MEMORIAL UNIVERSITY OF NEWFOUNDLAND

FACULTY OF ENGINEERING AND APPLIED SCIENCE

ENGINEERING 4913 FLUID MECHANICS I

Date: Friday 9 August 2002 — Instructor
Time: 9:00am to 11:30am — M. Hinchey

Write brief notes on any 7 of 9 of the following: (1) conservation laws (2) streamtube equations (3) external flows (4) system demand (5) pump selection (6) series vs parallel pumps (7) turbomachine swirl (8) metacenter (9) scaling laws for turbomachines. [14]

Describe briefly the pressure weight method for hydraulic gates. Also describe briefly the panel method for hydraulic gates. [14]

Describe a pressure iteration method and a flow iteration method that could be used to get the pressure and flow distribution throughout a pipe network. How could a turbine be modelled in a network? How could a pump be modelled in a network? [14]

A venturi meter can be used to measure flow rate in pipes. Derive an equation for the flow rate. Assume that all geometry is known. Also assume that high pressure minus low pressure is known. [15]

A water sprinkler can be used as a turbine. Derive an equation for its power when it has a known flow rate. Assume that all geometry is known. At what rotational speed does the power peak? [15]

A water bomber uses a pipe to pick up a load of water as it flies at a constant speed over the surface of a lake. The diameter of the pipe and the speed of the bomber are known. Derive an equation for the extra load on the bomber as it picks up water. How much power does the bomber supply to counteract this load? [14]

Describe how you would calculate the terminal speed of a blunt object falling through water. Assume that all geometry is known. Also assume that the weight of the object is known. [14]

Write brief notes on any 7 of 9 of the following: (1) conservation laws (2) streamtube equations (3) external flows (4) system demand (5) pump selection (6) series vs parallel pumps (7) turbomachine swirl (8) metacenter (9) scaling laws for turbomachines. [14]

Conservation Laws: Conservation of Mass states that the time rate of change of mass of a specific group of fluid particles in a flow is zero. Conservation of Momentum states that the time rate of change of momentum of a group of particles must balance with the net load acting on it. Conservation of Energy states that the time rate of change of energy of a group of particles must balance with heat and work interactions with its surroundings.

Streamtube Equations: For each of the conservation laws streamtube flows are assumed to be steady and uniform at each inlet and outlet. For conservation of mass density is usually assumed to be constant. For conservation of momentum applications gravity and friction forces are usually insignificant because geometry is small. For conservation of energy flows are usually taken to be adiabatic and friction is modelled as head loss. With these simplifications the streamtube equations are:

$$(CA)_{OUT} - (CA)_{IN} = 0 \qquad Q=CA \qquad \dot{M}=\rho Q$$

$$\dot{M}\,(U_{OUT}-U_{IN})=F_x \qquad \dot{M}\,(V_{OUT}-V_{IN})=F_y \qquad \dot{M}\,(W_{OUT}-W_{IN})=F_z$$

$$h_{OUT} - h_{IN} = h_S - h_L \qquad h = C^2/2g + P/\rho g + z$$

External Flows : Wall drag is most important for a long slender body like a submarine. Wake drag is most important for blunt bodies. Wave drag is usually most important for ships and high speed aircraft. Flow induced vibrations are important for slender structures in flows. Important numbers are:

$$Re = UD/\upsilon \qquad Fr = U/\sqrt{[gL]} \qquad St = [D/U]/T \qquad M = U/c$$

System Demand : This consists of two components: pressure/gravity head and head losses. On a head demand vs flow curve circumstances determine the typical operating point on it.

$$H = X + Y Q^2 \qquad Q = C A$$

$$X = \Delta (P/\rho g + z) \qquad Y = (fL/D + \Sigma K)/(2gA^2)$$

Pump Selection : One first calculates the specific speed based on the system operating point. This is a nondimensional number which does not have pump size in it: $N \sqrt{Q} / H^{3/4}$. This allows one to pick the appropriate type of pump. Next one scans pump catalogs of the type indicated by specific speed and picks the size of pump that will meet the system demand at the operating point while the pump is operating at its best efficiency point (BEP) or best operating point (BOP). Finally to prevent cavitation the pump is located in the system at a point where it has the Net Positive Suction Head or NPSH recommended by the manufacturer.

$$NPSH = P_s/\rho g + U_s U_s/2g - P_v/\rho g$$

Series vs Parallel Pumps : Pumps in series are used to get more pressure while pumps in parallel are used to get more flow.

Turbomachine Swirl : Swirl is the only component of fluid motion that has a moment arm around the axis of rotation or shaft. So it is the only one that can contribute to shaft power. The shaft power equation is:

$$\mathbf{P} = T\omega = \Delta(\rho Q \; R\omega \; V_t)$$

The swirl or tangential component of fluid velocity is V_t.

