The University of British Columbia
MECH 479

Module 1 - Introduction

September 1, 2022

Year 2022, Term 1 (Sept - Dec, 2021)

Module 1 – Course Objectives

- To expose students to fundamentals of computational fluid mechanics with emphasis on incompressible flows
- To make students confident of developing as well as using software related to computational fluid dynamics
- To provide in-depth mathematical insight and physical interpretation behind simplified numerical schemes for incompressible flows

Engineering Approach

In this course we shall be taking an engineering approach to CFD, and is targeted as an introductory course for the following student audiences -

- Those who would like to use CFD (commercial) – interested in gaining insights into the flow physics, and also hope to know the fundamentals
- Those who would like to apply CFD to specific problems (commercial/research) – interested in the flow physics as well as CFD method development
- Those who would like to device/design schemes to solve a new CFD need (research) – interested in the underlying method for the problem

The course offers an appreciation and insight into applying numerical methods for flow problems.

Part 1: 7 weeks (Fundamentals)

- Introduction - Governing Equations
- Finite difference scheme-Spatial discretization
- Finite difference scheme-Temporal Discretization
- Stability analysis – von Neumann method
- Solutions of heat transfer (Parabolic) and Poisson (Elliptic) PDEs
- Methods for Hyperbolic PDEs
- Solution of Euler Equations

Part 2: 6 weeks (Advanced Topics and Applications)

- Finite Volume and Finite Element Methods
- Pressure Correction Method for Incompressible Navier-Stokes equations
- Numerical Simulation of Driven Cavity
- Advanced Topics

Recommended Books

1. Anderson, J.D "Computational Fluid Dynamics", McGraw-Hill Education, 1995
2. Anderson, D.A., Tannehill, J.C., Pletcher, R.H., Computational Fluid Mechanics and Heat Transfer, McGraw Hill. 1984
3. Roach, P.J., Computational Fluid Dynamics, Hermosa Press, 1972
4. Hirsch, C., Numerical Computation of Internal and External Flows, Volume I & II, John Wiley & Sons, 1988
5. Ferziger and Peric, "Computational Methods for Fluid Dynamics" Springer Verlag
6. G.D. Smith, "Numerical solution of partial differential equations: finite difference method", Oxford University Press
7. Wesseling, "Principles of Computational Fluid Dynamics" Springer Verlag, 2002
8. Laney, C., "Computational Gas Dynamics", Camridge University Press, 1998

Chapter 1 - Introduction

The governing equations of fluid flow represent mathematical statements of the conservation laws of physics:

- The mass of a fluid is conserved;
- The rate of change of momentum equals the sum of the forces on a fluid particle (Newton's second law);
- The rate of change of energy is equal to the sum of the rate of heat addition to and the rate of work done on a fluid particle (first law of thermodynamics).

Throughout this course, the fluid will be regarded as a continuum. In other words, for the analysis of fluid flows at macroscopic length scales (say 1 m and larger), the molecular structure of matter and molecular motions will be ignored. The behaviour of the fluid will be described in terms of macroscopic properties, such as velocity, pressure, density and temperature, and their space and time derivatives. Such macroscopic behavior may be thought of as averages over suitably large numbers of molecules. A fluid particle or a fluid element is then the smallest possible element of fluid whose macroscopic properties are not influenced by individual molecules.

From the continuum fluid mechanics standpoint, the governing equations are usually the Euler equations or the full Navier-Stokes equation. Herein we briefly describe the well-known incompressible Navier-Stokes equations for a viscous fluid flow. Further details on the conservation laws and the compressible Navier-Stokes equations are provided in the slides.

Governing Equations of Incompressible Viscous Fluid

Continuity Equation

This equation is derived from the conservation of mass. It is assumed the fluid is incompressible.

$$\nabla \cdot \boldsymbol{u} = 0 \tag{1}$$

where $\boldsymbol{u}$ is the velocity vector.

