

**University of Anbar
College of Engineering
Mechanical Engineering Department
Postgraduate Studies**



Computational Fluid Dynamics

By

Dr. Mohammed Abed Ahmed

References :

1. Numerical Heat Transfer and Fluid Flow, suhas V. Pantankar, 1980
2. Computational Fluid Dynamics, Klaus A. Hoffmann and Steve T. Chiang, 2000.
3. Computational Fluid Dynamics, the basics with applications, John D. Anderson, J.R., 1995.
4. Computational Fluid Dynamics, An Introduction, John F. Wendt (Ed.), 2009.
5. Computational Fluid Mechanics and Heat Transfer, John C. Tannehill et. al., 1997.
6. An Introduction to Computation Fluid Dynamics, The Finite Volume Method, H. K. Versteeg and W. Malalasekera, 2007

Chapter one : Introduction

Fluid dynamics is a field of science which studies the physical laws governing the flow of fluids under various conditions. Great effort has gone into understanding the governing laws and the nature of fluids themselves, resulting in a complex yet theoretically strong field of research.

Computational Fluid Dynamics, usually abbreviated as CFD, is used to generate flow simulations with the help of computers. CFD involves the solution of the governing laws of fluid dynamics numerically. The complex set of partial differential equations are solved on a geometrical domain divided into small volumes, commonly known as a mesh (or grid).

CFD has enabled us to understand the world in new ways. We can see what it like to be in a furnace, model how blood flows through our arteries and veins. CFD enables analysts to simulate and understand fluid flows without

the help of instruments for measuring various flow variables at desired locations.

1.1 Comparison of Experimental, theoretical, and computational Approaches:-

There are basically three approaches or methods that can be used to solve a problem in fluid mechanics and heat transfer. These methods are:-

1. Experimental
2. Theoretical (analytical)
3. Computational (CFD)

The advantages and disadvantages of these approaches are summarized as:

Approach	Advantages
Experimental	1. Capable of being most realistic.
Analytical	1. Clean, general information, which is usually in formula form.
Computational	<ol style="list-style-type: none"> 1. No restriction to linearity 2. Complicated physics can be treated. 3. Time evolution of flow can be obtained.

Approach	Disadvantages
Experimental	<ol style="list-style-type: none">1- Equipment required2- Scaling problems3- Tunnel corrections4- Measurement difficulties5- Operating costs
Analytical	<ol style="list-style-type: none">1- Restricted to simple geometry and physics2- usually restricted to linear problems
Computational	<ol style="list-style-type: none">1- Truncation errors2- Boundary condition problems3- Computer costs

1.2 CFD Analysis process: -

The general process for performing a CFD analysis is outlined as:

- 1- Formulate the flow problem:

The first step of the analysis process is to formulate the flow problem by seeking answers to the following questions:

- what is known about the flow problem to be dealt with?
- what physical phenomena need to be taken into account?
- what is the geometry of the domain and operating conditions?
- Are there any internal obstacles or free surfaces?
- What is the type of flow (laminar/turbulent, steady/unsteady)?
- what is the objective of the CFD analysis to be performed?
- what is the easiest/cheapest/fastest way to achieve the goal?

2- Mathematical model:

- choose a suitable flow model and reference frame.
- Identify the forces which cause and influence

the fluid motion.

- Define the computational domain in which to solve the problem.
- Formulate conservation laws for the mass, momentum, and energy.
- Simplify the governing equations to reduce the computational effort:
 - * use available information about the prevailing flow regime.
 - * check for symmetries and predominant flow directions (1D, 2D, 3D)
 - * neglect the terms which have little or no influence on the results.
- Specify initial / boundary conditions.

3- Grid generation:

A grid is generated within the flow domain.

- The grid consists of finite-volume cells on which the CFD equations are approximated.
- Grids can be structured or unstructured.

4. Discretization process:

The PDE system is transformed into a set of algebraic equations.

- space discretization (approximation of spatial derivatives)

 - * finite differences / volumes / elements

 - * high- vs. low-order approximations

- Time discretization (approximation of temporal derivatives)

 - * explicit vs. implicit schemes.

5. Iterative solution strategy:

The coupled nonlinear algebraic equations must be solved iteratively

6. CFD simulations:

The computing times for a flow simulation depend on

- The choice of numerical algorithms and data structures.

- Stopping criteria for iterative solvers.

- discretization parameters (mesh quality, mesh size, time step)
- programming language (most CFD codes are written in Fortran)

7- postprocessing and analysis

Postprocessing of the simulation results is performed in order to extract the desired information from the computed flow field

- Domain geometry and grid display
- vector plots
- line and shaded contour plots
- validation of the CFD model