Al-Mustaqbal University

College of Engineering & Technology

Biomedical Engineering Department

Subject Name: CAD 2

4^{th} Class, Second Semester

Subject Code: [MU0114205]

Academic Year: 2024-2025

Lecturer: م.د علي كامل كريم

Email: ali.kamil.kareem@uomus.edu.iq

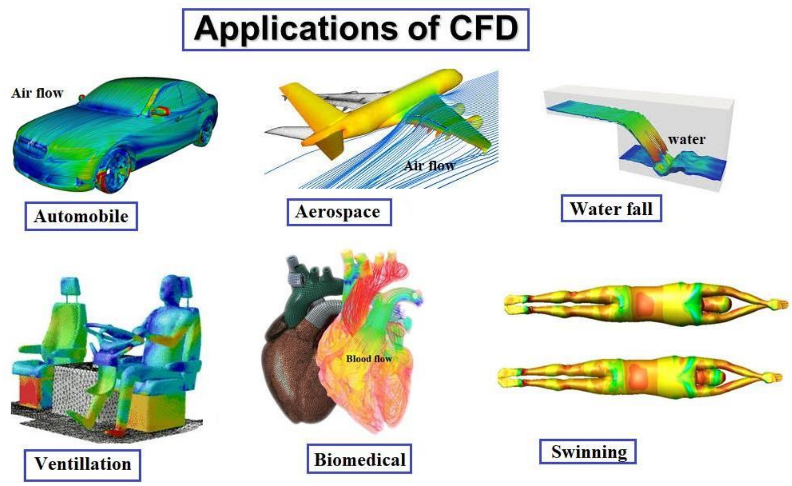
Lecture No.6

Lecture Title: [Fluid Analysis]

https://eng.uomus.edu.iq/DefaultDep.aspx?depid=13

Fluid Analysis

Introduction Flow simulations are widely used in engineering applications ranging from flow around airplane wings and hydraulic turbines to flow in blood vessels and other circulatory systems. We may gain a better understanding of the motion of fluid around objects as well as the fluid behavior in complex circulatory systems by conducting fluid analysis. Computational fluid dynamics (CFD) simulation complements experimental testing, helps reduce cost and turnaround time for design iterations, and has become an indispensable tool whenever practical design involving fluids is required.



Review of Basic Equations

We begin by briefly reviewing the fundamentals of fluid mechanics.

***1. Describing Fluid Motion:* -** In fluid dynamics, the motion of a fluid is mathematically described using physical quantities such as the flow velocity u, flow pressure p, fluid density ρ, and fluid viscosity ν. The f low velocity or flow pressure is different at a different point in a fluid volume. The objective of fluid simulation is to track the fluid velocity and pressure variations at different points in the fluid domain.

***2. Types of Fluid Flow:* -** There are many different types of fluid flow. A flow can be compressible ($\rho \neq$ constant) or incompressible ($\rho =$ constant), viscous ($\nu \neq 0$) or inviscid ($\nu = 0$), steady ($\partial u/\partial t = 0$), unsteady (($\partial u/\partial t \neq 0$) and laminar (streamline) or turbulent (chaotic). Furthermore, a fluid can be Newtonian (if the viscosity depends only on temperature and pressure, not on forces acting upon it; in other words, shear stress is a linear function of the fluid strain rate) or non-Newtonian (if the viscosity depends on forces acting upon it, i.e., shear stress is a nonlinear function of the fluid strain rate)

CAD 2

Navier–Stokes Equations

For the purpose of this', we limit ourselves to the study of incompressible Newtonian flows. All fluids are compressible to some extent, but we may consider most common liquids as incompressible, whose motion is governed by the following Navier–Stokes (N–S) equations:

$$\frac{\partial \mathbf{u}}{\partial t} = -\mathbf{u}\nabla\mathbf{u} + \nu\nabla^2\mathbf{u} - \frac{\nabla p}{\rho} + \mathbf{f}$$

which shows that the acceleration $\partial\mathbf{u}/\partial t$ of a fluid particle can be determined by the combined effects of advection $\mathbf{u}\nabla\mathbf{u}$, diffusion $\nu\nabla^2\mathbf{u}$, pressure gradient $\nabla p/\rho$, and body force f.

The N–S equations can be derived directly from the conservation of mass, momentum, and energy principles. Note that for each particle of a fluid field we have a set of N–S equations. A particle's change in velocity is influenced by how the surrounding particles are pushing it around, how the surrounding resists its motion, how the pressure gradient changes, and how the external forces such as gravity act on it.

