
1.1 What is Computational Fluid Dynamics?

"Computational fluid dynamics" is simply the use of computers and numerical techniques to solve problems involving fluid flow.

Computational fluid dynamics (CFD) has been successfully applied in many areas, including those that are the natural arena of civil engineers (highlighted below). Examples include:

- aerodynamics of aircraft and automobiles;
- hydrodynamics of ships;
- engine flows – IC engines and jet engines;
- turbomachinery – pumps and turbines;
- heat transfer – heating and cooling systems;
- combustion;
- process engineering – mixing and reacting chemicals;
- windpower;
- **wind loading – forces and dynamic response of structures;**
- **building ventilation;**
- **fire and explosion hazards**;
- **environmental engineering – transport of pollutants and effluent;**
- **coastal and offshore engineering – loading on coastal and marine structures;**
- **hydraulics – pipe networks, reservoirs, channels, weirs, spillways;**
- **sediment transport – sediment load, scour and bed morphology;**
- **hydrology – flow in rivers and aquifers;**
- oceanography – tidal flows, ocean currents;
- meteorology – numerical weather forecasting;
- high-energy physics – plasma flows;
- biomedical engineering – blood flow in heart, veins and arteries;
- electronics – cooling of circuitry.

This range of applications is very broad and encompasses many different fluid phenomena. Accordingly, many of the techniques used for high-speed aerodynamics (where compressibility is a dominant feature and viscosity comparatively unimportant) are different from those used to solve low-speed, frictional and gravity-driven flows typical of hydraulic and environmental engineering. Although many of the techniques learnt will be general, this course will focus primarily on **viscous**, **incompressible** flow by the **finite-volume** technique.

1.2 Basic Fluid Mechanics

1.2.1 Definitions

A *fluid* is a substance that continuously deforms under a shearing force, no matter how small.

Fluids may be *liquids* (having a definite volume and a free surface) or *gases* (expand to fill any container).

Fluid motion will be treated by *continuum mechanics* (not individual molecules). Note, however, that it is inter-molecular forces that give rise to viscosity.

Hydrostatics is the study of fluids at rest; *hydrodynamics* is the study of fluids in motion.

Hydraulics is the study of the flow of liquids (usually water); *aerodynamics* is the study of the flow of gases (usually air).

All fluids are compressible to some degree, but their flow can be approximated as *incompressible* (flow-induced pressure changes don't give rise to density changes) for velocities much less than the speed of sound (1480 m s^{-1} in water, 340 m s^{-1} in air).

An *ideal* fluid is one with no viscosity; it doesn't exist, but it can be a good approximation.

Real flows may be *laminar* (adjacent layers slide smoothly over each other) or *turbulent* (subject to "random" fluctuations about a mean flow). Turbulence is the natural state at high Reynolds number. The majority of engineering and environmental flows are fully turbulent.

1.2.2 Notation

<u>Geometry</u>

$\mathbf{x} \equiv (x, y, z)$ or (x_1, x_2, x_3)	position; (z is usually vertical)
t	time

<u>Field variables</u>

$\mathbf{u} \equiv (u, v, w)$ or (u_1, u_2, u_3)	velocity
p	pressure
($p - p_{atm}$ is the *gauge pressure*; $p^* = p + \rho g z$ is the *piezometric pressure*.)	
T	temperature
ϕ	concentration (amount per unit mass or per unit volume)

<u>Fluid properties</u>

ρ	density
μ	dynamic (or absolute) viscosity
$\nu \equiv \mu/\rho$	kinematic viscosity
κ	diffusivity of heat ($\kappa = k/\rho c_p$), salt, pollutant, etc.
$\gamma \equiv \rho g$	specific weight (weight per unit volume)
s.g. $\equiv \rho/\rho_{ref}$	specific gravity (or *relative density*); "ref" = water (for liquids) or air (for gases)
c	speed of sound

Dimensionless numbers

If U and L are representative velocity and length scales, respectively, then:

$\mathrm{Re} \equiv \dfrac{\rho UL}{\mu} \equiv \dfrac{UL}{\nu}$ Reynolds number (viscous flow; μ = dynamic viscosity)

$\mathrm{Fr} \equiv \dfrac{U}{\sqrt{gL}}$ Froude number (open-channel flow)

$\mathrm{Ma} \equiv \dfrac{U}{c}$ Mach number (compressible flow; c = speed of sound)

$\mathrm{Ro} \equiv \dfrac{U}{\Omega L}$ Rossby number (rotating flows; Ω = angular rotation rate)

Many other dimensionless combinations occur in fluid mechanics; see, e.g., White (2002).

