COE-C2003 Basic Course on Fluid Mechanics, S2020

Exam Fri 11.12.2020, 13:00 – 17:00 o'clock.

Explain the various stages in the questions. Only equations and final solutions are not enough for full points.

<u>RETURN EACH QUESTION (1 – 5) INTO ITS OWN RETURN BOX IN MyCourses BY 17:00 O'CLOCK !</u>

1. Consider a cylinder or sphere in a flow field.
   A) What happens to the drag coefficient when the Reynolds number of the flow is increased, and why is this ? (3p)
   B) Are there sudden changes in the drag coefficient ? If yes, why are those taking place ? (1p)
   C) For streamlined bodies, drag increases when Re –number is increased. Is this different from the above situation and why ? (2p)
   In your answer, you may also draw a picture, if it helps the understanding (do not attached a photograph). Maximum answer length is 2 pages including possible figures.

2. A converging elbow turns water through an angle of 135° in a vertical plane (see Fig. 1). The flow cross section diameter is 400 mm at the elbow inlet, section (1), and 200 mm at the elbow outlet, section (2). The elbow flow passage volume is 0.2 m^3 between sections (1) and (2). The water volume flowrate is 0.40 m^3/s and the elbow inlet and outlet pressures are 150 kPa and 90 kPa. The elbow mass is 12 kg. Calculate the horizontal (x-direction) and vertical (z-direction) anchoring forces required to hold the elbow in place.

A Draw the control volume that you can use to define the forces. Define the control volume accurately; how is it oriented compared to the elbow. (1p)

B Mark and name all the forces to the control volume you have drawn. (1p)

C Define the anchoring force components (x- and z – components) (4p)

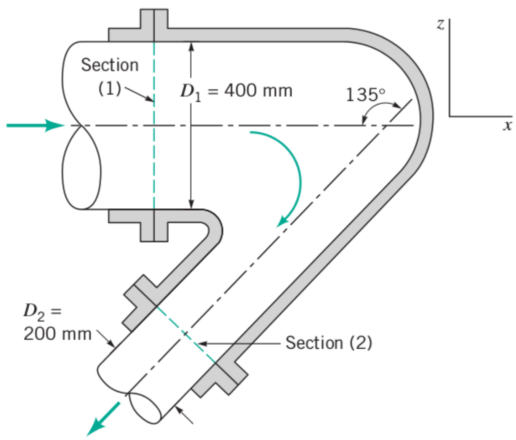

Figure 1. Question 2.

3. Water flows in a pipe according to the Figure 2. The pipe diameter is 2 cm (ε=1,5x10^{-6} m) and the flow rate is $(1.0y \times 0.35\ l/min)$. '$y$' is the last digit of your student number (If it is e.g. 5, then y =5, and the multiplier is 1.05. If it is e.g. zero, then y =0, the multiplier is 1.00). The flow exits the pipe as a free jet. The bends (elbows) in the pipe are threaded.

   a) How are losses in pipes typically classified ? Divide the losses in the present setup to these classes. (1p)

   b) Explain section-by-section, how the pressure is changing between the points (1) and (2). (2p)

   c) Define the pressure at the point (1) (3p)

