Adelaide University
Department of Mechanical Engineering
Examination for the Degree of B.E.

Fluid Mechanics 2(5526)
3 hours and 10 minutes

June 2001

Students are advised to devote 10 minutes to reading the paper and planning their approach.

Students may attempt any **THREE** problems.

All problems are of equal value.

The use of notes, books and suitable calculating devices is permitted in the examination room.

The "Given/Find/Schematic/Assumptions" protocol for solutions is not required for the short parts of each problem. (1(a), 2(a), 3(a), 4(a)).

Definitions of mathematical symbols do not need to be restated if they are in common use, well known or given in this examination paper.

State all assumptions.

Acceleration due to gravity, $g = 9.81\text{m/s}^2$.

Density of air, $\rho_{air} = 1.20\text{kg/m}^3$.

Atmospheric pressure, $p_{atmos} = 101.3$ kPa.

Kinematic viscosity of air, $\nu_{air} = 14.7 \times 10^{-6}$ m^2/s.

Please turn to next page

This page is intentionally blank

Please turn to next page

Question 1

A wind tunnel has been built to study the mechanisms of boundary-layer turbulence. Measurements are made in the boundary layer developed on the hydraulically smooth internal walls of the test section, which is a straight duct with a 230×230 mm (square) cross-section and a length $L = 4.5$ m. In order to avoid contamination of the test-section boundary layer by upstream disturbances, the boundary layer at the upstream end of the test section is removed by sucking it away through a porous surface. The free stream speed of flow entering the test section is $U_0 = 4.0$ m/s.

(a) Without artificially inducing transition, **where would you expect transition from laminar to turbulent boundary layer?**

(b) **Calculate the drag coefficient (C_D), loss factor (f) and power dissipation of the test-section flow at $U_0 = 4.0$ m/s.** Assume zero streamwise pressure gradient.

(c) **What is the maximum allowable equivalent sand-grain roughness ϵ of the test-section wall?**

(d) A boundary-layer trip placed at the upstream end of the test section produces immediate transition to a turbulent boundary layer.

   (i) Assuming a zero streamwise pressure gradient, **what are the momentum (θ) and displacement thicknesses (δ^*) at the downstream end of the test section ?**

   (ii) **Estimate the free stream speed at the downstream end of the test section.** (Hint: use the displacement thickness for zero pressure gradient.)

   (iii) If the floor of the test section has a hinge at the upstream, **how far should the downstream end be jacked up or down to produce the same static pressure at each end of the test section?**

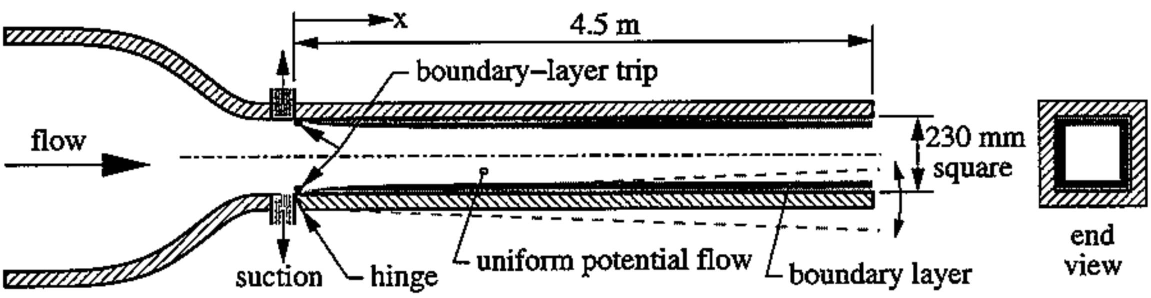

Figure 1: Boundary-layer wind tunnel.

