SNS COLLEGE OF TECHNOLOGY

(An Autonomous Institution)
Approved by AICTE, New Delhi, Affiliated to Anna University, Chennai
Accredited by NAAC-UGC with 'A++' Grade (Cycle III) &
Accredited by NBA (B.E - CSE, EEE, ECE, Mech & B.Tech. IT)
COIMBATORE-641 035, TAMIL NADU

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 III – FINITE ELEMENT TECHNIQUES

Implementation of FEM in CFD

In Computational Fluid Dynamics (CFD), the Finite Element Method (FEM) is implemented by dividing the fluid domain into small, interconnected elements, allowing for the numerical solution of the governing fluid flow equations within each element, ultimately providing an approximation of the overall flow behavior across the entire domain; this is particularly useful for complex geometries where other methods might struggle.

Here's a breakdown of key aspects:

1. Governing Equations:

- CFD aims to solve partial differential equations (PDEs) that describe fluid behavior, such as:
    - Continuity equation (mass conservation)
    - Navier-Stokes equations (momentum conservation)
    - Energy equation (energy conservation)
- These equations can be complex, nonlinear, and involve various terms representing convection, diffusion, and source terms.

2. Variational Formulation in CFD:

- Similar to general FEM, CFD utilizes variational formulations (weak forms) of the governing equations.
- This involves:

- Multiplying the PDEs by test functions.
- Integrating over the computational domain.
- Applying integration by parts to reduce the order of derivatives.

- This process transforms the strong form of the equations into a weak form, which is more amenable to numerical solution.

3. Discretization:

- The computational domain is discretized into finite elements, creating a mesh.
- Within each element, the fluid variables (velocity, pressure, temperature) are approximated using shape functions.
- The weak form is then applied to each element, resulting in a system of algebraic equations.

4. Assembly and Solution:

- The element equations are assembled into a global system of equations, representing the entire computational domain.
- Boundary conditions are imposed to define the fluid behavior at the domain boundaries.
- The resulting system of equations is solved using numerical solvers, which can be:
    - Direct solvers (e.g., Gaussian elimination)
    - Iterative solvers (e.g., conjugate gradient method)

5. Challenges in CFD-FEM:

- **Convection-Dominated Flows:**
    - CFD often deals with flows where convection dominates diffusion, which can lead to numerical instabilities.
    - Special stabilization techniques are required, such as:
        - Streamline upwind Petrov-Galerkin (SUPG) method
        - Galerkin least squares (GLS) method
- **Incompressibility:**
    - Solving incompressible flows requires special treatment of the pressure field to satisfy the continuity equation.
    - Mixed finite element formulations are commonly used.
- **Turbulence:**

  - Turbulent flows are highly complex and require turbulence models to approximate the effects of turbulence.
  - FEM can be used in conjunction with various turbulence models, such as Reynolds-averaged Navier-Stokes (RANS) models or large eddy simulation (LES).

6. Advantages of FEM in CFD:

- **Complex Geometries:**
  - FEM can handle complex geometries and unstructured meshes, which are common in CFD applications.
- **Accuracy:**
  - With appropriate stabilization techniques, FEM can provide accurate solutions for a wide range of fluid flow problems.
- **Flexibility:**

o FEM allows for adaptivity, where the mesh can be refined in regions of high gradients to improve accuracy.

Challenges of FEM in CFD:

- **Computational cost:**

  For large-scale problems, FEM can be computationally expensive due to the large number of degrees of freedom.

- **Mesh quality:**
  Maintaining high-quality mesh is crucial for accurate solutions, especially in complex geometries.

Common applications of FEM in CFD:

- **Fluid-structure interaction (FSI):** Analyzing the interaction between a deformable structure and the surrounding fluid.
- **Biofluid mechanics:** Modeling blood flow in complex vascular geometries.
- **Microfluidics:** Simulating fluid flow in microchannels with intricate designs.