Fluid Flow Fundamentals Handout

Reference

- These notes partially follow the textbook Chadwick, Morfett and Borthwick Hydraulics in Civil and Environmental Engineering, published by Spon Press.
- Page references are to the 4th edition, 2004, ISBN 0-415-30609-4, but both earlier and later editions will also suffice.
- Purchase of this book is highly recommended.

Introduction and layout of the lectures

We study the flow of a Newtonian fluid (i.e. water) within circular pipes of relevant length (pipeline), i.e. several thousand of time the value of the pipe diameter.

Why?

Water service or in particular the hydraulics of water distribution systems is generally assumed to be reliable and utility customers expect high-quality service. Therefore design and operation of water systems require a deep understanding of the flow in complex systems and the associated energy losses.

Pipelines are widely and typically used in several fields of engineering:

- Generally to convoy a fluid from supply to storage reservoirs;
- Aqueducts;
- Hydroélectrique plants, i.e. penstokes, etc. ;
- Oil-ducts;
- Etc.

We will learn the basic concepts that govern the hydraulics of pipelines in order to use this knowledge in understanding the verification and design problems of water distribution systems.

Layout of the lectures 1 - 5

- Lecture 1: Review of fluid mechanics and fundamental variables
- Lecture 2: Pipe flow hydraulics, pipelines and altimetry problems
- Lecture 3: Pipelines: verification problems
- Lecture 4: Systems involving pipelines: design problems
- Lecture 5: Pipe networks and

GLOSSARY

L, *l* = pipe length	$\bar{u}$= mean stream velocity	μ = dynamic viscosity
d = pipe diameter	*Q* = flow rate	*Re* = Reynolds number
k = pipe roughness	ρ = water density	*f* = friction factor
p = relative fluid pressure	γ = specific weight	C = Chezy coefficient
z = geodetic height	*R*= hydraulics radius	β = Darcy roughness
g = gravitational acceleration	α_c = Coriolis averaging coefficient	

Lecture 1. Review of Fluid Mechanics

1 Fundamental variables and total derivatives

Consider a control volume (Euler approach) and a Cartesian reference system x,y,z, the continuous flow field of a Newtonian fluid (i.e. constant dynamic viscosity, μ [M L^{-1} T^{-1}]) is completely described by 5 variables, that are the three scalar components of the velocity vector $\vec{u} = u(x,y,z,t)\vec{\imath} + v(x,y,z,t)\vec{\jmath} + w(x,y,z,t)\vec{k}$ [L T^{-1}], the density, $\rho = \rho(x,y,z,t)$ [M L^{-3}] and the thermodynamic pressure, $p = p(x,y,z,t)$ [M L^{-1} T^{-2}, i.e. Pa=kg m^{-1} s^{-2}] here intended to be relative to atmospheric pressure. To resolve 5 variables we need 5 (partial differential) equations describing the state, the continuity and the momentum balance in each point of the flow field. As such 5 equations are written in differential form, we need to supply initial conditions and boundary conditions at the contour of the control volume.

2 What is the meaning of "total derivative"?

If we calculate the total differential of any of the fluid variables above, i.e. the velocity component along the *x* direction, *u* we have

$$du = \frac{\partial u}{\partial x}dx + \frac{\partial u}{\partial y}dy + \frac{\partial u}{\partial z}dz + \frac{\partial u}{\partial t}dt,$$

from which by dividing by *dt* we have

$$\frac{du}{dt} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t},$$

being *u* the velocity with which the infinitesimal trajectory component dx is travelled by a particle in the time dt, and so on. The time derivative (in blue) is called local derivative as it represents how variables change locally. The other derivative multiplied time the velocity components (in red) are called convective derivative and are all nonlinear. These nonlinearities are responsible for the amazing complexity and patterns that fluid particles originate when flowing. In other words, such nonlinearities are responsible for determining the evolution of flow instabilities into complex patterns, transition phenomena, etc..

