INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal, Hyderabad - 500 043

Power Point Presentation

on

Computational Aerodynamics

III B Tech II Semester

Prepared by

Dr. G Malaikannan, Professor, Dept of Aeronautical Engg

Mr. G Satya Dileep, Assistant Professor, Dept of Aeronautical Engg

UNIT 1

INTRODUCTION TO CFD

Outline

- Approaches
- Introduction
  - Benefits
- Modeling of fluid
- Non Dimensional Representation
- Classification of PDE
- Finite Difference
- Consistency, Stability, Error analysis of schemes

Approaches

Approach	Advantages	Disadvantages
Experimental	Capable of being most realistic	Equipment required, Scaling problems, Measurement Difficulty, Operating Cost
Theoretical	General Information in Formula form	Restricted to simple geometry and physics, Usually for Linear problems
Computational	Complicated Physics, Details of Flow	Truncation Errors, Computer Costs, Boundary conditions

Introduction

- What is CFD?
  - CFD: A methodology for obtaining a discrete solution of real world fluid flow problems.
  - Discrete Solution: Solution is obtained at a finite_collection of space points and at discrete time levels
- For a reasonably accurate solution, the number of space points that need to involved is of the order of few millions. Solution is achievable only through modern high speed computers

The Benefits of CFD

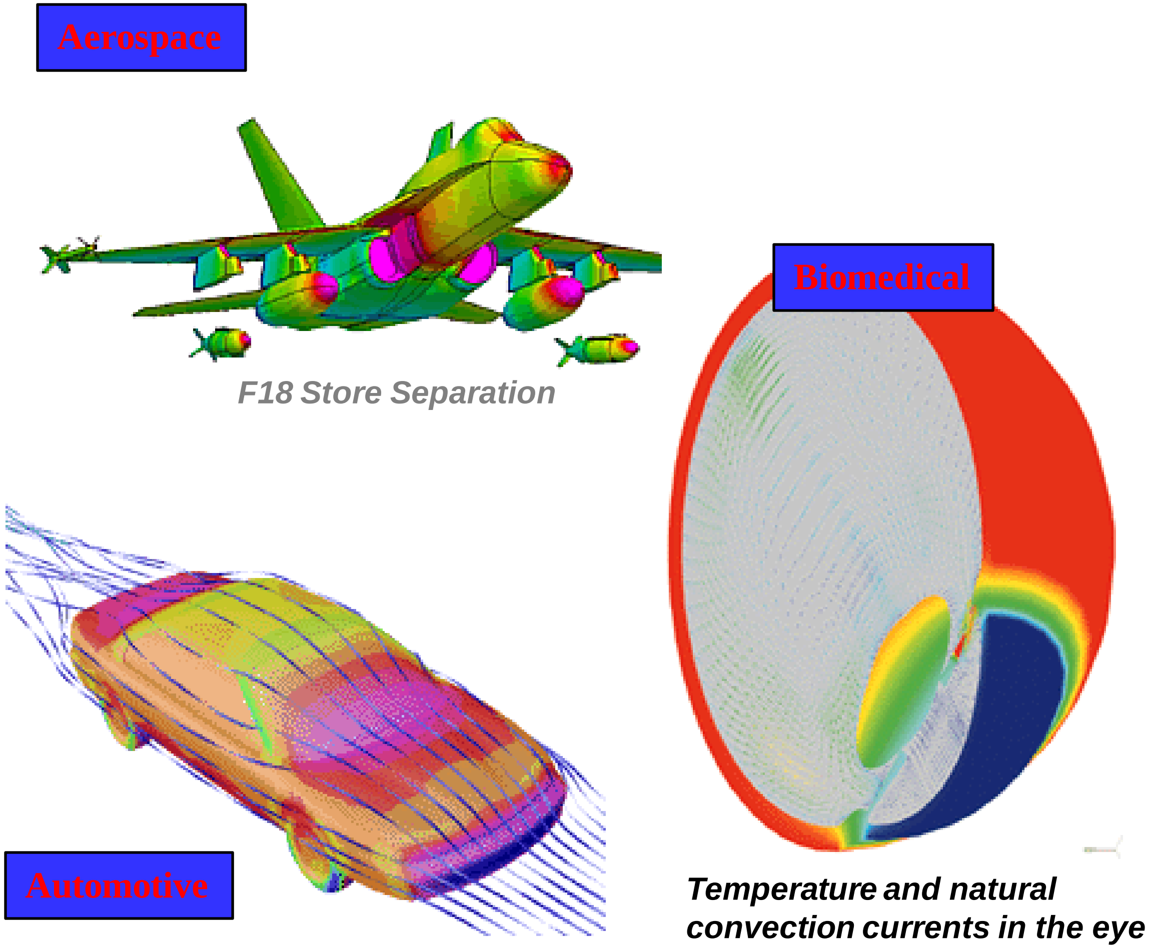
- Insight
  - Difficult to prototype or test through experimentation
  - Better Details
- Foresight
  - Better prediction: In a short time
- Efficiency
  - Design better and faster, economical, meet environmental regulations and ensure industry compliance.
  - CFD analysis leads to shorter design cycles and your products get to market faster.
  - In addition, equipment improvements are built and installed with minimal downtime.
  - CFD is a tool for compressing the design and development cycle allowing for rapid prototyping.