Metacenter : The metacenter M occurs at the intersection of two lines. One passes through the center of gravity or G and the center of buoyancy or B of a floating body when it is not rotated: the other is a vertical line through B when the body is rotated. Inspection of a sketch of these lines shows that if M is above G gravity and buoyancy generate a restoring moment whereas if M is below G gravity and buoyancy generate an overturning moment. One finds the location of M by finding the shift in the center of buoyancy generated during rotation and noting that this shift could result from a rotation about an imaginary point which turns out to be the metacenter. The important equations are:

$$M_W = M_V = V\ S \qquad V = \Sigma\Delta V \qquad (\Sigma d\Delta V)/V$$

$$M_W = \int r\ r\Theta\ w\ dr \qquad S = BM\ \Theta \qquad (\Sigma d\Delta A)/A$$

$$M_W = \Sigma\ r\ r\Theta\ w\ \Delta r \qquad GM = BM - BG$$

Scaling Laws for Turbomachines : Laws show how pressure and head and flow and power are changed when the size or speed of geometrically similar machines operating at the same BOP are changed. The laws are:

$$C_P = P\ /\ [\rho N^2 D^2] \qquad C_H = gH\ /\ [N^2 D^2] \qquad C_Q = Q\ /\ [ND^3]$$

$$C_{\mathbf{P}} = C_P C_Q = PQ\ /\ [\rho N^3 D^5] = \mathbf{P}\ /\ [\rho N^3 D^5]$$

Describe briefly the pressure weight method for hydraulic gates. Also describe briefly the panel method for hydraulic gates. [14]

The pressure weight method for hydraulic gates starts by surrounding the gate with vertical and horizontal lines. The fluid within these lines is considered frozen to the gate. Then the horizontal and vertical pressure forces on the lines are calculated. Force balances give the horizontal and vertical forces on the gate. If the gate has a pivot force times moment arm gives the moment about the pivot. The pressure depth law $P=\rho gh$ gives the pressure at points on surfaces. The weight equation $W=\rho gV$ gives weights of bodies of fluid. Geometry gives centroids and moment arms. The panel method for hydraulic gates starts by subdividing the gate surface into a finite number of facets or panels. The pressure depth law gives the pressure at the centroid of each panel. Pressure times panel area gives the force at the centroid. The panel unit inward normal allows one to break the force into components. A summation gives the total force on the gate in each direction. If the gate has a pivot a summation of each force times its moment arm gives the total moment about the pivot.

Describe a pressure iteration method and a flow iteration method that could be used to get the pressure and flow distribution throughout a pipe network. How could a turbine be modelled in a network? How could a pump be modelled in a network? [14]

In a pressure iteration method one would first assume pressure at each node in the network where it is not known. Then for each node one would assume pressures at the surrounding nodes to be fixed. Next for each pipe connected to the node one balances head loss with piezometric or pressure/gravity head: here pumps are treated as negative head losses while turbines are treated as positive head losses. This allows us to calculate the flow in each pipe and its direction. One then calculates the sum of the flows into the node treating flows in as positive and flows out as negative. If the $\Sigma Q>0$ then the node acts like a sink and the pressure there is too low and must be increased a bit. If the $\Sigma Q<0$ then the node acts like a source and the pressure there is too high and must be lowered a bit. Each node in the network is treated the same way. One sweeps through the network nodes again and again until the ΣQ for each node is approximately zero.

In a flow iteration method one assumes a distribution of flow which satisfies $\Sigma Q=0$ at each node in the network. The flow iteration method modifies flows throughout the network in a way which maintains $\Sigma Q=0$ at each node. In the method one identifies pipe loops in the network. Then for each loop one calculates the sum of the head losses as one moves around it in a clockwise sense. If flow in a pipe is clockwise head loss is taken to be positive whereas if flow is counterclockwise head loss is taken to be negative. For a loop if the $\Sigma h_L>0$ then there is too much clockwise flow: so flows must be reduced a bit in a clockwise sense. This decreases clockwise flows and increases counterclockwise flows. If the $\Sigma h_L<0$ then there is not enough clockwise flow: so flows must be increased a bit in a clockwise sense. This increases clockwise flows and decreases counterclockwise flows. Each loop in the network is treated the same way. One sweeps through the network loops again and again until the Σh_L for each loop is approximately zero. Special pseudo loops are used to connect reservoirs.