In a flow, acceleration $\vec{A}(x,y,z,t)$ [L T^{-2}] in each point of the control volume is therefore defined as

$$\vec{a} = \frac{d\vec{u}}{dt} = \frac{du}{dt}\vec{\imath} + \frac{dv}{dt}\vec{\jmath} + \frac{dw}{dt}\vec{k} = \frac{\partial\vec{u}}{\partial t} + u\frac{\partial\vec{u}}{\partial x} + v\frac{\partial\vec{u}}{\partial y} + w\frac{\partial\vec{u}}{\partial z},$$

3 What is steady flow?

This course is concerned with *steady flow*. This refers to flow that does not change with *time*, i.e. the term $\frac{\partial}{\partial t} = 0$ everywhere. This does not preclude the flow varying with distance, that is convective acceleration can be nonzero. For example, if a river is wider over there than over here, the mean flow speed over there will be slower than over here, because the flow rate [L^3T^{-1}] must remain the same because of continuity (i.e. conservation of mass). Same situation occurs for flow through pipe nozzles and convergent, divergent, curves, etc. But for the flow to be steady this situation must carry on unchanged in time.

4 What is uniform flow?

Uniform flow is a specific flow field for which $\vec{a} = 0$ everywhere. This is the case of steady flow in pipes with constant section, or in flume channels with constant width, slope and roughness. Notice, that a null convective acceleration does not mean that spatial derivative of the velocity components are zero, but that their product with the velocity components is. So, a convective acceleration can be zero either because the velocity component is or because the spatial derivative is zero. A good example is flow into pipes, where the axial velocity component is clearly function of the radial position, but not of the axial one.

5 Which are the equations of motion describing fluid flow?

As aforesaid, for 5 variables (functions) we need 5 Partial Differential Equations (PDEs), which are:

- State Equation
  This equation links density change rates to pressure change rates through the volumetric compressibility coefficient, ε [$ML^{-1}T^{-2}$]

  $$\frac{d\rho}{dt} = \frac{\rho}{\varepsilon}\frac{dp}{dt}.$$

  Notice that for incompressible fluids $\varepsilon \to \infty$, which means that density and thermodynamic pressure becomes independent, i.e. density is now a known constant in the problem, and pressure is the only variable, called mechanic pressure. In other words, for incompressible fluids, the problem reduces to 4 variables and 4 equations (only).

- Continuity equation
  In local form this equation is written as
  $$\frac{d\rho}{dt} + \rho\nabla\cdot\vec{u} = 0,$$
  or in extended form
  $$\frac{\partial\rho}{\partial t} + u\frac{\partial\rho}{\partial x} + v\frac{\partial\rho}{\partial y} + w\frac{\partial\rho}{\partial z} + \rho\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right) = 0.$$
  Notice that for steady flow, this equation becomes
  $$u\frac{\partial\rho}{\partial x} + v\frac{\partial\rho}{\partial y} + w\frac{\partial\rho}{\partial z} + \rho\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right) = 0,$$
  and for incompressible fluids ($\rho = const$)
  $$\nabla\cdot\vec{u} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0,$$
  and the flow is said to be solenoidal.
  The continuity equation can be rewritten in a more useful form for currents forming a deformable flow tube with longitudinal coordinate, s, and cross section $A(s,t)$ [L^2]. In this case, by introducing the flow rate, $Q(s,t)$ [$L\,T^{-3}$] we obtain
  $$\frac{\partial(\rho Q)}{\partial s} + \frac{\partial(\rho A)}{\partial t} = 0.$$
  Again, for incompressible fluids ($\rho = const$), it reduces to

$\frac{\partial Q}{\partial s} + \frac{\partial A}{\partial t} = 0,$

which is useful to describe the unsteady motion of blood into arteries, for instance, or water into pipes under very high pressure changes (see water hammer problem in HE4). In steady conditions, the equation before reduces to

$\frac{\partial Q}{\partial s} = 0$, i.e. the volumetric flow rate, *Q* is constant along *s.*

- Momentum equation

  This equation is written in vectorial form and involves three scalar equations. The most famous one describing the flow of Newtonian fluid in a field with gravity acceleration, g [L T^{-2}] is the celebrated Navier-Stokes equation

