

Lecture 10: Overview of Methods

We have (so far) seven two-way classifications of fluid flow:

Uniform $\leftrightarrow$ Non-uniform

Steady $\leftrightarrow$ Unsteady

Inviscid $\leftrightarrow$ Viscous

Laminar $\leftrightarrow$ Turbulent

Irrotational $\leftrightarrow$ Rotational

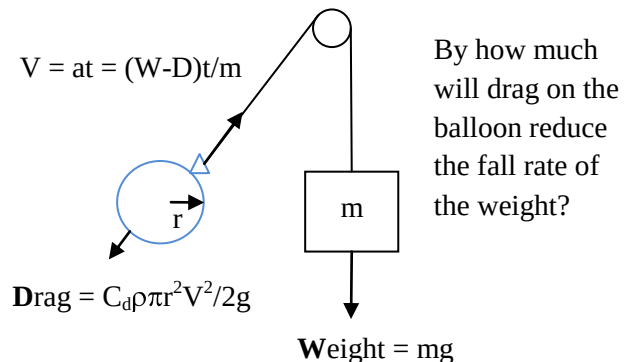
Internal $\leftrightarrow$ External

Incompressible $\leftrightarrow$ Compressible

In general flows described in the left hand column are easier to analyse and solve than those on the right. Simplified distributions and boundary conditions for the key conserved variables provide integral forms of the conservation laws that can be solved by hand. Where the distributions of flow conditions need to be specified in detail, a full analysis by partial differential equations may be required.

Sketched graphical methods are also possible for many simplified problems. It is strongly recommended that all fluid mechanics problems be expressed at first graphically, ie make a sketch of the situation with all knowns and unknowns noted, before proceeding to the maths. Sketches of this kind are known (in physics and engineering) as *free-body diagrams*:

1. Identify a suitable reference frame
2. Identify knowns and unknowns
3. Identify relevant theory
4. Manipulate and solve for unknowns

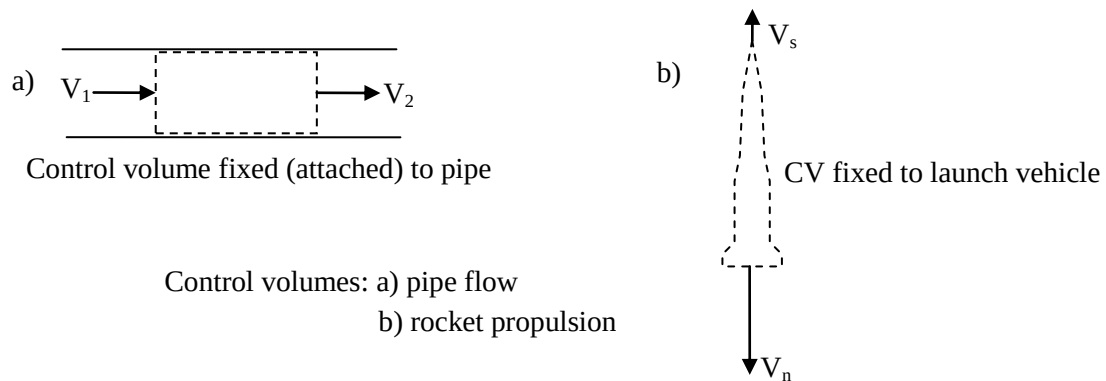


System and Control Volume Diagrams

Drawing a free-body diagram often involves a choice between a moving and a stationary reference frame. In framing the diagram we have to decide if the point of view should travel along with the fluid or if we should “observe” the system from a fixed location. Useful information can be derived from both perspectives. An easier solution may derive from one perspective or the other.

Eulerian (Stationary) Reference Frame: Control Volume methods.

