flow were laminar we could have some hope of solving for the shear stress exactly. However, the flow is certainly turbulent and the shear stress τ is given by the empirical parameter friction factor, f. Empirically, for flow in a pipe we describe the shear stress as $4\tau = \frac{1}{2} \rho v^2 f$. This form goes back to when we discussed the Moody diagram for turbulent pipe flow. The factor of 4 is just by convention. To simplify things we will assume the friction factor is a constant and will not bother to account for its dependence on Reynolds number. The resulting equations for conservation of momentum

$$\frac{\partial}{\partial r} \left(\frac{1}{2} \rho \bar{v}_r^2 + P \right) = -\frac{\rho \bar{v}_r^2 f}{4H}$$

- Using the above equation and your prior expression for v_r from conservation of mass, you can actaully integrate the equations and solve for the pressure as a function of radius. The boundary conditions is that the pressure at the outer radius matches that of the atmosphere. You can use Wolfram alpha or some other symbolic manipulator if that helps, but it is easy enough to do by hand.

- You should now be able to compare what this relatively simple model predicts versus what the experiment says. You should be able to select a single friction factor empirically that models the data reasonably well..

III. COMSOL

1. Create a 2D channel of height 2. Make the domain go from $-1 < y < 1$. Make the inlet channel 5 units long. Create a second channel of height 1 from $-0.5 < y < 0.5$ that is fed by the larger one, see Figure 3. Make the inlet velocity field $u = (1 - y^2)$. Set the outlet pressure as zero. Start with Reynolds number of 10. Compute and plot the pressure as a function of distance down the channel. Increase the Reynolds number by a factor of 10 and recompute and re-plot. Keep increasing by factors of 10 until the simulation breaks down and high Reynolds number. Compare the last good solution to that you would get from Bernoulli's equation. Think about the behavior at low Reynolds number and whether you could have a simple model for the pressure in that case too.

2. Now we will use comsol to compute lift on a flat plate airfoil. Create a large domain, about 20 units long by 10 high seems to work fine. Cut out a thin flat plate airfoil with a small inclination angle (start with about 4 degrees). Make the airfoil $\ell = 2$ units long and 0.1 units thick. Give the leading and trailing edges a little fillet to make the flow smoother around the entrance/exit of the airfoil, see Figure 4.

    - Start by using the laminar flow, stationary solver, with a Reynolds number of 1000. Compute the total lift and drag force (integrate the total stress in the x and y directions around the airfoil boundary). Convert these forces per unit width to drag and lift coefficients using,

    $$C_L = \frac{\text{Force/width}}{\frac{1}{2} \rho U^2 \ell}.$$

    Note that if you set the density to 1 and the length to 2, the Comsol computed force is just the lift coefficient. Change the angle of attack, α, and get C_L as a function of alpha (you can do this by adding a

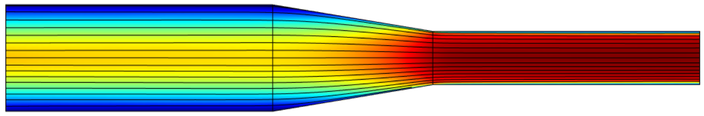

FIG. 3 Schematics for comsol problem 1 with the converging nozzle.

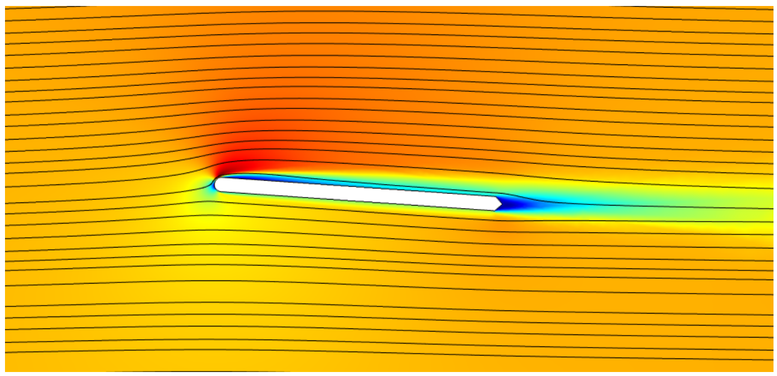

FIG. 4 Schematics for comsol problem 2 with the airfoil.

parameter by right clicking on global parameters, adding angle as a parameter in the geometry, and then adding a parametric sweep under the solver). You should be able to get a solution up to about 10 degrees, beyond that you may have trouble getting converged solution due to the inherent unsteady motion that would set in above the stall condition. Compare your computed results for lift to the theoretical one for a flow with zero viscosity,

$$C_L = 2\pi \sin(\alpha).$$

Here you should have a single plot which is computed C_L and C_D as a function of angle, while also comparing the C_L to the theoretical.