  $\frac{d\vec{u}}{dt} + \nabla(gz) + \frac{1}{\rho}\nabla p - \frac{\mu}{\rho}\nabla^2\vec{u} = 0.$

  Notice, that all nonlinearities of such an equation are contained in the total derivative of the velocity vector. For ideal fluid with zero viscosity (or at high Reynolds number where the viscosity term can be neglected compared to others), this equation reduces to the Euler equation

  $\frac{d\vec{u}}{dt} + \nabla(gz) + \frac{1}{\rho}\nabla p = 0,$

  which after projection along a fluid trajectory, gives

  $\frac{1}{g}\frac{\partial U(s)}{\partial t} + \frac{\partial}{\partial s}\left(\frac{U^2(s)}{2g}\right) + \frac{\partial z(s)}{\partial s} + \frac{1}{\gamma}\frac{\partial p(s)}{\partial s} = 0.$

  By introducing the further hypotheses of incompressible fluid and steady flow conditions, the latter reduces to the Bernoulli Theorem, i.e.

  $\frac{\partial}{\partial s}\left(z(s) + \frac{p(s)}{\gamma} + \frac{U^2(s)}{2g}\right) = 0$, i.e.

  $H = z(s) + \frac{p(s)}{\gamma} + \frac{U^2(s)}{2g} = const.$

  Each of the terms in the above equation express an energy per unit weight of fluid, i.e. the potential (sum of the geodetic and the pressure) and the kinetic ones, respectively. This helps to represent each energy as a height, whence the reference to *H* as Total Head. Extension to currents (or streams) can be done by averaging over the power conveyed by the stream and equalizing it to that of an equivalent stream with constant velocity profile equal to the stream mean velocity, U_m, which gives

  $H = z(s) + \frac{p(s)}{\gamma} + \alpha\frac{U_m^2(s)}{2g} = const.$

  When the velocity profile is almost flat, e.g. as in turbulent flow conditions, then the averaging Coriolis coefficient, α, can practically be considered unitary.

  Last, it is interesting to rewrite the Navier-Stokes equation for a current of incompressible real fluid, which is

  $\frac{1}{g}\frac{\partial U_m(s)}{\partial t} + \frac{\partial}{\partial s}\left(z(s) + \frac{p(s)}{\gamma} + \alpha\frac{U_m^2(s)}{2g}\right) = -\frac{\tau_w}{\gamma R},$

  where *R* [L] is the hydraulic radius defined as in the next section 6, and τ_w[M L^{-1} T^{-2}] is the wall shear stress. Hence, the term on the right-hand side represents how friction forces contribute to momentum changes.

6 Hydraulic Radius

- We have been largely discussing circular pipes so far, at least implicitly. These are easy to size by diameter. However not all conduits are circular, and channels are obviously not circular. We thus have a different and more general measure, the Hydraulic Radius R. This is defined by:
  $R = \frac{A}{P}$.
- Where A is the cross-sectional area of the flow and P is the wetted perimeter, i.e. the length of the interface between solid boundary and flowing fluid (so for a channel this includes the banks and the bed, but not the water-air interface at the surface).
- It will be apparent that for a circular pipe flowing full:
  $R = \frac{A}{P} = \frac{\frac{\pi d^2}{4}}{\pi d} = \frac{d}{4}$.
- Whilst for a rectangular channel of depth *h* and breadth *b*:
  $R = \frac{A}{P} = \frac{hb}{2h+b}$.

7 What gives rise to friction in a flow?

- It is readily apparent in practical situations that flow of a fluid gives rise to frictional resistance.
- For example, we need to impart power to pump flow along, even on the level, because of friction.
- We can conceptualize friction as arising from shear stress against pipe (or channel) walls, as shown in the figure.