A free body diagram drawn in a stationary or *Eulerian* reference frame is based on a fixed volume through which fluid flows. The volume is fixed in size as well as location, with boundaries chosen to suit the problem. This is known as a *Control Volume*:

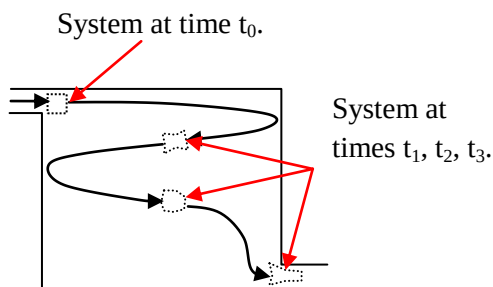


As can be seen in the case of rocket propulsion, a reference frame can be defined as stationary relative to a moving control volume. There is no absolute standard of rest or zero velocity. In modelling vehicle drag, a vehicle is a “negative” control volume over which fluid passes.

Control volumes provide points of comparison, such as upstream/downstream, at which conserved properties can be compared: for example the total rate of energy entering and leaving a control volume is the same. What happens inside a control volume is secondary – it can be treated as a “black box”.

Lagrangian (Moving) Reference Frame: System methods.

A system is a defined group of fluid elements that moves through a specified flow field. The geometry of the field may be defined by a control volume which is fixed relative to the moving system.



A moving or *Lagrangian* reference frame is attached to the system, so an observer in that reference frame would see changes such as shape, density or thermodynamic properties, as the local dynamics of that reference frame. Systems are rarely treated as black boxes, since these internal dynamics are the point of interest for this type of analysis.

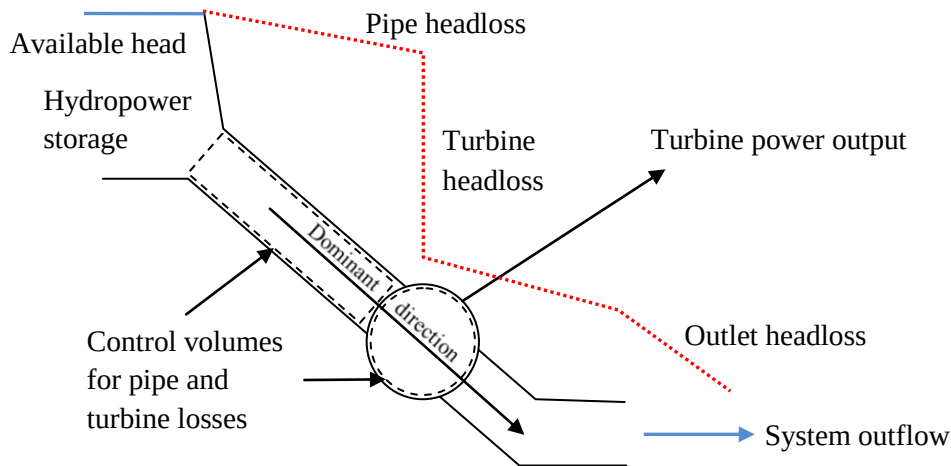
Eulerian methods are more common than Lagrangian, because the fixed reference frame is easier to visualise and more practical for problems involving fixed boundary conditions. Many theoretical solutions are derived in the Lagrangian frame and must be converted to an Eulerian frame by a suitable transformation. This is discussed further in the Integral Methods, Section 4.

One-, Two- and Three-Dimensional Methods

Often there is a dominant direction of flow, with changes along that direction dominating the dynamics of the flow. Variability in other directions may be insignificant or unimportant compared to the dominant direction. These flows can be treated as one-dimensional, with variables considered to change only on the dominant flowpath or dimension.

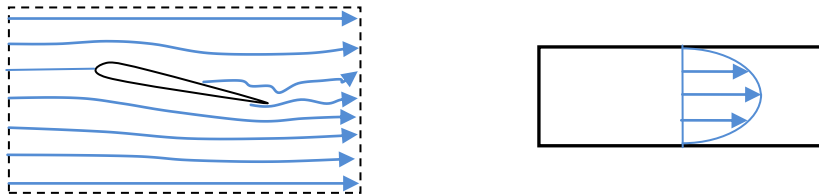
A very wide range of problems can be solved by one-dimensional methods, including unsteady pipe and/or channel flow in fully turbulent, viscous and compressible systems.