- Now change the problem to turbulent flow (use the k-e option), stationary solver, with a Reynolds number of 10,000. It may be easiest just to start over with a new problem selecting the turbulent flow as the physics at the beginning. Recompute everything as before and create the same graph as in the previous problem. Note that this will take longer to compute so don't compute more than every 1 degree. Also, as you start to change the angle of attack, you might find that the solution stops converging around 4 degree. If you switch to the transient flow solver, it will work OK beyond that, but then the solution time increases further. If you are using the transient flow solver, just compute every 2 degrees.
- There is some data for a flat plate on the website. I took this from a paper at Reynolds number of 40,000. Data was taken from two sources. Compare your simulation to this data. Also, check out this report from 80 years ago for measurements of lift and drag on different airfoil shapes. https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/19930091108.pdf Look at the data for the first shape which is the thinnest symmetric shape and eyeball how good your computation is in relation to data an an airfoil shape.

IV. VORTEX DYNAMICS

The vorticity, ω, is related to the solid body rotation of infinitesimal fluid particles. The vorticity is given as the curl of the velocity field, $\nabla \times \mathbf{v}$. The 2D ideal vortex has a velocity field in radial coordinates given as

$$v_\theta = \frac{\Gamma}{2\pi r}$$

1. Show that the circulation around this vortex is given by Γ.
2. Show that, except at the origin of the vortex, the vorticity everywhere else is zero.
3. Since the vorticity is zero everywhere except at the origin of the vortex, the velocity field can be shown to follow a potential. When $\nabla \times \mathbf{v} = 0$ we can define the velocity potential, ϕ. The potential is related to the velocity through $\nabla\phi = \mathbf{v}$. Prove that we can define a velocity potential by showing that $\nabla \times \nabla\phi = 0$. You can do this simple vector calculus proof by carrying out the operation component by component.
4. Find the velocity potential which gives the ideal vortex velocity field.
5. Since the velocity is defined by $\nabla\phi = \mathbf{v}$, show that the equation for fluid motion is $\nabla^2\phi = 0$. What's nice about this result is that the equation is linear, thus when flow has no vorticity we can add up the velocity potentials for different simple flows to get a more complex one. We are allowed to use superposition.

In 2D the vorticity equation is

$$\frac{D\omega}{Dt} = 0$$

The interesting thing about this equation is that it says that vorticity is locked in the fluid and just goes with the flow. Vorticity of individual fluids particles does not change. This means that if we have a single ideal point vortex, it sits there spinning and the vortex cannot move itself. If we have two vortices, since the velocity field away from the origin is irrotational, the velocity field of each vortex adds up everywhere in the fluid. The velocity field of one vortex moves the other at its center and vice-versa. To simulate point vortices, we can write a very simple program using these facts.

We go to each point vortex in the simulation. We compute the velocity of that vortex's center by adding up the velocity field of all other vortices. We can then move the vortex in simulation a small distance using Euler integration, i.e. new position is current position plus velocity times time. We then update all vortex positions and move on. We will review this method in class.

Write a simple program to simulation the motion of vortices. Confirm your simulation with two vortices. The two vortices should move each other in a straight line when given the same strength, but opposite spin directions. The two vortices should orbit each other in a circle if given the same strength and the same direction. If you are feeling good about this, try to extend to four vortices and see if you can simulate the approach of two vortices with a wall as discussed in the book.

V. BOUNDARY LAYERS

1. A passenger plane is flying at cruise altitude of 30,000 ft. How far from the leading edge of the wing does the boundary layer become turbulent, if at all? Considering the flow along the fuselage, what is the boundary layer thickness on the fuselage near the back of the plane? These are estimates so be clear in what you are assuming for size, speed, and fluid properties.
2. A car is traveling at 35mph (15 m/s). Consider the flow on the hood of the car. How far from front does the boundary layer become turbulent, if at all ? Estimate the boundary layer thickness over the length of the hood. These are estimates so be clear in what you are assuming for size, speed, and fluid properties.
3. For water flowing at a meter per second over a long flat plate, how far down does the flow transition to turbulence. Make a dimensional plot of boundary layer thickness versus distance to get a feel for the order of magnitude of numbers involved. Your plot should include the overlap of the laminar to turbulent transition.
4. A one square meter plate is held horizontal to the water flow at 1 m/s. Compute the total drag force acting on the flat plate. Consider whether the flow is laminar, turbulent, or mixed when computing your answer.