Why use CFD?

- Analysis and Design
  - Simulation-based design instead of “build & test”
    - More cost effective and more rapid than EFD
    - CFD provides high-fidelity database for diagnosing flow field
  - 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)
- Knowledge and exploration of flow physics

Where is CFD used?

- Where is CFD used?
  - **Aerospace**
  - **Automotive**
  - **Biomedical**
  - Chemical Processing
  - HVAC
  - Hydraulics
  - Marine
  - Oil & Gas
  - Power Generation
  - Sports
  - Pollutant Monitoring



F18 Store Separation

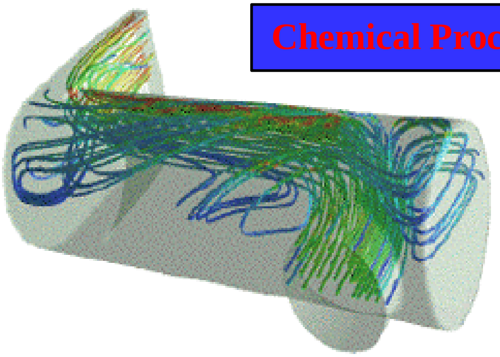
Temperature and natural convection currents in the eye following laser heating.

Fig: 1.1 Various applications of CFD in other streams

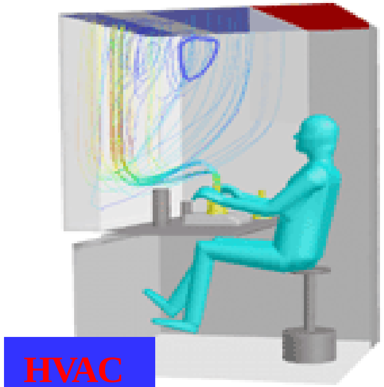
Where is CFD used?

- Where is CFD used?
  - Aerospacee
  - Automotive
  - Biomedical
  - **Chemical Processing**
  - **HVAC**
  - **Hydraulics**
  - Marine
  - Oil & Gas
  - Power Generation
  - Sports

Chemical Processing



Polymerization reactor vessel - prediction of flow separation and residence time effects.