In **Cartesian** coordinates, if we take $\boldsymbol{u} = u\hat{\imath} + v\hat{\jmath} + w\hat{k}$, where $[u, v, w]$ are the components of the velocity along the **Cartesian** basis vectors $\left[\hat{\imath}, \hat{\jmath}, \hat{k}\right]$, then the divergence of velocity is given by -

$$\nabla \cdot \boldsymbol{u} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \tag{2}$$

Cartesian coordinates will be mainly used in this chapter unless otherwise stated.

Momentum Equation

This equation is derived from the conservation of momentum.

$$\frac{\partial \boldsymbol{u}}{\partial t} + \boldsymbol{u} \cdot \nabla \boldsymbol{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \boldsymbol{u} \tag{3}$$

where ρ is the density, p is the pressure. Note that they both can be functions of position and time in general but for incompressible flows, the density is assumed to be a constant. ν is known as the kinematic viscosity which can also be expressed as $\frac{\mu}{\rho}$, where μ is the dynamic viscosity of the fluid.

Non-dimensional forms

We can form dimensionless variables to further simplify the equations. We let

$$[x^*, y^*, z^*] = \left[\frac{x}{L}, \frac{y}{L}, \frac{z}{L}\right]$$

$$\boldsymbol{u}^* = \frac{\boldsymbol{u}}{U}$$

$$p^* = \frac{p}{\rho U^2}$$

$$t^* = \frac{t}{L/U}$$

where L is the reference length and U is the reference velocity. Thus, the continuity and the momentum equations can be re-written as

$$\begin{gathered} \nabla \cdot \boldsymbol{u}^* = 0 \\ \frac{\partial \boldsymbol{u}^*}{\partial t^*} + \boldsymbol{u}^* \cdot \nabla \boldsymbol{u}^* = -\nabla p^* + \frac{1}{Re}\nabla^2 \boldsymbol{u}^* \end{gathered} \tag{4}$$

where $Re = \frac{U\rho L}{\mu}$ is the **Reynolds number** of the flow. Physically, it represents the ratio of the inertial (momentum) forces to the viscous forces. The asterisk (*) is often dropped for convenience. The Reynolds number is a very important parameter in CFD that defines the "ratio of convection to diffusion":

- Re $\ll 1$ implies diffusion dominated: "Stokes-like"
- Re $\gg 1$ implies convection dominated: "Euler-like"

Boundary Conditions (B.C.)

The boundary conditions would vary on a case-by-case basis and would depend on the type of flow coming into and going out of the domain being studied. We shall look at these in detail further in the course.
Apart from these, we should keep in mind the following boundary conditions on walls (or fluid-solid interfaces) - for **viscous** flows, we have $\boldsymbol{u}_{fluid} = \boldsymbol{u}_{wall}$ for the fluid on the solid wall. This condition is known as no-slip condition. For **inviscid** flows, $\boldsymbol{u}_{fluid} \cdot \boldsymbol{n}_{wall} = 0$, where $\boldsymbol{n}_{wall}$ is the normal vector at the wall. This is called the no-penetration condition, or slip condition.

Well-posed problem

Definition: A problem is well-posed when there exists one solution to the problem (Existence), which must also be the only solution (Uniqueness) and depends continuously on all the given data (Continuity).

To determine whether or not a problem is well-posed, we must consider the governing equations, the boundary conditions and the initial conditions. PDEs require proper initial conditions (ICs) and boundary conditions (BCs) to define what is known as a well-posed problem.

Classification of flow physics behaviors

For proper initial and boundary conditions (and the numerical discretization), it is necessary to classify a a well-posed mathematical model of a fluid flow into two forms:

- Equilibrium problems: The problems in this category are generally steady state situations, e.g. the steady state distribution of temperature in a rod of solid material as well as many steady fluid flows. These steady state problems are mathematically described by elliptic equations.
- Marching problems: Transient heat transfer, all unsteady flows and wave phenomena are examples of problems in the second category, the marching or propagation problems. These problems are governed by parabolic or hyperbolic equations.