For example a turbine operates as a complex three-dimensional flow, but energy along a streamline passing through a turbine can be treated one-dimensionally:



Two-dimensional aspects of a problem, such as pipe diameters and channel depths, are often used to calculate unknowns in the dominant flowpath and thus may be regarded as parameters of a one-dimensional model rather than boundaries of a two-dimensional model. Bends in the flowpath can likewise be treated as one-dimensional attributes by reducing their effect to bend losses as a function of bend angle.

In contrast, a two-dimensional model typically involves a lateral distribution of some relevant quantity, such as a velocity profile or field, which is essential to the problem:



External flows are often treated as two-dimensional models, for example to calculate streamline profiles around a submerged object that is assumed to be uniform in the 3rd (depth) dimension. This type of modelling is relatively complex, involving potentiometric solution of two-dimensional vector fields or a full numerical solution of the Navier-Stokes equations, both beyond the scope of this unit.

Fully three-dimensional solutions of the potentiometric or Navier-Stokes equations only became possible with the advent of high-speed computing. Applications include advanced external flow modelling of aircraft and spacecraft, ship hull design, atmospheric weather and climate modelling, ocean circulation modelling, and a huge range of internal flow design applications for ducts and ducting components, building interiors and exteriors, and industrial processes.

Some two-dimensional software packages are available as free downloads or trial versions. Download and application instructions are provided in the Tutorials online.

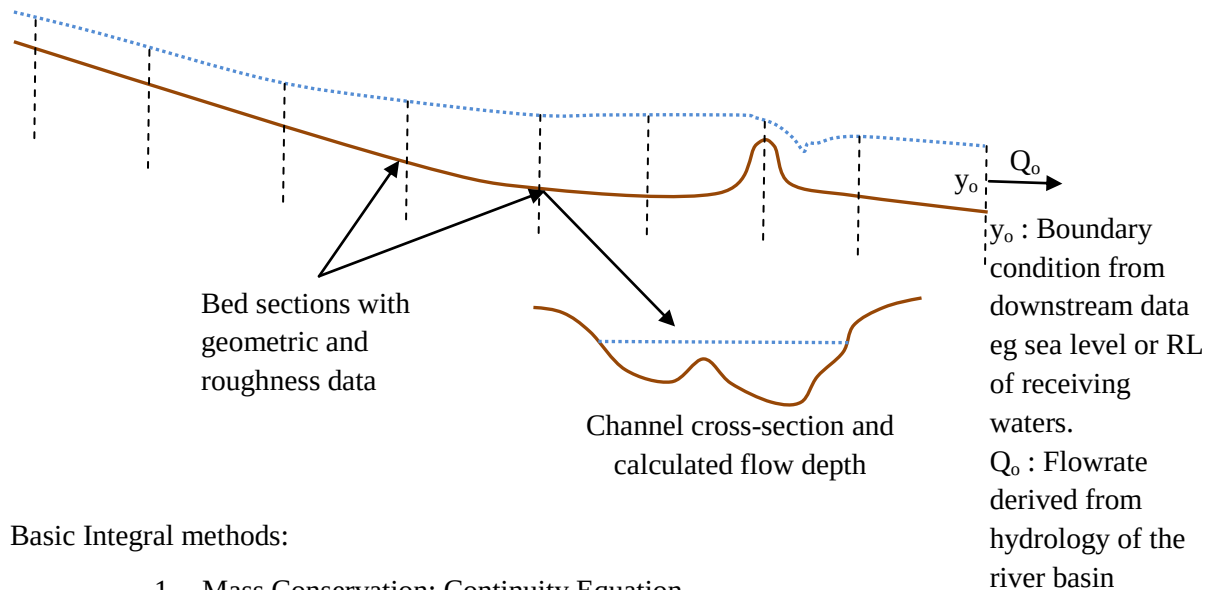
Integral and Differential Methods

One-dimensional flows have relatively simple boundary conditions, typically just the upstream or downstream average flow conditions such as velocity and depth, plus cross-sectional and roughness information. These simple inputs provide for relatively simple solution methods, known as *integral* methods, that can usually be performed by hand. Integral methods are covered in Section 4 and 6.