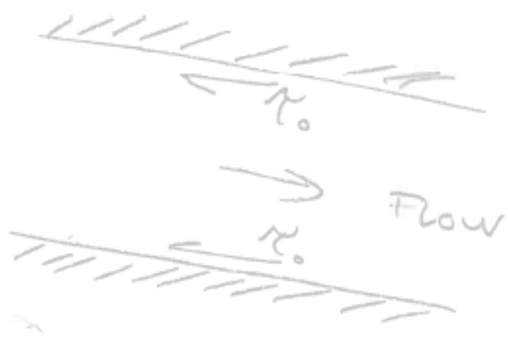

Figure 1

- However, this gives rise to a conundrum: we have already remembered, in the previous section, that there is a ‘no slip’ condition at the walls – the fluid is not moving, so friction can be doing no work at the walls.
- In fact the friction arises from resistance within the fluid as successive layers further away from the wall slide slightly faster over the preceding layer, until the area of constant velocity at the centre of the flow (or near the surface, if in a channel).

- The area where there is rapid change in velocity close to the wall is called the *boundary layer*.
- If we use the concept of a 'wall shear stress' we are doing so as a convenient, but essentially incorrect, approximation to reality that works when we are interested in the mean velocity and the flow in a pipe and not in the detail of the flow within it.
- Fortunately this is generally the case in practical civil engineering applications.
- In FM2, you obtained the Hagen-Poisouille law as an analytical solution of Navier-Stokes equations linking the energy dissipation per unit length, *j* (or S_F) into a circular pipe of diameter D, to flow characteristics and physical properties

  $j = \frac{32\mu\bar{u}}{\gamma d^2}$.

  This relation can be reworked by introducing the dissipation along a distance equal to the pipe diameter, D, normalized to the kinetic term, i.e.

  $\frac{Dj}{\frac{U_m^2}{2g}} = \frac{32\mu\bar{u}}{\gamma d^2}\frac{d}{\frac{\bar{u}^2}{2g}}$, that is

  $f = \frac{64}{\frac{\rho\bar{u}d}{\mu}} = \frac{64}{Re}$,

  where *Re* is the celebrated Reynolds number for pipe flow and f is the friction factor. Notice, that this relationship is *dimensionless,* so it describes friction losses for any pipe flow of Newtonian fluids flowing in laminar motion conditions

8 What are the fundamentals of laminar and turbulent flow?

- Laminar and turbulent flow
  - Laminar flow moves in smooth layers – *laminae*. Viscous forces predominate and small disturbances are damped out.
  - Turbulent flow is where inertial forces (i.e. the motion) becomes more significant and small disturbances are not damped out.
  - Turbulent flow looks like random, 3D eddying motions superimposed on top of laminar flow.
  - The eddies range in size from the width of the flow domain (e.g. pipe diameter) down to a size so small that it is damped out by viscosity.
- Reynolds number
  - This is the non-dimensional number, which expresses the ratio between inertial forces and viscous forces. Large Reynolds number > more significant inertial forces > more likely to be turbulent flow.
  - $Re = \frac{\rho\bar{u}d}{\mu} = \frac{\bar{u}d}{\nu}$
  - Re < 2000 *in a pipe* > laminar flow; Re > 4000 *in a pipe* > turbulent flow.
  - Different numerical limits for different flows.
  - Many other expressions of Re, eg grain Reynolds Number used in sediment transport.
- Flow profiles
  - A flow profile is a plot of flow velocity against distance across a flow, e.g. across a pipe diameter or the depth of a channel.

- In a circular pipe flowing full, the flow profile of laminar flow is parabolic in shape with the maximum flow velocity in the pipe centre:
- In turbulent flow the profile is 'flatter' – a logarithmic curve – with the velocity variation much more significant near the pipe walls.
- In each case there is a 'no slip' condition, i.e. the fluid adjacent to the wall moves at the same velocity as the wall, which for pipes in civil engineering is usually zero.
- The reason that the flow profile in turbulent flow is flatter is that the turbulent eddies mix longitudinal momentum more effectively across the flow – an eddy of slower moving fluid near the wall circulates into faster moving fluid away from the wall, dragging the faster moving fluid back a bit, and tending to accelerate itself.
- In laminar flow, momentum is transferred across the fluid only by viscous shearing between the *laminae*.
- The flow profiles sketched are for pipes, but the same change in curvature from laminar to turbulent flow also occurs in other flow patterns such as open channels.

