SNS COLLEGE OF TECHNOLOGY

(An Autonomous Institution)
Coimbatore – 641 035, Tamil Nadu

DEPARTMENT OF AEROSPACE ENGINEERING

Subject: 19ASB304 Computational Fluid Dynamics

Title: CFD Mesh Generation Techniques

By

Name : Jawahar Adithya C.M

Year: Third Year

Branch: Aerospace Engineering

CFD Mesh Generation Techniques

- Mesh generation is a crucial step in Computational Fluid Dynamics (CFD) simulations. It involves discretizing the computational domain into smaller elements or cells, allowing for the numerical solution of fluid flow equations.

- That the choice of mesh generation technique depends on factors like the complexity of the geometry, accuracy requirements, computational resources, and solver capabilities. A well-designed mesh plays a crucial role in obtaining reliable and accurate CFD results.

CFD Mesh Generation Techniques

Here are some common mesh generation techniques used in CFD:

- Structured Meshing
- Unstructured Meshing
- Hybrid Meshing
- Delaunay Triangulation
- Advancing Front Method
- Meshing Using Tetrahedralization
- Meshing Using Boundary Layer Techniques
- Grid Generation Software

CFD Mesh Generation Techniques

- In structured meshing, the domain is divided into a regular grid of quadrilateral (2D) or hexahedral (3D) elements. It offers good numerical accuracy and is efficient for simple geometries, but it becomes challenging for complex and irregular geometries.
- Unstructured meshing allows for more flexibility in handling complex geometries. The domain is divided into irregular polygons (2D) or polyhedral elements (3D), such as triangles, quadrilaterals, tetrahedra, or hexahedra. This technique is widely used in CFD due to its versatility.

CFD Mesh Generation Techniques

- Hybrid Meshing: Hybrid meshing combines structured and unstructured meshing techniques. It involves creating structured grids in regions with simple geometries and using unstructured grids in complex regions. This approach balances accuracy and computational efficiency.
- Delaunay Triangulation: Delaunay triangulation is a method used to generate unstructured triangular meshes. It optimizes the placement of nodes (vertices) within the domain to ensure that no node lies inside the circumcircle of any triangle. It is widely used for surface meshing.

CFD Mesh Generation Techniques

- Advancing Front Method: The advancing front method is a technique used for generating unstructured meshes. It starts from a known boundary and progressively fills the domain by advancing a front of mesh cells. This method is suitable for domains with complex boundaries.
- Meshing Using Tetrahedralization: Tetrahedralization is a technique used to generate unstructured tetrahedral meshes. It involves dividing the domain into tetrahedra using algorithms like Delaunay tetrahedralization or advancing front methods. Tetrahedral meshes are commonly used in complex 3D geometries.

CFD Mesh Generation Techniques

- Meshing Using Boundary Layer Techniques: In CFD simulations, it is often necessary to accurately resolve the boundary layer near solid surfaces. Techniques like prism layers or boundary layer meshing are employed to generate high-resolution grids in these regions, ensuring accurate modeling of viscous flows.
- Grid Generation Software: Several commercial and open-source software packages are available for automated mesh generation, such as ANSYS ICEM CFD, Pointwise, Gambit, Gmsh, and OpenFOAM's snappyHexMesh. These tools offer a range of capabilities and options for mesh generation.


Thank You