In general, integral methods are based on an estimate of longitudinal changes in energy and momentum by integration along the length of the flowpath. Typically the integral formula is used in its fully integrated form, ie closely or exactly summed over ranges of interest. Given information at an upstream or downstream section, conditions at the other end of a control volume can be calculated.

For example the standard step method of open channel hydraulics:

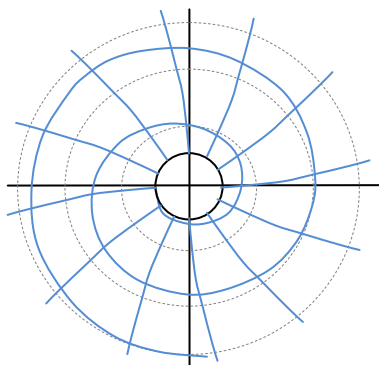
Channel long-section and calculated flow surface profile: given a downstream specification of flow depth y_o and flowrate Q_o , the flow depth at upstream cross-sections can be inferred.



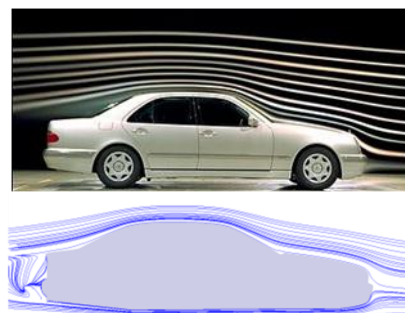
Basic Integral methods:

1. Mass Conservation: Continuity Equation
2. Energy Conservation: Bernoulli Equation, Standard Step Method
3. Momentum Conservation: Pumps and Turbines, Propulsion, Hydraulic Jumps

Two- and three-dimensional flows, on the other hand, require a more detailed specification of the boundary conditions, including shapes of internal and external bodies, pressure and velocity fields, sources of heat and so on. These flows require a *differential* treatment, where rates and derivatives in the flow are modelled by computational methods. Typically the *Navier-Stokes Equations* are used. For approximately irrotational flows, potentiometric or ideal flow methods may be accurate. Both methods are too advanced for this course.



Potentiometric solution of sink vortex

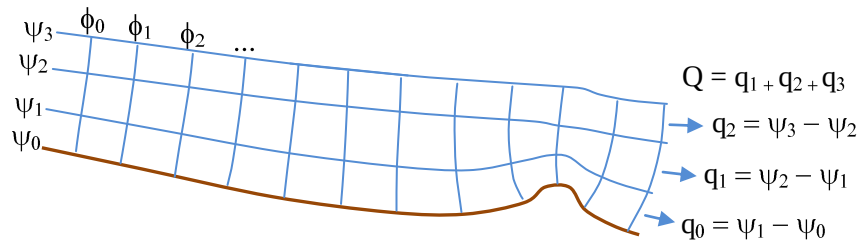


Experimental and computational solutions of vehicle drag

Ideal and Real Fluids: Potentiometric and Boundary Layer Methods

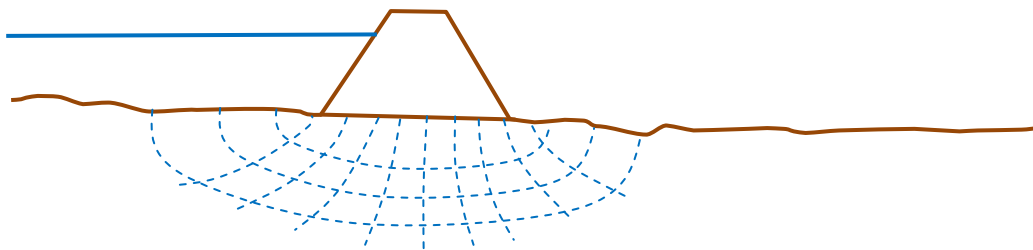
Ideal fluids, that is incompressible, irrotational flows in which viscous effects are insignificant, can be modelled by associating each streamline with a *stream number* ψ (psi) related to the continuity of the flow: starting from the boundaries of the flow, the increment of discharge from there to any streamline gives the stream number of that streamline.