1.2.3 The Fluid-Flow Equations

Statics

At rest, pressure forces balance weight. This can be written mathematically as

$$\Delta p = -\rho g \Delta z \qquad \text{or} \qquad \frac{\mathrm{d}p}{\mathrm{d}z} = -\rho g \tag{1}$$

The same equation also holds in a moving fluid if there is no vertical acceleration, or, as an approximation, if vertical acceleration is much smaller than g. If density is constant then (1) integrates to give

$$p^* \equiv p + \rho g z = constant$$

p^* is called the *piezometric pressure*; it represents the combined effect of pressure and weight. For a constant-density flow without a free surface, gravitational forces can be eliminated entirely from the equations by working with the piezometric pressure.

Thermodynamics

Pressure, density and temperature are connected by an *equation of state*. An example is the *ideal gas law*:

$$p = \rho R T \,, \qquad R = R_0/m \tag{2}$$

where R_0 is the universal gas constant, m is the molar mass and T is the absolute temperature.

Dynamics

The most important equations are those governing fluid motion. Although expressible in many different ways (Sections 2 and 3) they fundamentally represent conservation of:

- mass;
- momentum;
- energy.

The Role of the Energy Equation

The role of energy is different in compressible and incompressible flows. For compressible flow internal energy may be changed by heat input; fluid density and pressure are governed by the laws of thermodynamics and it is necessary to solve an energy equation. However, for incompressible flow, the energy equation is (as in particle mechanics) purely a mechanical-energy equation, directly derivable from – and equivalent to – the momentum equation. **For CFD of incompressible flow there is no need to solve a separate energy equation**.

Nevertheless, in theoretical work, the energy equation can be convenient. For steady incompressible flow it can be expressed as *Bernoulli's equation* along a streamline:

$$\Delta(\frac{p}{\rho} + gz + \tfrac{1}{2}U^2) = work\ done\ on\ fluid \tag{3}$$

where $\Delta(\ )$ gives the change in energy per unit mass and the RHS represents the energy (per unit mass) input by pumps or removed by turbines or friction. For compressible flows the energy per unit mass is supplemented by the *internal energy* e and (3) becomes

$$\Delta(e + \frac{p}{\rho} + gz + \tfrac{1}{2}U^2) = heat\ supplied\ to\ fluid + work\ done\ on\ fluid \tag{4}$$

The quantity $e + p/\rho$ is called the *enthalpy*.

1.3 Different Ways of Expressing the Fluid-Flow Equations

Fluid flows are governed by *conservation laws* for the transport of:

- mass;
- momentum;
- energy;
- any additional constituents;

together with *constitutive relations* (e.g. for viscous stress or heat flux) and relationships between fluid properties (e.g. the ideal gas law).

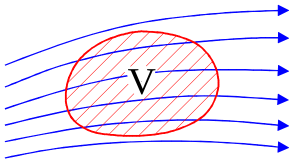
There are many different ways of expressing these physical principles mathematically.

1.3.1 Integral (or Control-Volume) Form of the Governing Equations

In continuum mechanics, conservation laws are expressed most fundamentally in *integral* ("total-amount-of") form. Here, one considers how the total amount of some physical quantity (mass, momentum, energy, …) changes within a *control volume*.

If we consider an arbitrary control volume then the total amount within that volume can only change because of:

- transport across the surface of the control volume (*flux*); or
- creation (or destruction) within the control volume (*source*).



Mass has no sources (can't be created or destroyed), but forces are the source of momentum.

In practice we do two things:

(1) We actually consider the **rate of change** of the physical property; thus:

$$\begin{pmatrix}\textbf{RATE OF CHANGE}\\ \textit{inside } V\end{pmatrix}+\begin{pmatrix}\textbf{FLUX}\\ \textit{through boundary}\end{pmatrix}=\begin{pmatrix}\textbf{SOURCE}\\ \textit{inside } V\end{pmatrix}$$

The second term is the **net outward flux** through the surface ("flow out – flow in").

(2) The *flux* (i.e. rate of transport across a surface) has two components:
advection – movement with the fluid flow;
diffusion – net transport by random (molecular or turbulent) motion.

Then, for an arbitrary control volume V:

$$\begin{pmatrix}\textbf{RATE OF CHANGE}\\ \textit{inside } V\end{pmatrix}+\begin{pmatrix}\textbf{ADVECTION}+\textbf{DIFFUSION}\\ \textit{through boundary}\end{pmatrix}=\begin{pmatrix}\textbf{SOURCE}\\ \textit{inside } V\end{pmatrix} \tag{5}$$

The important point is that there is a **single generic scalar-transport equation** of the form (5), regardless of whether the physical quantity is x-, y- or z-components of momentum, amount of pollutant or whatever. Thus, instead of dealing with lots of equations we can consider the numerical solution of the general scalar-transport equation (Section 4).