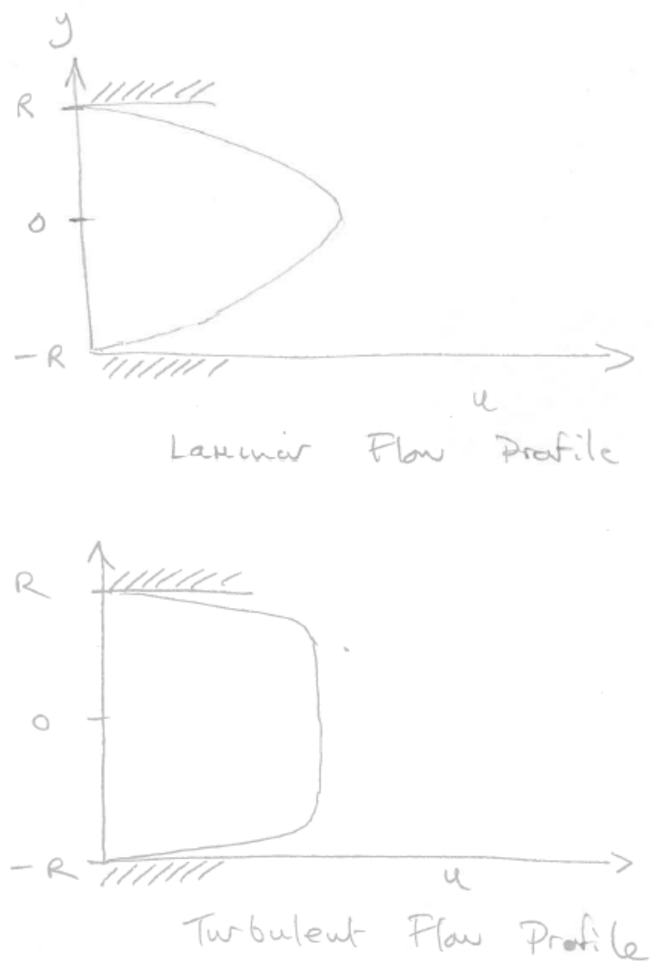

Figure 2

- Mean velocity

  - For almost all practical civil engineering purposes, we are interested in the mean flow velocity, i.e. the integral of the local velocity with respect to the distance across the pipe.
  - We call this $\bar{u}$ (occasionally u, U_m or v and many other things.)
  - The flow, discharge or flow discharge Q= $\bar{u}$A where A is the cross sectional area (see also section 5).
  - We sometimes use the term *cumecs* to mean m^3/s.
- Wall shear stress
  - The wall shear stress can always be expressed as $\tau_w = \gamma R j$

- Civil engineering flows are turbulent
  - In all practical civil engineering applications that I can imagine, flows are turbulent. If your calculations indicate laminar flows, you are almost certainly wrong.
  - You cannot tell a flow is laminar or turbulent just by looking at it. A flow can look superficially quite smooth but still be turbulent. Do not confuse turbulence with eddying, a more specific pattern of recirculation that occurs when flows pass an obstacle such as a step, or with rough, white water. Rough, white water will be turbulent, but not all turbulent flows look like this: two different concepts.
  - The fact that almost all flow are turbulent allows to easily assume a Coriolis averaging coefficient, α_c, equal to unity.

9 Why does turbulent flow give rise to more friction than laminar?

- It will be apparent from the previous section that in laminar flow the boundary layer tends to be wider, though the friction within it is still less as there are no turbulent eddies mixing momentum across it.
- In both laminar and turbulent flow, the very narrow part of the boundary layer very close to the wall is always laminar – a *laminar sub-layer*. This is because so close to a solid boundary layer, there is no space for turbulent eddies to exist – the wall supresses them.

10 Why does the pipe (or channel) *roughness* matter?

- However, it is also apparent in practice that friction is greater in a ‘rough’ pipe such as one made from concrete, than it is in a smoother pipe, such as one made from plastic or (uncorroded) steel. Why? If the fluid against the wall is stationary why would this matter?
- The answer is that the laminar sub-layer is so thin that undulations in a rough wall – which in practice are very small – still protrude through the laminar sub-layer and interfere with the boundary layer proper, causing increased resistance to the flow.
- This was quantified by Nikuradse (1894-1979) who made pipes artificially rough and conducted careful experiments (see diagram on Powerpoint or on p102 of Chadwick et al.)
- We thus conclude that friction is affected by:
  - The motion of the flow, defining the level of turbulence (and quantified by Reynolds Number)
  - The significance of the pipe wall roughness (relative to the diameter of the pipe).