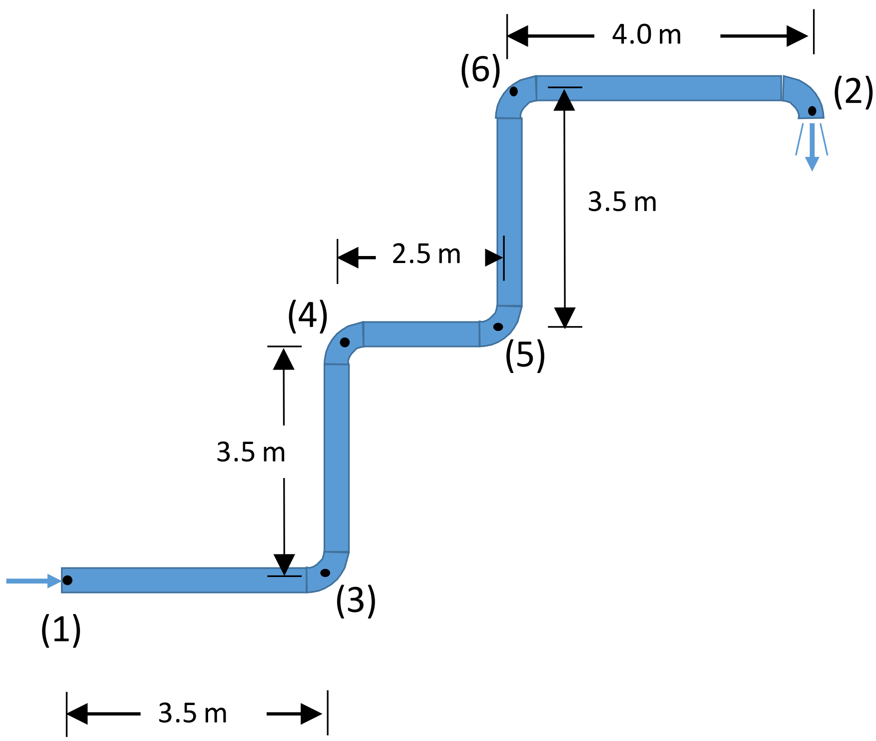

Figure 2. Question 3.

4. An inward flow radial turbine (see Figure 3) involves a nozzle angle, α_1, of 60° and an inlet rotor tip speed, U_1, of 6 m/s. The ratio of the rotor inlet to outlet diameters is 2.0. The absolute velocity leaving the rotor at section (2) is radial with a magnitude of 12 m/s. Determine the energy transfer per unit of mass of fluid flowing through this turbine if the fluid is (a) air or (b) water. (6p)

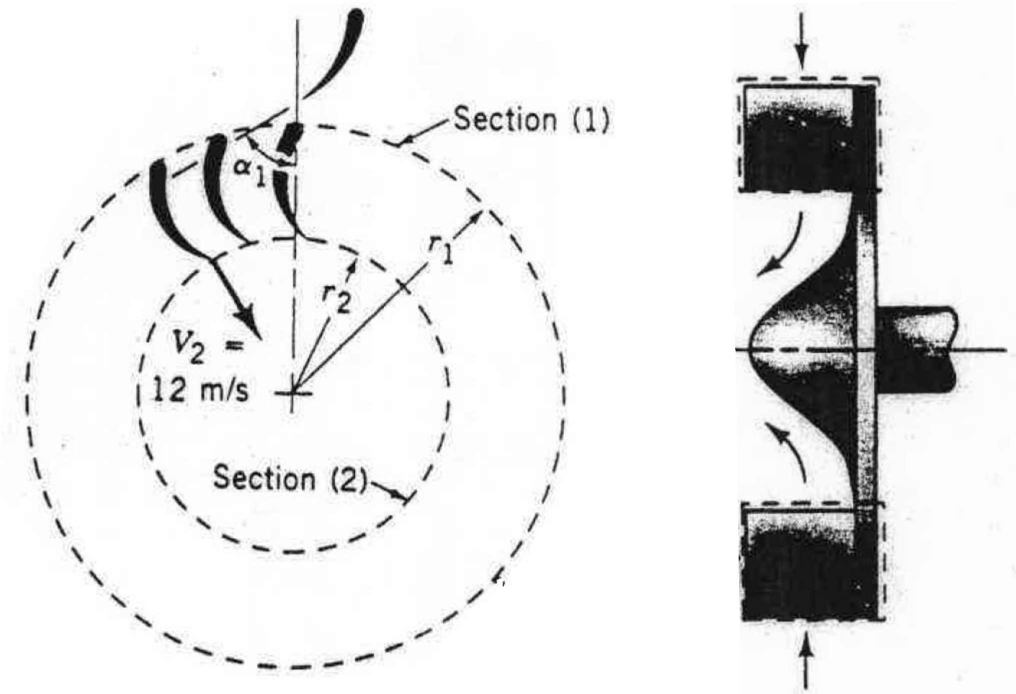

Figure 3. Question 4.

5. Fuel is flowing out from an engine nozzle with high velocity. The velocity inside the nozzle can be considered to be small ($V_1 \approx 0$) . Any height differences between the inside and outside of the nozzle can be considered negligible. The density of the fuel is ρ_l =600 kg/m³.
   a) Use the Bernoulli equation and express the fuel exit velocity V_2 as a function of the injection pressure P_{inj}, back pressure in the cylinder P_2, and density of the fuel (2p)
   b) Your task is the do the above in Matlab so that the injection pressure is between P_{inj} =100 – 900 bar and the backpressure is between 10 - 50 bar. Now, make a 2D contour plot of the injection velocity as a function of injection pressure and backpressure. Use the contourf –command. Show the coding in your answer. (4p)