A similar transformation provides a *potential number* ϕ (phi) which varies with flow velocity along a streamline. The stream and potential functions are related in a way that creates a rectilinear grid at all points in the streamflow:



Potential flows can be sketched by hand, with only a little effort required to develop a regular grid that follows the known boundary conditions. Wherever the flowlines converge, velocity is higher as per the continuity equation. This is set out in more detail in the supplementary chapter “Potential Flow” in the “Complete Lectures” online.

Potential flows are particularly useful for groundwater flows: the rate of head loss as a flow passes through porous media (eg soil) provides a basis for determining flowrates and velocities of seepage under dam walls or of recharge through pumped aquifers:



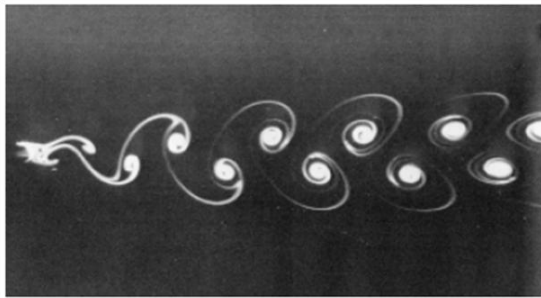
Potential flow provides an accurate description of groundwater flow under a dam wall. The flownet shown here is a first attempt and can be corrected by erasing and redrafting until the corners of the grid are nearly square.

The underlying mathematics of potential flow allow analytic solutions of a range of flow types that are approximately irrotational in nature, such as bends, sinks and simple aerofoils. Even if not necessarily accurate, the insights obtained from potential analysis are usually valuable. The mathematics are unfortunately fairly advanced. Ideal flows are not covered in this course.

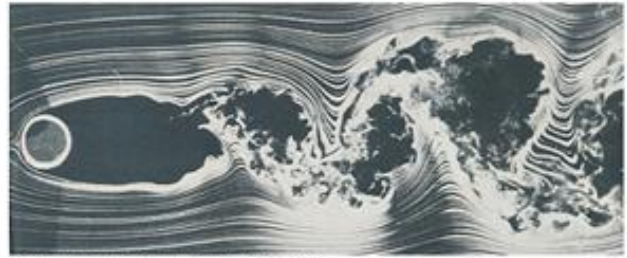
Real fluids are as the name suggests, the real thing under laboratory conditions at best, or more usually in real, uncontrolled settings. Real fluids are subject to viscous drag, rotation and compression, and the effects of unsteady and non-uniform turbulent velocity fluctuations. Real fluids have proven extremely complex to describe, through a linked experimental and theoretical process extending over many centuries.

Despite this complexity, real fluids often behave as though they are ideal in regions of uniform flow outside of the *boundary layer*. This is a narrow region of flow near any surface or boundary, where the flow is dominated by viscous effects.

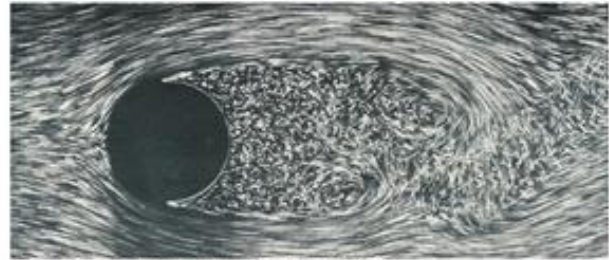
The dynamics of boundary layers are complex, involving mixed laminar, turbulent and wake-separated phases, each of which adds to the drag on a submerged body:



Karman vortex street behind a cylinder placed in uniform flow.
 $Re \sim 300$ [Courtesy: Sadotoshi Taneda; from An Album of Fluid Motion by Van Dyke (1982)]



(a) Visualizing turbulent cylinder wake at $Re = 10000$
[Courtesy: Thomas Corke and Hassan Nagib; from An Album of Fluid Motion by van Dyke (1982)]



(b) a closer look at $Re = 2000$ - patterns are identical as in (a)
[Courtesy: ONERA plc, Werle & Gallon (1972) from An Album of Fluid Motion by van Dyke (1982)]

Section 5 covers real fluids and the linked experimental-theoretical process by which they were eventually described.