HVAC

Streamlines for workstation ventilation

Hydraulics

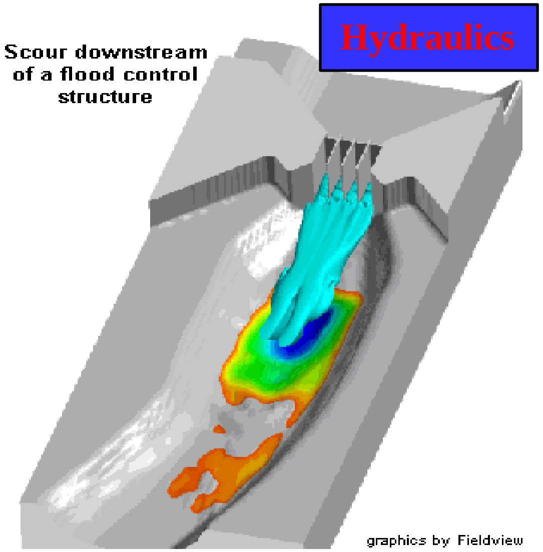
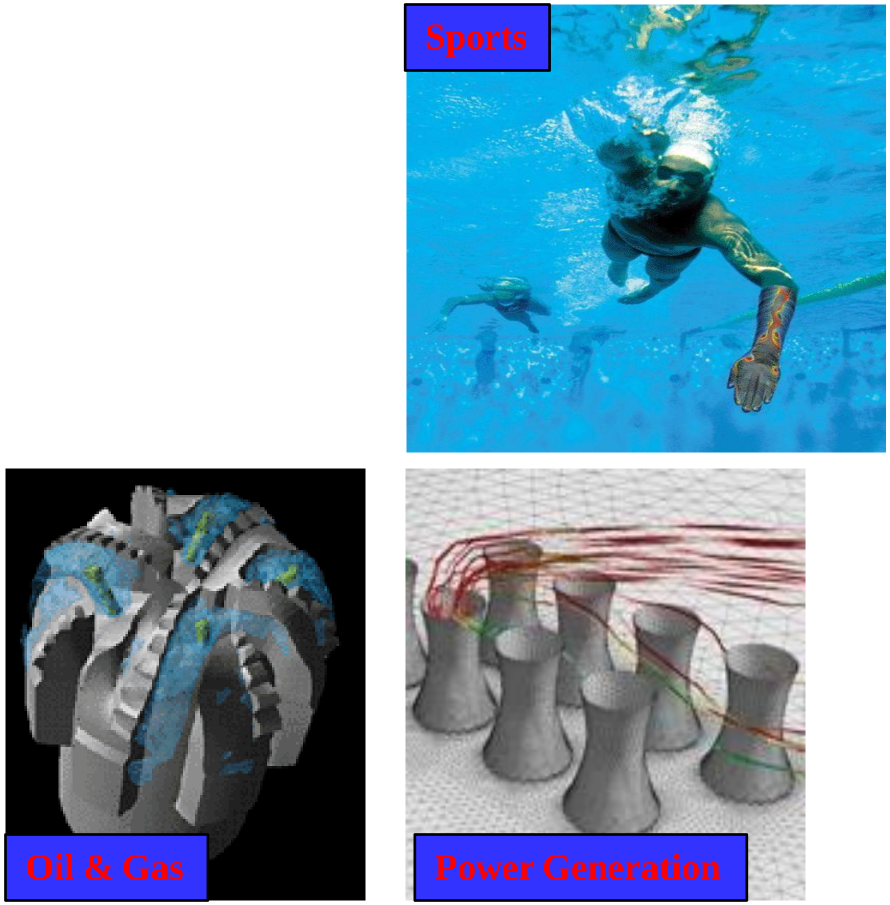


Fig: 1.2 Various applications of CFD in other streams

Where is CFD used?

- Where is CFD used?
  - Aerospace
  - Automotive
  - Biomedical
  - Chemical Processing
  - HVAC
  - Hydraulics
  - **Marine**
  - **Oil & Gas**
  - **Power Generation**
  - **Sports**