Question 2

Figure 2 is a diagram of the fuel system for the Sydney 2000 Olympic relay torch. The fuel is a mixture of propane and butane which is stored in a state of pressurised liquid/vapour equilibrium. The storage pressure is the vapour pressure, which, at 20 °C, is $p_{vap} = 477.2$ kPa. Vapour pressure is a property of the fuel and is a function of fuel temperature. A plastic dip tube (with an inside diameter of 4 mm and a length of 180 mm) extends from the bottom to the outlet of the canister. As liquid fuel flows from the canister, liquid fuel in the canister evaporates so that the pressure inside the canister remains equal to the vapour pressure. The fuel temperature change during this process is negligible.

The fuel pipe connecting the canister to the burner is drawn copper tube with an inside diameter of 1.1 mm. Flow rate is regulated by a tiny orifice of diameter $d_o = 0.075$ mm. The length of copper tube upstream of the orifice is 70 mm. Fuel in this part of the tube is the liquid state (ρ_l, u_l). The tube carrying fuel from the orifice to the burner has a length of 230 mm. The large pressure drop across the orifice causes the fuel to vaporise downstream of the orifice. The heat required for vaporisation is provided by passing the fuel pipe through the torch flame. The fuel enters the burner at atmosperic pressure.

Some properties of Olympic torch fuel

density of liquid at 20 °C	ρ_l	559 kg/m^3
kinematic viscosity of liquid at 20 °C	ν_l	0.213×10^{-6} m^2/s
density of vapour at 50 °C	ρ_v	2.03 kg/m^3
kinematic viscosity of vapour at 50 °C	ν_v	8.40×10^{-6} m^2/s

(a) To calculate the flow rate, the design engineer assumed that the effect of the orifice is to convert all the pressure energy of the fuel into kinetic energy so that

$$\frac{p_{vap}}{\rho_l} = \frac{p_{atmos}}{\rho_l} + \frac{1}{2}u_l^2,$$

hence

$$Q = \left(\frac{\pi d_o^2}{4}\right)\sqrt{\frac{2(p_{vap} - p_{atmos})}{\rho_l}}.$$

List the features of the flow which this calculation does not take into account.

(b) **Calculate the fuel flow rate.** Assume that fuel in the tube between the orifice and the burner is a vapour at a temperature of 50 °C. Neglect all energy losses except the specific energy loss through the orifice. Use an orifice discharge coefficient of 0.7.

(c) For the flow rate which you have calculated, **calculate the pipe-flow skin-friction pressure loss in each of the two copper tubes. Why can we ignore the pressure loss in the dip tube?** Assume that the entry length for fully developed pipe flow is zero.

(d) **Estimate the total effect of the pipe-flow losses and the final exit loss on the flow rate calculated in part (b).** Ignore other local losses. As in part (a), assume $Q \propto \sqrt{\Delta p_L}$.