- All methods of practically quantifying friction embody these principles, giving rise to various formulae

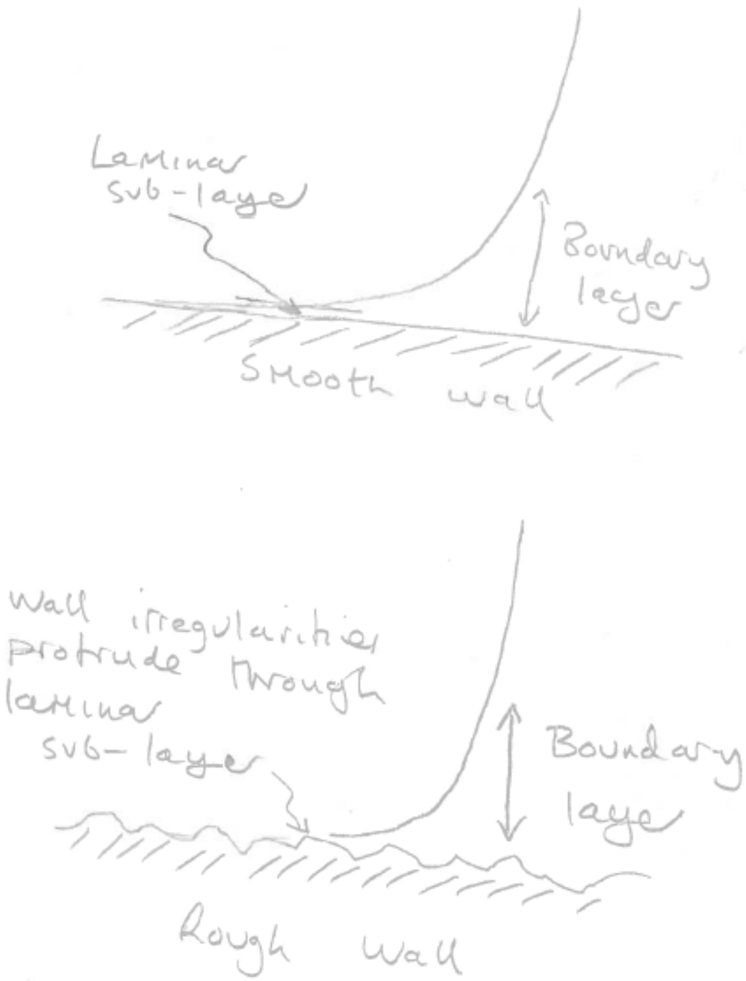

Figure 3

11 How do we express friction losses in turbulent flow?

It is interesting to obtain the structure of the wall shear stress, τ_w for a turbulent flow occurring within rough pipes. We do this by means of dimensional analysis:

$$\tau_w = f(\rho, \mu, \bar{u}, d, k_s),$$

where k_s, is the mean height of the wall roughness asperities. By choosing $\rho, \bar{u}, d$ as dimensionally independent variables, one obtains

$$\frac{\tau_w}{\rho \bar{u}^2} = \phi(Re, \frac{k_s}{d}).$$

By introducing $\tau_0 = \gamma R j$, and the friction factor, f one proves that

$$f = \psi(Re, \frac{k_s}{d}).$$

12 How do we quantify friction losses in fluid flows (for practical engineering purposes)?

- This section is fully discussed in class and the relevant formulae are on the data sheet.
- For pipe flow calculations in the UK we use the Darcy formula with friction factor f, which we quantify from the Moody Diagram, Colebrook White equation or perhaps more helpfully the Barr formula which is slightly simpler and nearly as accurate.
- Alternatively, we can use the Hydraulics Research tables, which are a tabulation of the Moody Diagram.
- For channel flow calculations, and pipe flow calculations in the USA, we use the Manning formula.
- The other formulae discussed in class are for interest, information and historic reasons, but are not recommended for practical use.