Discretisation of the integral form of the governing equations is the basis of the *finite-volume method* which is the subject of this course.

1.3.2 Differential Forms of the Governing Equations

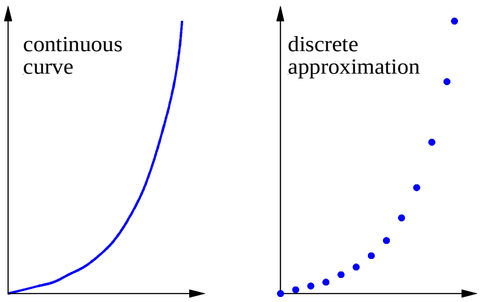
In regions without shocks, interfaces or other discontinuities, the fluid-flow equations mechanics can also be written in equivalent **differential** forms. By using spatial derivatives these describe what is going on in the vicinity of a **point** rather than over a whole control volume. Mathematically, they can be derived from the corresponding integral equations simply by making the control volume infinitesimally small.

It is also possible to derive other differential forms of the governing equations (for example, Laplace's equation for velocity potential in inviscid flow).

The differential form of the governing equations leads to the *finite-difference* method.

1.4 Basic Principles of CFD

The approximation of a continuously-varying quantity in terms of values at a finite number of points is called *discretisation*.



The fundamental elements of any CFD simulation are:

(1) The **fluid continuum is discretised**; i.e. field variables (ρ, u, v, w, p, …) are approximated by their values at a finite number of *nodes*.

(2) The **equations of motion are discretised**; i.e. approximated in terms of values at nodes:

differential or integral equations → algebraic equations
(*continuum*) (*discrete*)

(3) The **system of algebraic equations is solved** to give values at the nodes.

The main stages in a CFD study are:

Pre-processing:
– problem formulation (governing equations & boundary conditions);
– construction of a computational mesh.

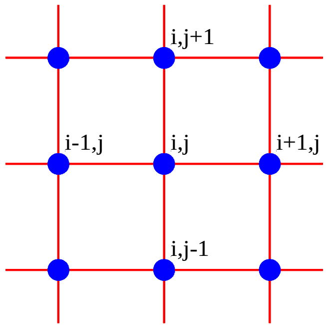
Solving:
– numerical solution of the governing equations.

Post-processing:
– plotting and analysis of results.

This is seldom a one-way process – the sequence may be repeated several times with different meshes to establish the desired accuracy, or with different values of a parameter to examine sensitivity to that variable.

1.5 The Main Discretisation Methods

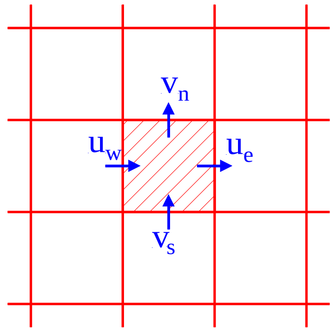
<ins>Finite-Difference Method</ins>



Discretise the governing **differential** equations directly; e.g.

$$0 \quad = \quad \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \quad \approx \quad \frac{u_{i+1,j} - u_{i-1,j}}{2\Delta x} + \frac{v_{i,j+1} - v_{i,j-1}}{2\Delta y}$$

<ins>Finite-Volume Method</ins>



Discretise the governing **integral** equations directly; e.g.

$$\textit{net mass outflow} \quad = \quad (\rho uA)_e - (\rho uA)_w + (\rho vA)_n - (\rho vA)_s \quad = \quad 0$$

<ins>Finite-Element Method</ins>

Express the solution as a weighted sum of *shape functions* – substitute into the governing equations (often in the form of a *variational principle*) and solve for the *degrees of freedom* (i.e. the weights):

$$u(\mathbf{x},t) = \sum u_m N_m(\mathbf{x},t)$$

Other specialist methods (e.g. *spectral methods*) are used in advanced theoretical work.

This course will focus almost exclusively on the finite-volume method.

The **finite-element** method is popular in **solid mechanics** (geotechnics, structures) because:

- it has considerable geometric flexibility;
- general-purpose codes can be used for a wide variety of physical problems.

The **finite-volume** method is popular in **fluid mechanics** (aerodynamics, hydraulics) because:

- it rigorously enforces **conservation**;
- it is **flexible** in terms of both **geometry** and the variety of **fluid phenomena**;
- it is directly relatable to **physical quantities** (mass flux, etc.).