SNS COLLEGE OF TECHNOLOGY

Coimbatore-35
An Autonomous Institution

Accredited by NBA – AICTE and Accredited by NAAC – UGC with 'A++' Grade
Approved by AICTE, New Delhi & Affiliated to Anna University, Chennai

DEPARTMENT OF AEROSPACE ENGINEERING

19ASB304 – COMPUTATIONAL FLUID DYNAMICS FOR AEROSPACE APPLICATIONS

III YEAR VI SEM

UNIT-III FINITE ELEMENT TECHNIQUES

TOPIC: Finite Element Techniques in Computational Fluid Dynamics

NAME: Mr.N.Venkatesh., M.Tech
Assistant Professor
Aerospace Engineering
SNS College of Technology

What is CFD?

- Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process

→ We are interested in the **forces** (pressure , viscous stress etc.) acting on surfaces (**Example: In an airplane, we are interested in the lift, drag, power, pressure distribution etc)**

→ We would like to determine the **velocity field** (**Example: In a race car, we are interested in the local flow streamlines, so that we can design for less drag)**

→ We are interested in knowing the **temperature distribution (Example: Heat transfer in the vicinity of a computer chip)**

Advantages

- Relatively low cost.
  - CFD simulations are relatively inexpensive, and costs are likely to decrease as computers become more powerful.
- Speed.
  - CFD simulations can be executed in a short period of time.
- Ability to simulate real conditions.
  - CFD provides the ability to theoretically simulate any physical condition.
- Comprehensive information.
  - CFD allows the analyst to examine a large number of locations in the region of interest, and yields a comprehensive set of flow parameters for examination.

Limitations

- The CFD solutions can only be as accurate as the physical models on which they are based.
- Solving equations on a computer invariably introduces numerical errors.
  - Round-off error: due to finite word size available on the computer. Round-off errors will always exist (though they can be small in most cases).
  - Truncation error: due to approximations in the numerical models. Truncation errors will go to zero as the grid is refined. Mesh refinement is one way to deal with truncation error.
- Boundary conditions.
  - As with physical models, the accuracy of the CFD solution is only as good as the initial/boundary conditions provided to the numerical model.

Purpose and Aim

- Main purpose is Simulation-based design instead of “build & test” hence saving a lot of time since it enables easy repetitions.
  For example combustion explosions(unrepeatable).
- Aim is the simulation of physical fluid phenomena that are difficult for experiments
  - Full scale simulations (e.g., ships and airplanes)
  - Environmental effects (wind, weather, etc.)
  - Hazards (e.g., explosions, radiation, pollution)
  - Physics (e.g., planetary boundary layer, stellar evolution)

How it works?

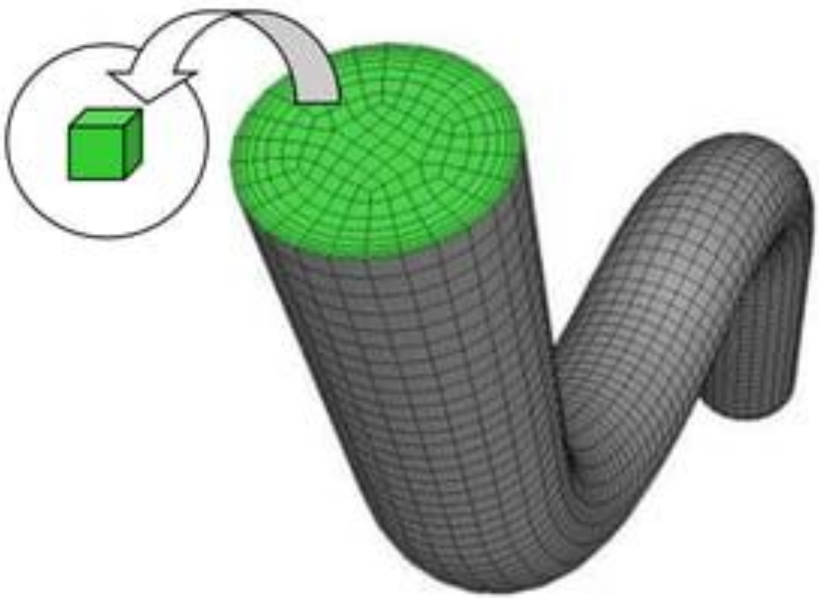
- Analysis begins with a mathematical model of a physical problem.
- Conservation of matter, momentum, and energy must be satisfied throughout the region of interest.(Continuity, Momentum equation and Energy equation).
- Fluid properties are modeled empirically.
- Simplifying assumptions are made in order to make the problem tractable (e.g., steady-state, incompressible, inviscid, two-dimensional etc.)

How It Works?

- Appropriate initial and boundary conditions are provided for the problem.
- CFD applies numerical method called discretization to develop approximations of the governing equations of fluid mechanics in the fluid region of interest.
- The solution is post-processed to extract quantities of interest (e.g. lift, drag, torque, heat transfer, separation, pressure loss, etc.).