Please turn to next page

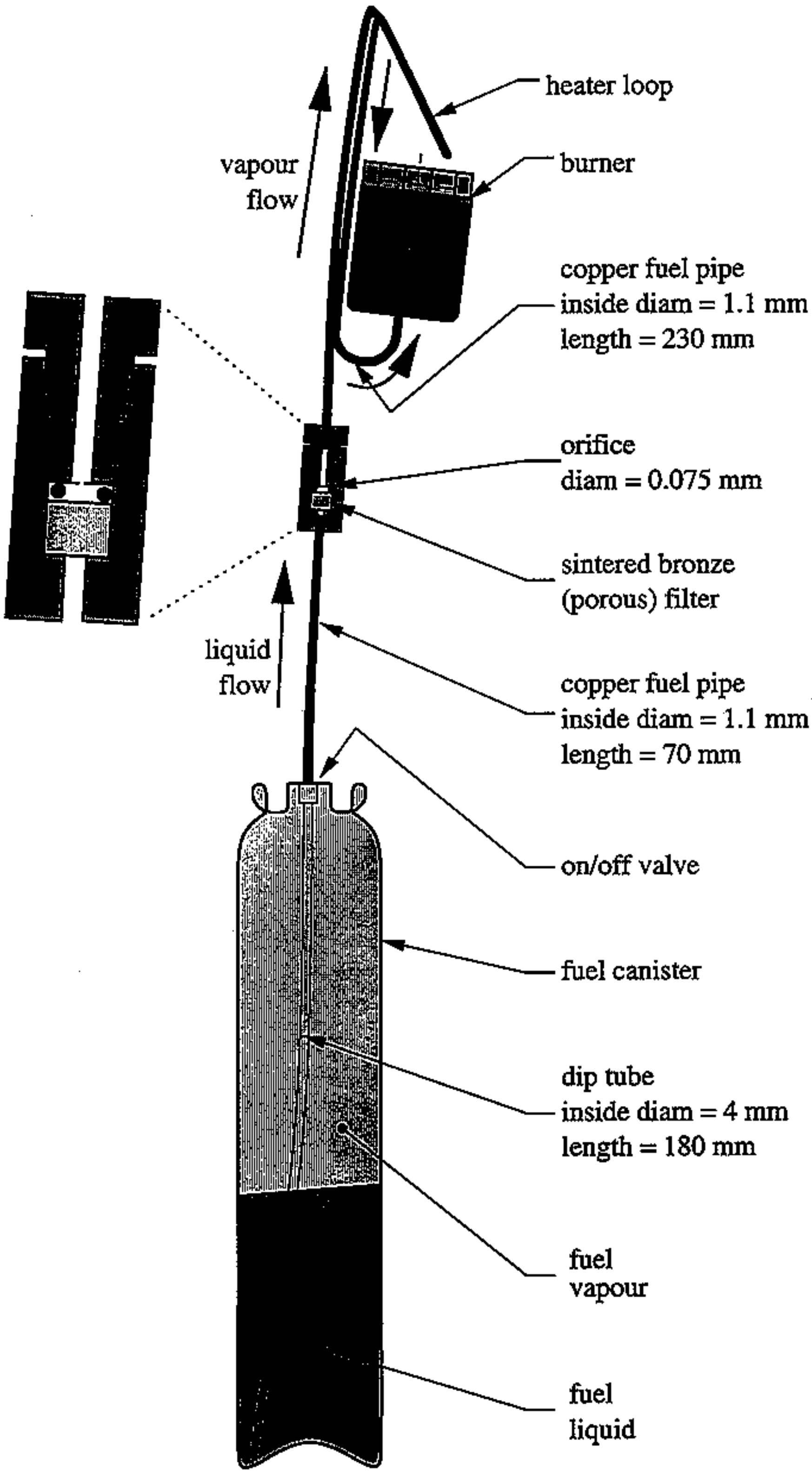

Figure 2: Fuel system of the Sydney 2000 Olympic relay torch.

Please turn to next page

Question 3

(a) Bodies immersed in a flow are usually classified as either streamlined or bluff bodies. **How should you classify the large delta-wing glider shown schematically in Figure 3 and why?**

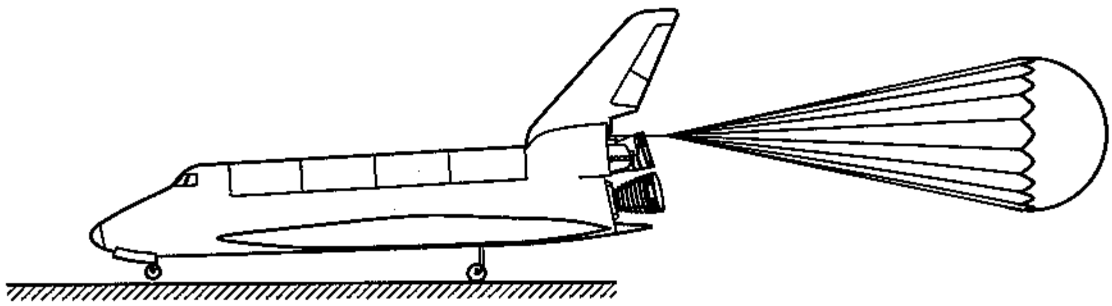

Figure 3: Parachute braking of a large delta-wing glider

(b) The glider has a rather complicated shape, so for the purpose of estimating its drag coefficient, it is modelled as a cylinder and a pair of rectangular wings with dimensions as shown in Figure 4. The boundary-layer drag coefficient of the cylinder can be considered to be the same as for a flat plate. Pressure coeffient over the nose is 0.2 (to take account of the rounded nose) and the pressure coefficient over the tail-end of the cylinder is -0.5. **Estimate the total drag coefficient of the glider** (C_{Dg}) at the landing speed of 103 m/s. The pressure coefficients for the nose and tail are based on the diameter of the cylinder, but your value for C_{Dg} should be based on the actual projected frontal area of the glider, $A_f = 62.8$ m^2.