A venturi meter can be used to measure flow rate in pipes. Derive an equation for the flow rate. Assume that all geometry is known. Also assume that high pressure minus low pressure is known. [15]

A venturi meter uses the fact that when flow speeds up as it passes through a constriction pressure drops. This pressure drop can be measured and conservation of mass and energy allows one to derive an equation for the flow rate down the pipe. One assumes that the fluid is incompressible and that because the meter is short friction losses are insignificant. To simplify one can also assume that the pipe is horizontal. This allows us to ignore gravity. Conservation of mass gives

$$(\rho CA)_{OUT} - (\rho CA)_{IN} = 0$$

This gives for an incompressible fluid

$$C_{IN} = C_{OUT}\ A_{OUT}\ /\ A_{IN}$$

Conservation of energy gives with z_{OUT} equal to Z_{IN}

$$h_{OUT} = h_{IN}$$

$$(C^2/2g + P/\rho g)_{OUT} = (C^2/2g + P/\rho g)_{IN}$$

Substitution of C_{IN} into this gives an equation for C_{OUT} in terms of the pressure drop P_{IN} minus P_{OUT}.

$$C_{OUT} = [\sqrt{2(P_{IN}-P_{OUT})/\rho}]/[\sqrt{1-(A_{OUT}/A_{IN})^2}]$$

The flow rate is

$$Q = C_{OUT}\ A_{OUT} \qquad A = \pi D^2/4$$

A water sprinkler can be used as a turbine. Derive an equation for its power when it has a known flow rate. Assume that all geometry is known. At what rotational speed does the power peak? [15]

The power equation for any turbine is

$$\mathbf{P} = T\omega = \Delta(\rho Q\ R\omega\ V_t)$$

For a sprinkler only the tangential component of fluid speed has a moment arm about the vertical or spin axis. One can assume that at the axis the tangential speed V_t is zero. To simplify we will assume here that the sprinkler nozzle is aligned perpendicular to the radial direction. In this case the exit tangential speed is

$$V_t = Q/A \cos\Theta - R\omega = V_o - R\omega$$

where Q is the flow rate, A is the nozzle area, Θ is the nozzle angle relative to the horizontal, R is the distance out from the axis to the nozzle and ω is the sprinkler rotation rate. Substitution into the power equation gives

$$\mathbf{P} = \rho Q\ R\omega\ (V_o - R\omega)$$

Peak power occurs where $d\mathbf{P}/d\omega$ is zero. This gives

$$\omega = V_o/[2R]$$

which is half the maximum or no load speed of the turbine.

A water bomber uses a pipe to pick up a load of water as it flies at a constant speed over the surface of a lake. The diameter of the pipe and the speed of the bomber are known. Derive an equation for the extra load on the bomber as it picks up water. How much power does the bomber supply to counteract this load? [14]

We can assume that the water tank onboard the bomber is large and open to atmosphere. We can also assume that once the water gets into the tank its momentum relative to the bomber is zero. In addition we can assume that the pipe is short so that gravity and friction losses are insignificant relative to kinetic head. This means that there is no resistance to water moving into the pipe. Conservation of Energy in this case gives for a constant diameter pipe $C_{OUT} = C_{IN}$. When the water gets into the tank its C is zero. In a bomber frame of reference Conservation of Momentum gives

$$\dot{M}\,(U_{OUT}-U_{IN}) = F_x = -R_x$$

where U is velocity in the direction of flight. Because C is zero inside the tank U_{OUT} is zero. For a bomber moving from left to right with speed S, U_{IN}=-S. Substitution into momentum gives

$$\dot{M}\ S = F_x = -R_x$$

Conservation of Mass at the pipe entrance shows that

$$\dot{M} = \rho\ S\ A = \rho\ S\ \pi D^2/4$$

Manipulation gives for the force on the bomber

$$R_x = -\ \rho\ S\ \pi D^2/4\ S$$

The power needed to overcome this force is

$$\mathbf{P} = F_x\ S = \rho\ \pi D^2/4\ S^3$$

The bomber picks up water mass at the rate $\dot{M}$. This means its weight increases at g times this rate. The bomber would also have to supply the extra lift necessary to support this weight.

Describe how you would calculate the terminal speed of a blunt object falling through water. Assume that all geometry is known. Also assume that the weight of the object is known. [14]

For a blunt body terminal speed occurs when drag balances all other forces acting on the body. Under water these would be weight and buoyancy. So the force balance gives:

$$\mathbf{D} = \mathbf{W} - \mathbf{B}$$

Experimental data gives for blunt bodies

$$\mathbf{D} = C_D\ A\ \rho U^2/2$$

where C_D is a drag coefficient, A is the profile area of the body as seen from the front and U is the flow or terminal speed. The drag coefficient depends mainly on body shape. Manipulation gives:

$$U = \sqrt{2\mathbf{D}/[C_D \rho A]}$$