Discretization

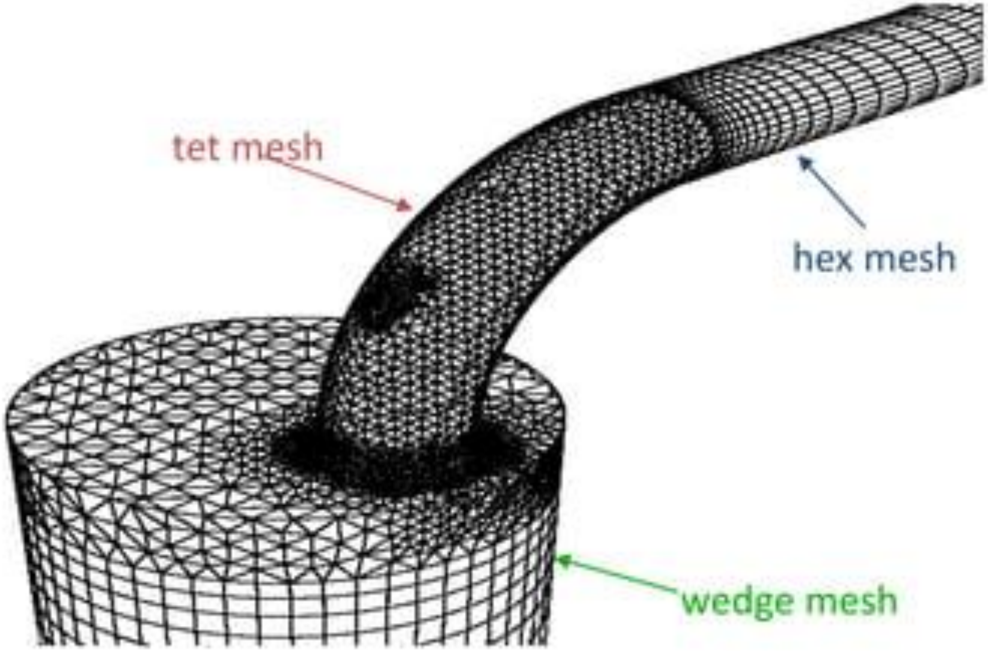
- Domain is discretized into a finite set of control volumes or cells. The discretized domain is called the "grid" or the "mesh."
- General conservation (transport) equations for mass, momentum, energy, etc., are discretized into algebraic equations.
- All equations are solved to render flow field.



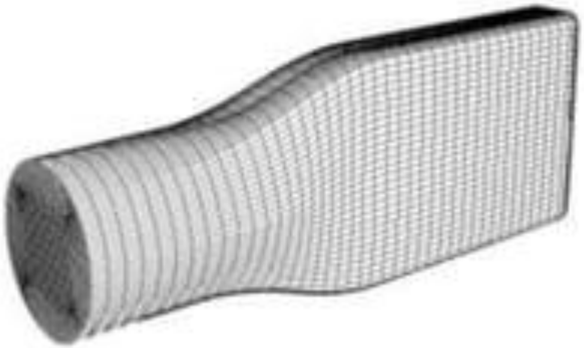
Fluid region of pipe flow discretized into finite set of control volumes (mesh).

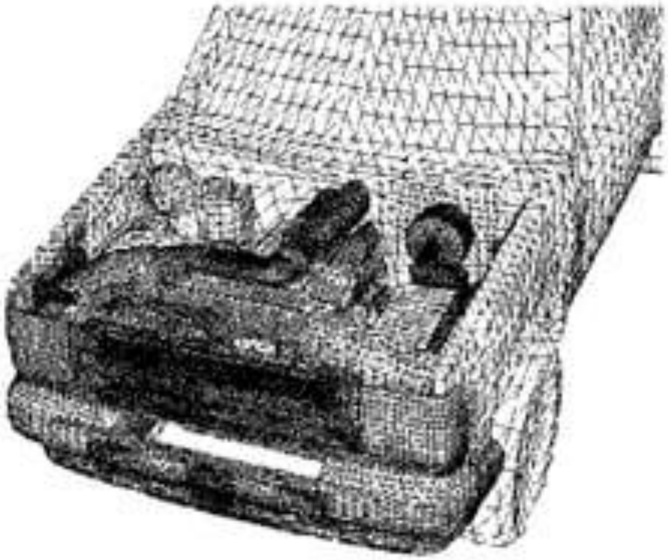
Types Of Meshes

- Tri/tet vs. quad/hex meshes
- Hybrid mesh



Hybrid mesh for an IC engine valve port





Finite Volume Method Used In Discretization

- The finite volume method (FVM) is a common approach used in CFD codes, as it has an advantage in memory usage and solution speed, especially for large problems, high Reynolds number turbulent flows, and source term dominated flows (like combustion).
- In this method the governing partial differential equations are recast in the conservative form and then solved over a discrete control volumes and thus guarantees the conservation of fluxes through a particular control volume.
- Here Q is the vector of conserved variables, F is the vector of fluxes V is the volume of the control volume element, and A is the surface area of the control volume element. The finite volume equation yields governing equations in the form:

$$\frac{\partial}{\partial t}\iiint Q\,dV + \iint F\,d\mathbf{A} = 0,$$

Finite element method

- The finite element method (FEM) is used in structural analysis of solids, but is also applicable to fluids.
- It is much more stable than the finite volume approach. However, it can require more memory and has slower solution than the FVM.