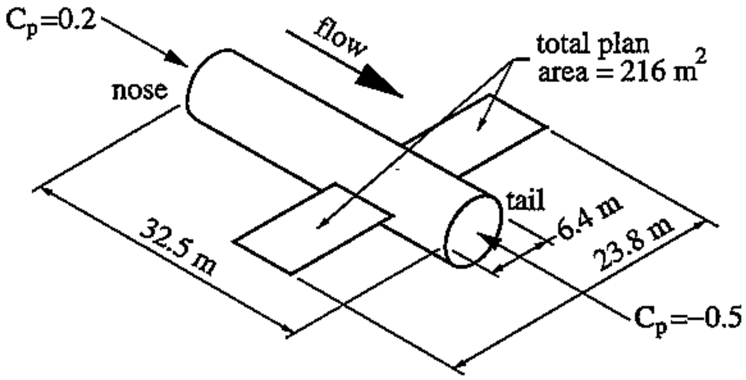

Figure 4: Simplified model of glider.

(c) The glider uses a parachute to provide the initial braking during landing. There is no time delay between "main wheels touch down" and full braking effect of the parachute. The initial landing speed is 103 m/s. At a speed of $U_1 = 50$ m/s the parachute is jettisoned and the wheel brakes are applied. Acceleration provided by the wheel brakes is a constant $a_b = -0.15g$.

Please turn to next page

(i) **At what time and distance after touchdown are wheel brakes applied?** Assume the **constant** value for drag coefficient C_{Dg} calculated in part (b).

(ii) The runway is 3.219 km (2 miles) long. **Is it long enough?**

Other information you may require:

- diameter of the parachute, $d_p = 6.0$ m
- drag coefficient of the parachute, $C_{Dp} = 1.2$
- mass of the glider, $m = 104,000$ kg
- total projected frontal area of glider, $A_f = 62.8$ m^2

(d) As the glider rolls along the runway, the vertical component of fluctuating velocity in the turbulent wake can lift sand particles from the ground. Assume that the effective vertical component of fluctuating velocity in the wake is 10% of the glider speed. **What is the size of the largest particle which can be lifted and suspended in this way at a taxying speed of 5 m/s ?** The density of sand particles is $\rho_s = 2600$ kg/m^3. Assume that the particles are spherical.

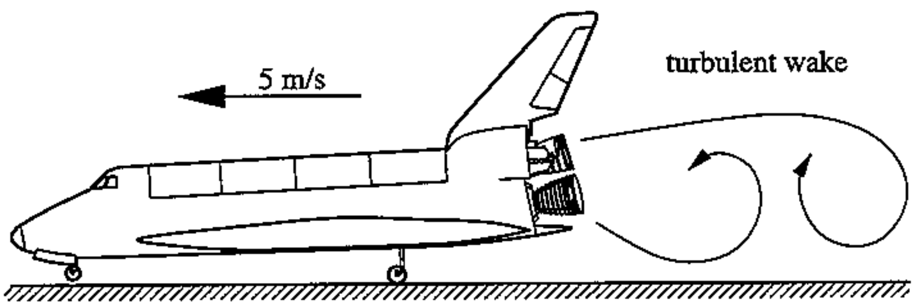

Figure 5: Turbulent wake of the glider.

Question 4

A smaller wind tunnel is required to do the CET2 "Lift, drag and pressure measurement" practical on an aerofoil. Figure 6 shows that the new wind tunnel will essentially consist of a centrifugal fan exhausting into a diffuser followed by a smooth contraction. The cross-sections of the diffuser and the contraction are square and have the dimensions shown in the Figure.