Experimental Methods and Dimensional Analysis

Very few exact analytic solutions are possible in fluid mechanics, and all numerical solutions must be calibrated to real world data. Experiment is thus essential both as a basis for model calibration and as the primary stage of theoretical development.

Experiment is based on physical modelling of real flows over a range of scales, from small scale models to full scale prototypes of conduits, structures, fluid machinery, vessels, waterbodies or landscapes. Comparisons between scales must allow for the effects of various forces at different scales.

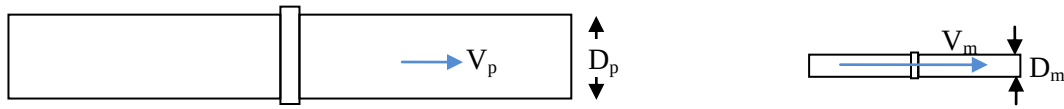
For example a landscape model has a far higher surface tension than the real landscape, because the model scale brings surface elements such as boulders and trees closer to the scale of the surface tension layer or “skin”. This affects the way the model behaves and may reduce its accuracy.

Kinematic differences are also important: relative velocities over different spatial scales create differential time scales, for example a small wave resembles a large wave at higher time rate. To reproduce the dynamics of a large wave, a small wave can be filmed and replayed at slow speed.

To compare a model with a full scale system or prototype, we need to find points of *similitude*, ie where the behaviour of the velocity and pressure fields are similar between model and prototype. Similitude is found, sometimes only approximately, by the methods of *dimensional analysis*.

In physical systems, certain dimensionless numbers recur in the description of a range of phenomena, usually indicating the involvement of important combinations of forces such as viscous and momentum forces. These numbers can be compared and equated between systems at different scales to find ranges of similar flow behaviours.

For example a large concrete pipe and a small PVC model with scaled surface roughness show similar levels of turbulence and drag characteristics when their *Reynolds Number* is the same:



Reynolds' Number $Re = \frac{VD\rho}{\mu}$: find a common value $\frac{V_p D_p \rho}{\mu} = \frac{V_m D_m \rho}{\mu}$

Prototype and model are kinematically similar when this relation is true. Thus a small pipe with a large velocity has similar drag characteristics to a large pipe with a small velocity.

Dimensional analysis is an invaluable tool for interpretation of experimental data and formulation of consistent theory. It is used in the design of scale models, such as of hulls (ships, submarines, aerospacecraft) or infrastructure (dams, pipelines, bridges) for estimation of full-scale prototype performance. Dimensional analysis is developed further in Section 5.