13 How do we think in terms of *head* and *head loss*?

- Like all moving things, fluid flows have potential energy and kinetic energy.
- Kinetic energy is related to their motion and is typically quantified as *velocity head (assuming $\alpha_c = 1$)*:

$$H_u = \frac{\bar{u}^2}{2g}$$

- Which is related to velocity squared as you would expect for kinetic energy, but expressed in units of length – head.
- Potential energy in fluid relates to two things: height above a datum (hence potential to fall down with gravity and so convert potential to kinetic energy), and pressure – potential to expand or squirt out of a confining container.
- Pressure head can be expressed as:

$$H_p = \frac{p}{\gamma} = \frac{p}{\rho g}$$

- Where ρ is the fluid density, whilst head due to height is expressed as an elevation above a datum, z.
- In the case of an open channel or pipe flowing partially full, the height of the free surface is related to potential energy, which is obvious if you imagine opening a sluice gate to drain a lock – the depth gets lower as water is accelerated, exchanging its potential energy in the form of depth for kinetic energy in the form of motion.
- If flow does some work which dissipates energy, for example as heat or noise, then there is clearly an energy loss which translates as a head loss. An example of head loss is due to friction as discussed above, and in this case the head loss due to friction is generally dissipated as heat.
- Civil engineers tend to use head rather than pressure or energy for convenience. Historically pressures, etc, were measured by length in a manometer or stand-pipe so head is natural measurement – but it accounts for the same basic concept.

- We will however repeatedly end up measuring something by head and still calling it energy, so beware of this.

14 How does this relate to *Bernoulli's Equation* (Section 5)

- The above discussion is readily related to Bernoulli's equation, encountered in Second Year. In terms of head, this is:

$$H = \frac{\bar{u}^2}{2g} + \frac{p}{\rho g} + z = constant$$

- along an unbroken streamline.
- In second year, a condition of applying Bernoulli was that friction losses were negligible – which is reasonable in several practical circumstances. However we can add in a term for head loss and write the following version of Bernoulli for flow along a streamline linking points 1 and 2 where there is energy loss:

$$\frac{\bar{u}_1{}^2}{2g} + \frac{p_1}{\rho g} + z_1 = \frac{\overline{u_2}^2}{2g} + \frac{p_2}{\rho g} + z_2 + h_f$$

Equation 1

- $h_f = jl$ is the head loss due to friction between the two sections distant l.
- This equation is in terms of head, but can still be viewed as energy, the first term on each side being kinetic energy and the second and third different forms of potential energy.
- Bernoulli can be derived directly from energy principles but the derivation presented in Second Year was most likely arrived at by integrating an equation of motion along a streamline.
- Note that we will repeatedly encounter head loss as a positive number – so a head gain (for example from a pump) would be a negative head loss! Beware of this also.

15 How does this relate to the Momentum Equation (2^{nd} Year)

- And finally, while on the subject of Second Year, we must remember the steady state momentum equation which we will use several times this year.
- Key here are two facts:
  - Momentum is a vector, so we consider x and y (and maybe z) directions separately, applying the equation separately for each direction; and
  - We need a *Control Volume* defining with precision the section of flow in which we are interested, and we apply momentum across the control volume.
- You must always sketch the Control Volume to be certain of doing the correct calculation.
- The momentum equation for steady state flow is then:

$$F_B + F_G + F_P = \sum \rho Q u_{out} - \sum \rho Q u_{in}$$

Equation 2

- The mean velocities u_{out} and u_{in} refer to velocities out of and into the control volume in the particular direction under consideration, and the large Σ indicates that these are summed over all inlet and outlet streams passing the control volume. F_B is the force on solid bodies in the control volume (including any pipe walls etc), F_G is the gravitational force, if any, together with other *body forces* that might apply such as, in certain circumstances, Coriolis forces or forces due to rotation of the whole system, and F_P is the force due to any pressure difference across the Control Volume. All the F terms are the forces which apply in the coordinate direction under consideration at the time.