Flow of lubricating mud over drill bit

Flow around cooling towers

Fig: 1.3 Various applications of CFD in other streams

Steps in CFD Analysis

Pre-processing

Step 1

Step 2

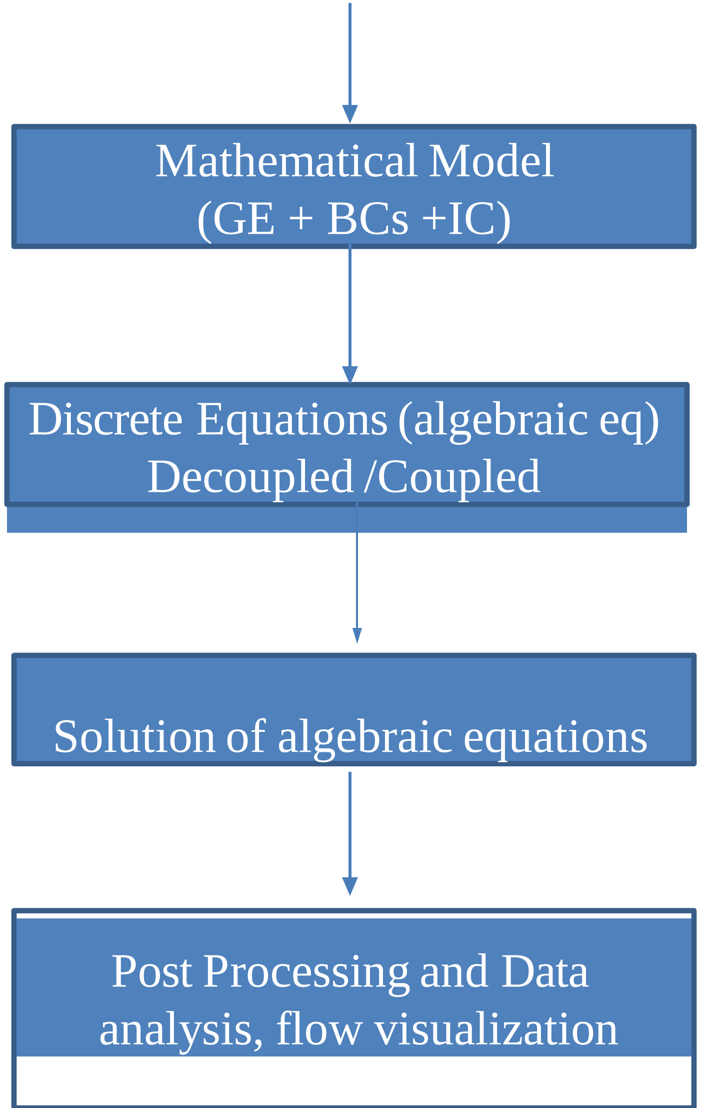
Step 3

Processing/ Computing

Step 4

Post Processing

Step 5



Mathematical Model
(GE + BCs +IC)

Discrete Equations (algebraic eq)
Decoupled /Coupled

Solution of algebraic equations

Post Processing and Data
analysis, flow visualization

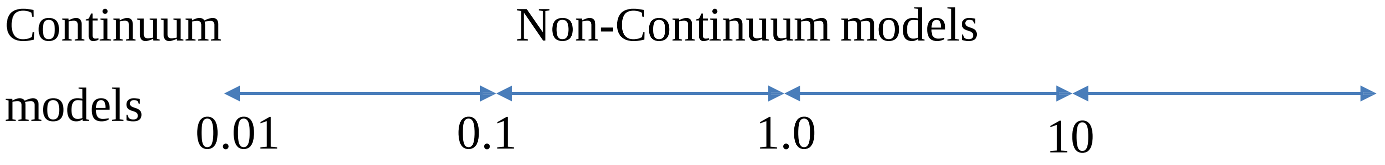
Physical Laws,
Levels of approximation:
Constitutive Behavior,
Simplifying approximation, Semi-empirical models

Discretization of flow domain:
Grid/Mesh Generation

Discretization of GE+BCs:
Numerical Methodology

Modelling of Fluid Flow

- Continuum models
  - – Each Macroscopic property = f(r, t)
- Non Continuum models
  - – Micro, slip flow, molecular
    - Based on the Knudsen number=Ratio of Mean free path/length scale



Continuum models

Non-Continuum models

0.01

0.1

1.0

10

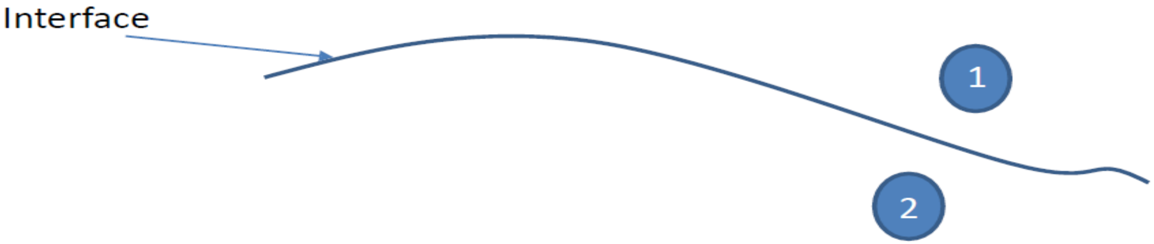
- Overlap region: 0.01 to 0.1
  - – Levels of approximation: temp and velocity jump at soild-fluid interface

Conservative form

- All conservative forms are considered most suitable for CFD
  - The conservative properties can be easily preserved at discrete level.
  - For high speed flows having discontinuous features like shock waves, the fluxes in the conservation equations remain well behaved across these almost discontinuous features, and therefore the flow behavior is better captured at the discrete level.