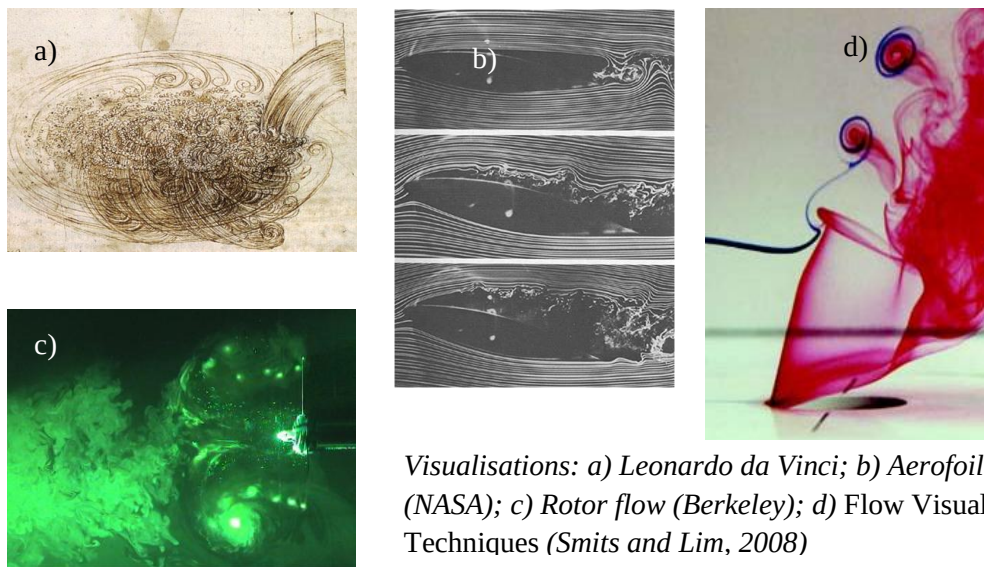
Flow Measurement

One of the key observations in any fluid experiment or control procedure is the total volumetric flow rate for a given case, often coupled with measurement of the flow velocity at particular points. A wide range of instruments for measuring fluid flowrate and velocity are available in applications ranging from river monitoring to duct gauging and control devices to aircraft airspeed indicators.

Flow measurement generally uses measurable correlations between flow rate or velocity and *head*, or the height of fluid potential above some datum. Standard devices such as venturi meters and pitot tubes, and installations such as gauging weirs and orifices, are described by tables or simple equations relating flow rates to the height dimensions of the flow. Flow measurement is covered in Section 4.

Visualisation Methods: Streamlines, Streaklines, Pathlines

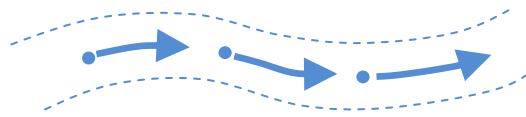
A common type of experimental design is the direct observation and imaging of flow pathlines to resolve velocity fields. Three-dimensional flows in particular produce complex, structured patterns of streamlines and eddies that may have multiple solutions in even the most detailed differential analysis: experimental studies of flow paths help to parameterise complex models and identify stable solutions.



Visualisations: a) Leonardo da Vinci; b) Aerofoil testing (NASA); c) Rotor flow (Berkeley); d) Flow Visualisation Techniques (Smits and Lim, 2008)

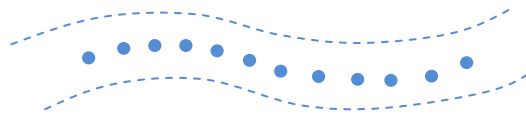
Common techniques include the use of fluorescent tracer particles, laser fluorescence, dye injections and time-lapse photography to reveal streamlines and flow velocities. Different types of flow path information can be obtained:

- A *Pathline* is the locus of points traversed by a given particle as it moves through the flow. A fluorescent time lapse photograph reveals particles as they move in that time interval. Velocity is proportional to the lengths of pathlines:



Short-interval pathlines revealed by time-lapse photography of (three) fluorescent particles

- A *Streakline* is the locus of a series of particles emerging at different times from one point, for example bubbles with neutral buoyancy injected at uniform intervals and photographed. Velocity is proportional to the spacing of bubbles.