Finite difference method

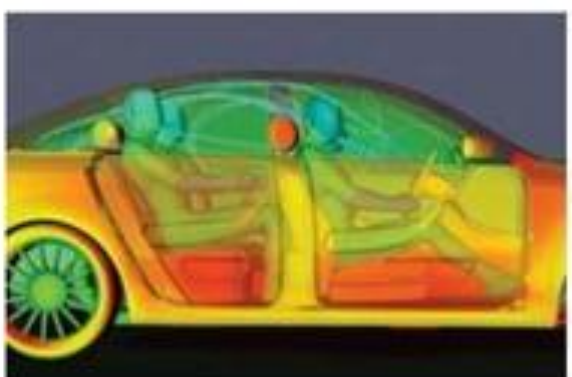
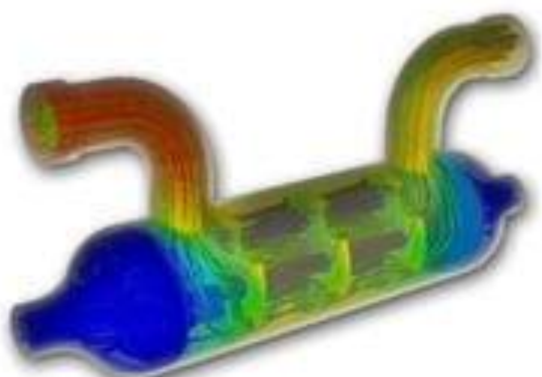
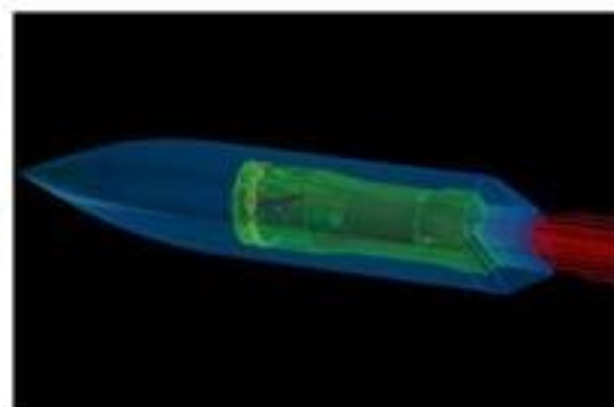
- The finite difference method (FDM) has historical importance and is simple to program.
- It is currently only used in few specialized codes, which handle complex geometry with high accuracy and efficiency by using embedded boundaries or overlapping grids (with the solution interpolated across each grid).

Numerical Model Setup

- For a given problem, you will need to:
  - Select appropriate physical models.
  - Define material properties.
    - Fluid.
    - Solid.
    - Mixture.
  - Prescribe operating conditions.
  - Prescribe boundary conditions at all boundary zones.
  - Set up solver controls.
  - Set up convergence monitors.

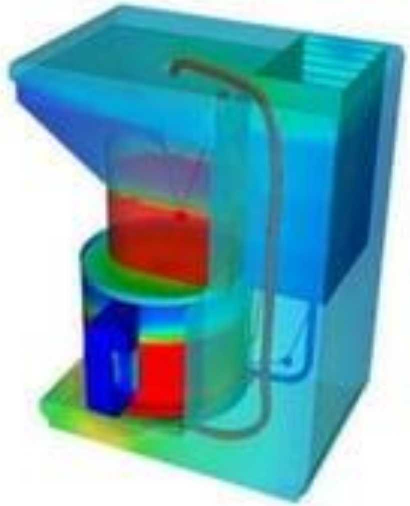
Applications

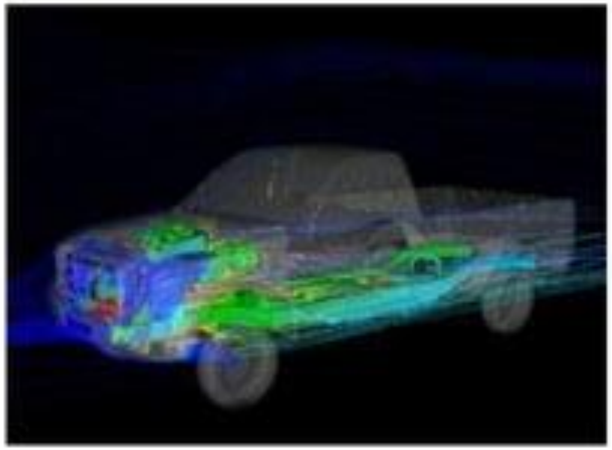
- Car safety thermal imaging using CFD
- Heat exchanger imaging
- Imaging of missile prototypes

Applications

- Electronics thermal analysis
- Designing of super duty vehicles like trucks , tempos etc.(based on thermal stress points)
- Thermal comfort in office environment





Thank you