DEPARTMENT OF AEROSPACE ENGINEERING

Faculty Name	:	**Dr.A.Arun Negemiya, ASP/ Aero**	Academic Year	:	**2024-2025 (Even)**
Year & Branch	:	**III AEROSPACE**	Semester	:	**VI**
Course	:	**19ASB304 - Computational Fluid Dynamics for Aerospace Application**			

UNIT I - FUNDAMENTAL CONCEPTS

Overview of Computational Fluid Dynamics and its Applications

Introduction to Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is the science of calculation of fluid flow and related variables using a computer. Usually, the fluid body is divided into cells or elements forming a grid. Then the equations for unknown variables are solved for each grid. This requires a good amount of computing resources.

Computational Fluid Dynamics (CFD) is the technique of replacing the Partial Differential Equations (PDE) governing the fluid flow by a set of algebraic equations and solving them using a digital computer.

The transport equations that govern the fluid motion or flow are,

1. Continuity Equation
2. Momentum Equation
3. Energy Equation

Application of CFD

- Aerodynamics of aircraft and vehicles
- Hydrodynamics of ships
- Power plants

- Turbomachinery
- Electrical and electronics engineering
- Chemical process engineering
- External and internal environment of the building
- Marine engineering
- Environmental engineering
- Hydrology and Oceanography
- Meteorology
- Biomedical engineering

The availability of affordable high-performance computing hardware and the introduction of user-friendly interfaces have led to a tremendous increase in CFD usage in the industrial community.

Approaches to studying Fluid Mechanics

There are three approaches to study the fluids and their behavior they are,

1. **Analytical Method**

   In this approach, the transport equations that govern the fluid flow are solved mathematically to obtain specific solutions for various problems.

2. **Experimental Method**

   This approach is based on utilizing the experimental setups to carry out the experimentation and to get the flow characteristics by flow visualization or by data acquisition to understand the phenomenon.

3. **Computational Method**

   In this method, the numerical approach is used to solve the governing equations of the fluid motion to obtain numerical solutions for the physical problems.

Major advantages of CFD over the Experimental Fluid Dynamics

1. Leas time in design and development is significantly reduced
2. CFD can simulate flow conditions which are not possible in experimental model tests

3. CFD can provide more detailed and comprehensive information

4. CFD is more cost-effective than the experimental setup

5. CFD utilizes low energy

Components of a CFD code

CFD codes are the structured numerical algorithms that are used to solve fluid flow problems. To provide easy access to their solving power all the commercial CFD codes include the three main elements.

- **A Pre-processor**
- **A Solver**
- **A Post-processor**

A Pre-processor

The main activity in the pre-processor is to give input to the code in the form suitable for the solver by using the Graphical User Interface (GUI), the main activities in the pre-processor are,

- Definition of geometry and the region of interest
- Grid generation
- Definition of fluid properties
- Specification of boundary conditions
- Specification of solver criteria

A Solver

A solver takes the input from the preprocessor and forms the governing equations as PDEs, which are again converted into algebraic form to solve the equations by the numerical approach. The numerical approach's iterations stop when the solver criteria in the preprocessor match.

A Post-processor

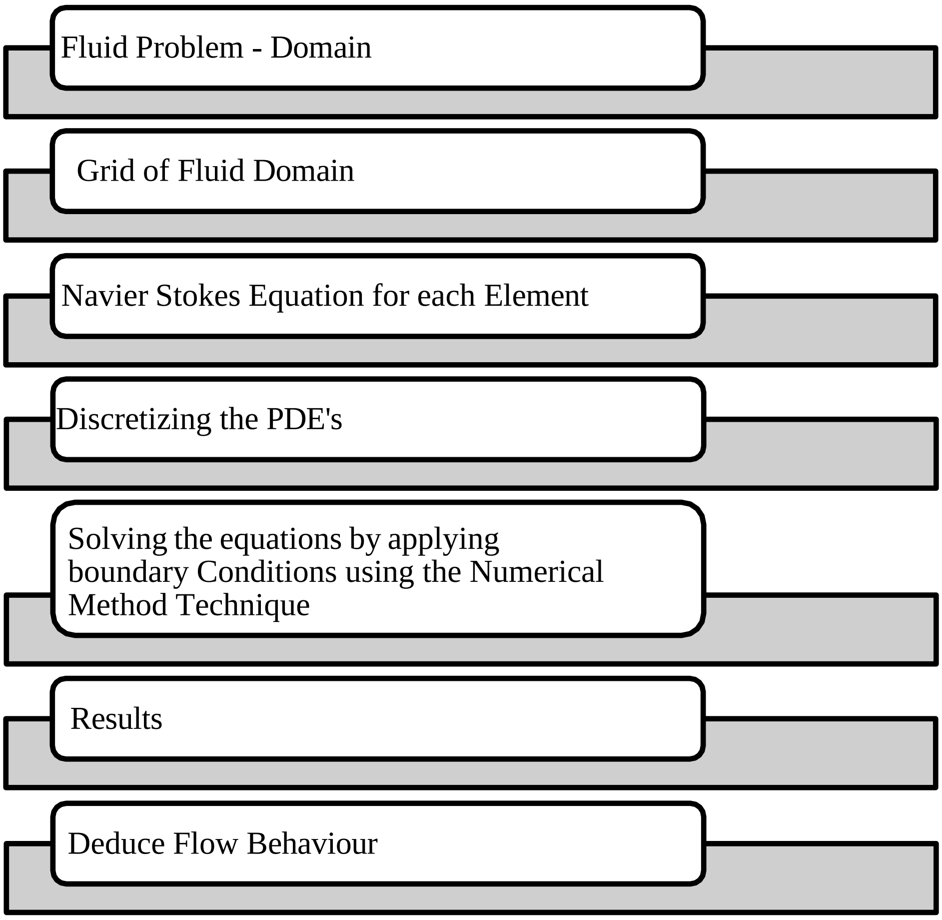
A post-processor takes the raw data from the solver and displays the results in the form the user desires. The features of the post-processor includes

- Displaying geometry and grid
- Vector and Contour plots

- Particle tracking
- Pressure, Temperature, and Density data

The exponential growth of the speed, memory, and computing power of digital computers in the modern era has led to the extensive development of many CFD codes which are generally used by the academia, industry, and research fields. Some of the commercial CFD codes generally used are **CFX, FLUENT, COSMOFLOW, STAR-CD, and FLOW-3D.**

Overview of Computational Fluid Dynamics



Overview of CFD

The complete steps followed in the CFD process are illustrated in the above figure.

The steps are described below,

1. The fluid problem is taken and the physics of the problem is studied and based on it the fluid domain is created.

2. The fluid domain is divided into finite grids

3. For each grid element the governing equations that govern the fluid flow are formed.

4. The PDEs governing the fluid domain are discretized into the Algebraic equations.

5. These algebraic equations are solved by applying the boundary conditions using the numerical method technique.

6. From the results of the algebraic equations the flow behavior is predicted.