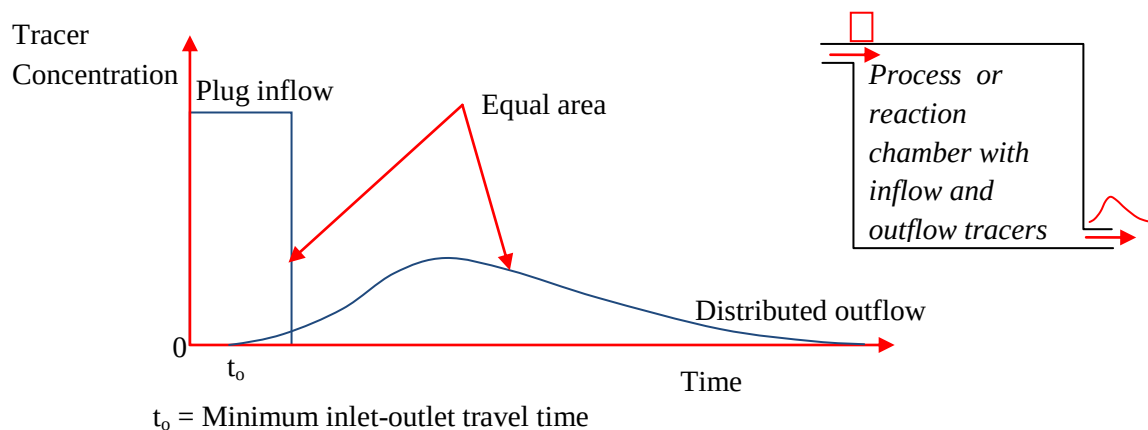


Streakline indicated by neutral-buoyancy bubbles released from an injection point

- A *Streamline* is a locus of points tangential to the flow velocity at each point. This theoretical line relates to the mathematical description of the flow and may be derived from a pathline or streakline analysis. (See the Vector Calculus Refresher.pdf in the Tutorials online)

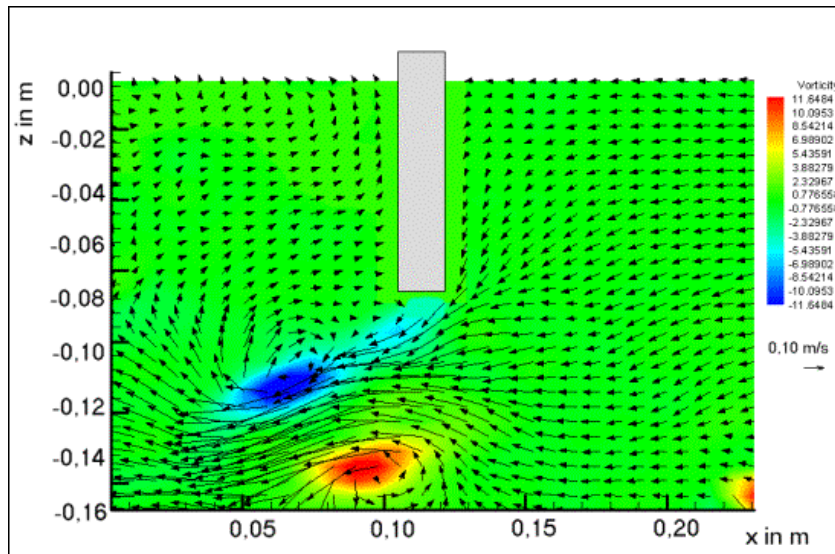
Residence Time Methods

Tracers such as saline solutions or dyes can be processed as input-output diagrams: A plug flow (essentially a uniform spike of injected tracer) transforms into a log-normal distribution as it passes through a chamber or process. The total amount of tracer is constant in the inflow and outflow, but the concentration of output varies as a function of the distribution of *travel times* within the process:



The time-distribution of outflow following a plug inflow is known as a *residence time distribution*, since it maps the net distribution of travel times within a process. Chemical reactions, thermal exchanges and mass transport (sediment etc) can also be simulated and the results related to the residence time distribution.

For example, a long tail in the distribution of a tracer outflow is indicative of stagnation points within the flow: a diminishing fraction of the tracer is retained in stagnant zones for times significantly longer than the average residence time. Stagnation may reduce the performance of chemical reaction chambers or the quality of airflow within architectural designs.



Flow field with stagnation points (“eddies”). A tracer moving through this field will tend to stay resident within eddies for longer periods than in other zones of the field.

Time-dependent processes within complex internal fluid flows can be adequately mapped for some applications using residence time distributions, without the need for a complete 2- or 3-D visualisation or simulation.

Reading Çengel and Cimbala 2006 (Reading 5) Section 1-5 to 1-10
Review Learning Outcomes for Section 3

Software See the online Tutorial “Powder Toy Software”