Further mathematical descriptions of elliptic, parabolic and hyperbolic equations will be provided in the next module.

Some well-known special canonical flow cases

Under certain special flow conditions, the continuity and momentum equations can be simplified. This may then lead to a simple (analytical) solution. Some examples are -

- **Steady Flow**

  where $\frac{\partial \boldsymbol{u}}{\partial t} = 0$. This simplifies to a pure boundary value problem.

- **Irrotational Flow**

  where the vorticity $\boldsymbol{\omega} = \nabla \times \boldsymbol{u} = 0$. In this case, there exists a flow potential function ϕ such that $\boldsymbol{u} = \nabla\phi$ which implies that

  $$[u, v, w] = \left[\frac{\partial \phi}{\partial x}, \frac{\partial \phi}{\partial y}, \frac{\partial \phi}{\partial z}\right]$$

  We can easily check that $\nabla \times \nabla\phi = 0$ is always satisfied. In this case, the continuity condition takes the form

  $$\nabla^2 \phi = 0 \tag{5}$$

  This is also known as the Laplace equation. In 2-D, this is equivalent to

  $$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} = 0$$

while in 3-D, we have

$$\frac{\partial^2\phi}{\partial x^2}+\frac{\partial^2\phi}{\partial y^2}+\frac{\partial^2\phi}{\partial z^2}=0$$

- **1-D unsteady inviscid flow**

  from the momentum equation, we can simplify flows which can be assumed to be 1-D in the form

  $$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}=-\frac{1}{\rho}\frac{\partial p}{\partial x} \tag{6}$$

  or

  $$\frac{\partial u}{\partial t}+\frac{\partial}{\partial x}\left(\frac{p}{\rho}+\frac{1}{2}u^2\right)=0 \tag{7}$$

  The latter equation is also known as the unsteady Bernoulli equation.

- **2-D and 3-D unsteady inviscid flow**

  we can combine the continuity and momentum equation to form

  $$\nabla^2 p=-\rho\nabla\cdot(\boldsymbol{u}\cdot\nabla\boldsymbol{u}) \tag{8}$$

  which is a Poisson equation.

- **Stokes equations**

  By removing unsteady and convection term in the Navier-Stokes equations, we get the steady Stokes equations:

  $$\nabla p-\mu\nabla^2\boldsymbol{u}=0 \tag{9}$$
  $$\nabla\cdot\boldsymbol{u}=0 \tag{10}$$

  This is an appropriate fluid model for very viscous fluids (large μ) and/or for slow flows.

- **Creeping flow (Re $\ll$ 1)**

  again we can combine the continuity and momentum equation to form

  $$\nabla^2 p=0 \tag{11}$$

  which is a Laplace equation (for pressure)

What is CFD and How does it work?

CFD is a process of solving and analysing underlying physical systems involving fluid flow, heat transfer and associated phenomena via computer simulations. The tools behind CFD process are very powerful and span a wide range of industrial and non-industrial application areas to name a few, aerodynamics of aircraft, ship hydrodynamics, turbomachinery, nuclear engineering, chemical processing, environmental, meterology (weather prediction) and biomedical engineering, etc.

The foundation of CFD codes relies on numerical algorithms that can tackle fluid flow and heat transfer problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. All codes contain three key components: (i) a pre-processor, (ii) a solver and (iii) a post-processor. We will explore the functions of each of these components within the context of a commercial CFD code.

We next consider the following topics with increasing order of complexity and their usages into a commercial CFD solver:

1. Integration of ODEs and PDE classification
2. Taylor Series and Finite Difference Approximation (Strong Form)
3. Stability Analysis and Temporal Discretization
4. Weak (Integral) Form Methods and Hyperbolic PDEs
5. Training on Matlab and Commercial CFD Software
6. Advanced Numerical Discretization Techniques
7. Numerical Methods for Incompressible Navier-Stokes Equations
8. Advanced Topics