In 3-D Cartesian coordinates, the N–S equations become:

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = \nu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) - \frac{\partial p}{\partial x} + f_x \qquad (2)$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = \nu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) - \frac{\partial p}{\partial y} + f_y \qquad (3)$$

$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = \nu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) - \frac{\partial p}{\partial z} + f_z \qquad (4)$$

where u, v, and w are components of the particle's velocity vector $\mathbf{u}$.

In CFD modeling, the N–S equations for particle motion are numerically solved, along with specified boundary conditions, on a 3-D grid that represents the fluid domain to be analyzed. For more details on numerical solution techniques adopted in CFD, please refer to the theory guide in *ANSYS* documentation.

CAD 2

3 Modeling of Fluid Flow

Practical aspects of CFD modeling are discussed next. The topics include fluid domain specification, meshing, boundary condition assignments, and solution visualization.

.3.1 Fluid Domain

A fluid domain is a continuous region with respect to the fluid's velocity, pressure, density, viscosity, and so on. Figure .2. illustrates examples of an internal flow and an external flow. For an internal flow (see Figure 2a), the fluid domain is confined by the wetted surfaces of the structure in contact with the fluid. For an external flow (see Figure 2b), the fluid domain is the external fluid region around the immersed structure.

3.2 Meshing

In CFD analysis, mesh quality has a significant impact on the solution time and accuracy as well as the rate of convergence. A good mesh needs to be fine enough to capture all relevant flow features, such as the boundary layer and shear layer and so on (see Figure.3), without overwhelming the computing resources. A good mesh should also have smooth and gradual transitions between areas of different mesh density, to avoid adverse effect on convergence and accuracy.

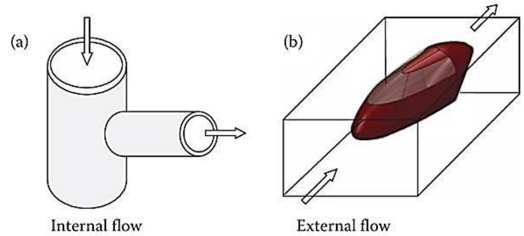


FIGURE .2
Flow region definition: (a) internal flow and (b) external flow.

.3.3 Boundary Conditions

Appropriate boundary conditions are required to fully define the flow simulation, as the flow equations are solved subject to boundary conditions. The common fluid boundary conditions include the inlet, outlet, opening, wall, and symmetry plane . An inlet condition is used for boundaries where the flow enters the domain. An outlet condition is for where the flow leaves the domain. An opening condition is used for boundaries where the fluid can enter or leave the domain freely. A wall condition represents a solid boundary of the flow model. For fixed walls, this typically means a no-slip boundary condition for the flow velocity. A symmetry plane condition is used for planes exhibiting both geometric and flow symmetry.

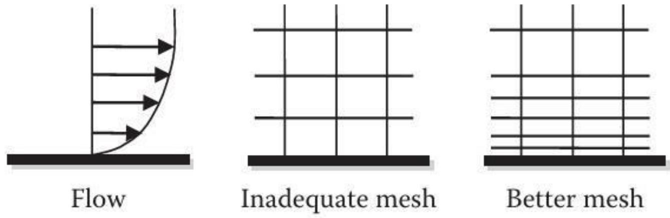
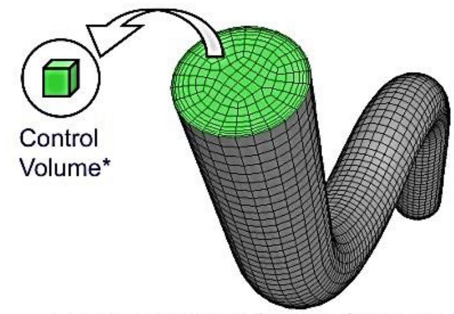


FIGURE .3
The resolution of a mesh needs to adequately represent the flow feature.

CAD 2

- **ANSYS CFD solvers are based on the finite volume method**
  - Domain is discretised into a finite set of control volumes
  - General conservation (transport) equations for mass, momentum, energy, species, etc. are solved on this set of control volumes



Fluid region of pipe flow is discretised into a finite set of control volumes.

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi\, dV}_{\text{Unsteady}} + \underbrace{\oint_A \rho\phi \mathbf{V}\cdot d\mathbf{A}}_{\text{Convection}} = \underbrace{\oint_A \Gamma_\phi \nabla\phi \cdot d\mathbf{A}}_{\text{Diffusion}} + \underbrace{\int_V S_\phi\, dV}_{\text{Generation}}$$

Equation	Variable
Continuity	1
X momentum	u
Y momentum	v
Z momentum	w
Energy	h

  - Partial differential equations are discretised into a system of algebraic equations
  - All algebraic equations are then solved numerically to render the solution field