(a) The wind tunnel contains 3 stainless-steel wire mesh screens at the positions shown in Figure 6. The purpose of the screens is to maintain and improve the uniformity of the flow, and to prevent boundary-layer separation in the diffuser. The screens are woven from $d_w = 0.376$ mm diameter wire. The ratio of open area to total area of the screen is $\beta = 0.582$. The pressure drop across each screen is given by the empirical formula,

$$\frac{\Delta P}{q} = (0.52 + 66 Re_w^{-4/3}) \left(\frac{1}{\beta^2} - 1 \right),$$

where q is the dynamic pressure of the undisturbed flow speed (U), and the nominal Reynolds number of the wire is $Re_w = U d_w / \nu$. **Find an expression for the loss coefficient K_w of the screen.**

(b) One purpose of a diffuser is to convert kinetic energy of the flow into pressure energy. **Calculate the full cone angle of the diffuser.** From your lecture notes **estimate what the loss coefficient of the wind tunnel diffuser would be if it did not have screens. What does this imply about the "efficiency" or effectiveness of this (screenless) diffuser?**

(c) Figure 7 is the **static** pressure characteristic of the fan shown in Figure 6. The fan static pressure characteristic is different from a pump specific energy characteristic because it does **NOT** include the specific kinetic energy of the flow $U^2/2$ at the fan exit. Therefore, when you plot the system resistance curve to find the flow rate, you should subtract $\rho U^2/2$ at the fan exit from the system resistance curve.

(i) **Plot on Figure 6 at least two points of the system resistance curve.** Neglect boundary-layer skin friction loss. Assume that the screens suppress any separation in the diffuser. (Hint: The intended flow speed at the wind tunnel exit is 25 m/s.)

(ii) **What is the flow speed at the wind-tunnel exit when the fan motor is driven at 400 volts?**

Don't forget to hand in the graph with your answers.

Please turn to next page

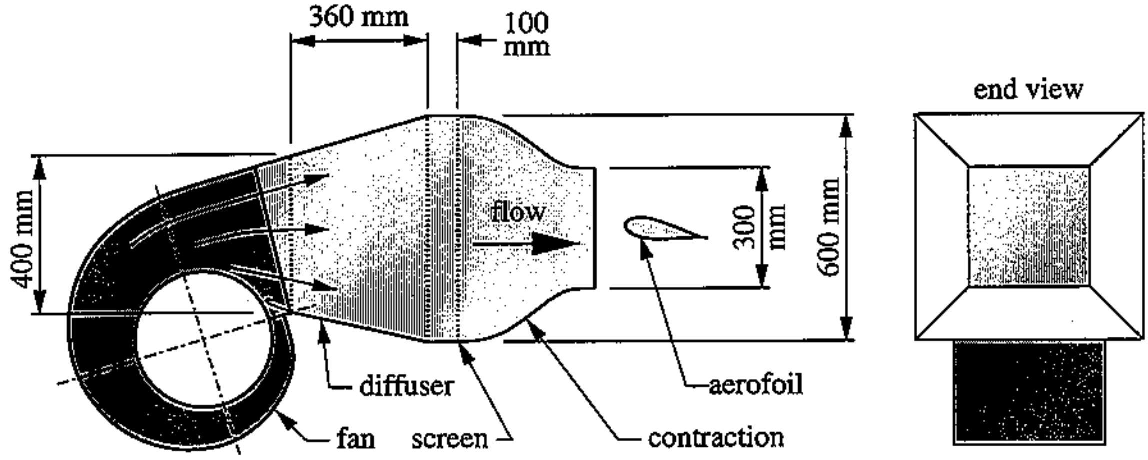

Figure 6: Wind tunnel for CET2 "Lift, drag and pressure measurement"

Please turn to next page

This page is intentionally blank

Please turn to next page

Name: Student No.:

Hand this sheet in with your answers

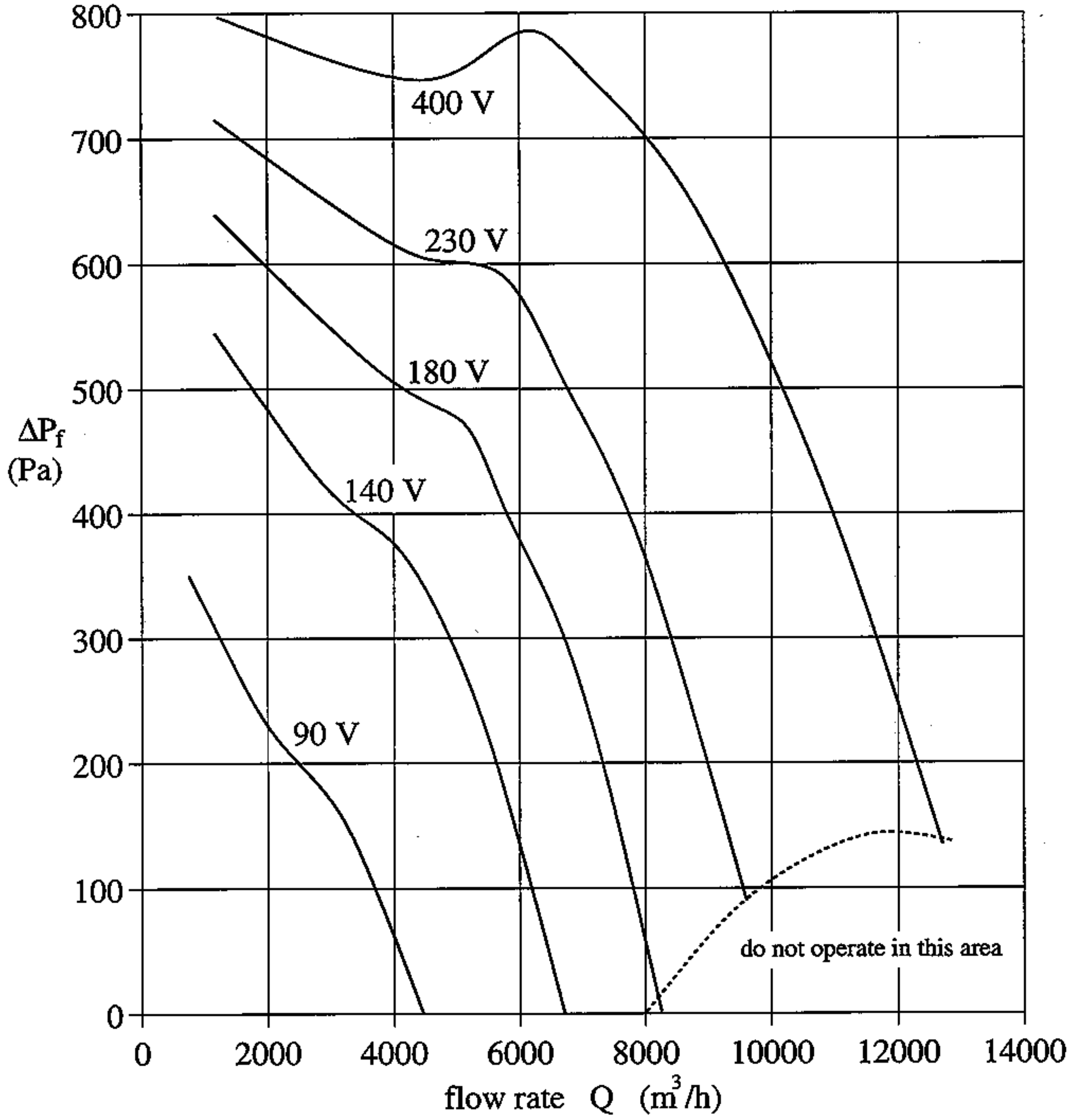

Figure 7: **Static pressure** characteristic of the wind-tunnel fan.

End of examination paper