Boundary Conditions

- Kinematic Boundary Conditions
  - V and T
  - No penetration



  - If the interface is impervious, i.e. material on either side does not penetrate the interface.

$$(V_n)_1 = (V_n)_2 \text{ or} \left(\vec{V}.\hat{n}\right)_1 = \left(\vec{V}.\hat{n}\right)_2 = (V_S.\hat{n})$$

  - Example: Solid-Fluid interface, liquid vapour interface, interface of immiscible fluids.

Boundary Conditions

- Deforming Interface

$$\emptyset(\vec{r},t) = Const. - time\ dependent\ surface$$

$$d\emptyset \cong \frac{\partial\emptyset}{\partial t}\delta t + \nabla\emptyset . \delta\vec{r_s} = 0$$

$$\frac{\partial\emptyset}{\partial t} + \vec{V_s} . \nabla\emptyset = 0$$

$$\frac{\partial\emptyset}{\partial t} + \vec{V_1} . \nabla\emptyset = \frac{\partial\emptyset}{\partial t} + \vec{V_2} . \nabla\emptyset$$

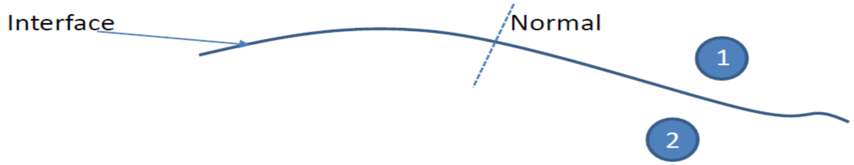
- No Slip

$$\left(\vec{V}.\hat{t}\right)_1 = \left(\vec{V}.\hat{t}\right)_2$$

Implication: Fluid motion can be induced by setting an interface into motion.

Boundary Conditions

- Thermal Boundary Conditions



- Continuity of Temp and heat flux across the interface.

$$(T_1) = (T_2) = (T_s)$$

Or

$$k_1(\nabla T)_1.\hat{n} = k_1(\nabla T)_2.\hat{n} = (V_S.\hat{n})$$

- Dynamic Boundary Condition

$$(\sigma_{nn})_1 - (\sigma_{nn})_2 = Y\left(\frac{1}{R_1} + \frac{1}{R_2}\right)$$

Non-Dimensional Representation

- Reduction in the number of parameters for a given problem large no of dimensional solution obtained in less number of trials.

- Permits a relative comparison of magnitudes of various terms in the GE Simplified but reasonable accurate models

- Permits a normalization of the order of magnitude of the various terms helps in achieving and flow sustaining accuracy in discrete solutions of variables that are having different orders of magnitude

Process of Non-Dimensional Represents

- Chose scales for time, space cords, flow variables. Only constants are left untouched.

Variables	Scale
t	t_s
Space coord	L_s
density	$\square_s$
velocity	U_s
Temp	T_s, $T\text{-}T_o/ \square T_s$
E, e	C_vT_s
P	$P\text{-}P_o /\square_s U^2_s$, P_s

Continuity

$$\frac{\rho_S}{t_S}\frac{\partial \rho^*}{\partial t^*} + \frac{\rho_S U_S}{L_S}\left(\vec{\nabla}.\rho^*\vec{V}^*\right) = 0$$

$$\left(\frac{L_S}{t_S U_S}\right)\frac{\partial \rho^*}{\partial t^*} + \left(\vec{\nabla}.\rho^*\vec{V}^*\right) = 0$$

$$\frac{L_S}{U_S} : Residence\ time\ Scale$$

- In the absence of any imposed time scale , $t_S = \frac{L_S}{U_S}$

$$\frac{\partial \rho^*}{\partial t^*} + \left(\vec{\nabla}.\rho^*\vec{V}^*\right) = 0$$

Momentum

$$\left(\frac{L_S}{t_S U_S}\right)\frac{\partial \rho^* \vec{V}^*}{\partial t^*} + \vec{\nabla}.\left[\rho \vec{V}^* \vec{V}^* \left(\frac{\rho_{s} U_S^2}{t_s}\right) - \left(\frac{\mu_S U_S}{L_S^2}\right)\mu^*\left(\nabla \vec{V}^* + \nabla \vec{V}^{*T}\right)\right]$$

$$+\frac{2}{3}\left(\frac{\mu_S U_S}{L_S^2}\right)\nabla\left(\mu^*\nabla\vec{V}^*\right) = -\frac{\rho_S U_S^2}{L}\nabla p^* + \rho_S g \rho^* \hat{e}_g$$

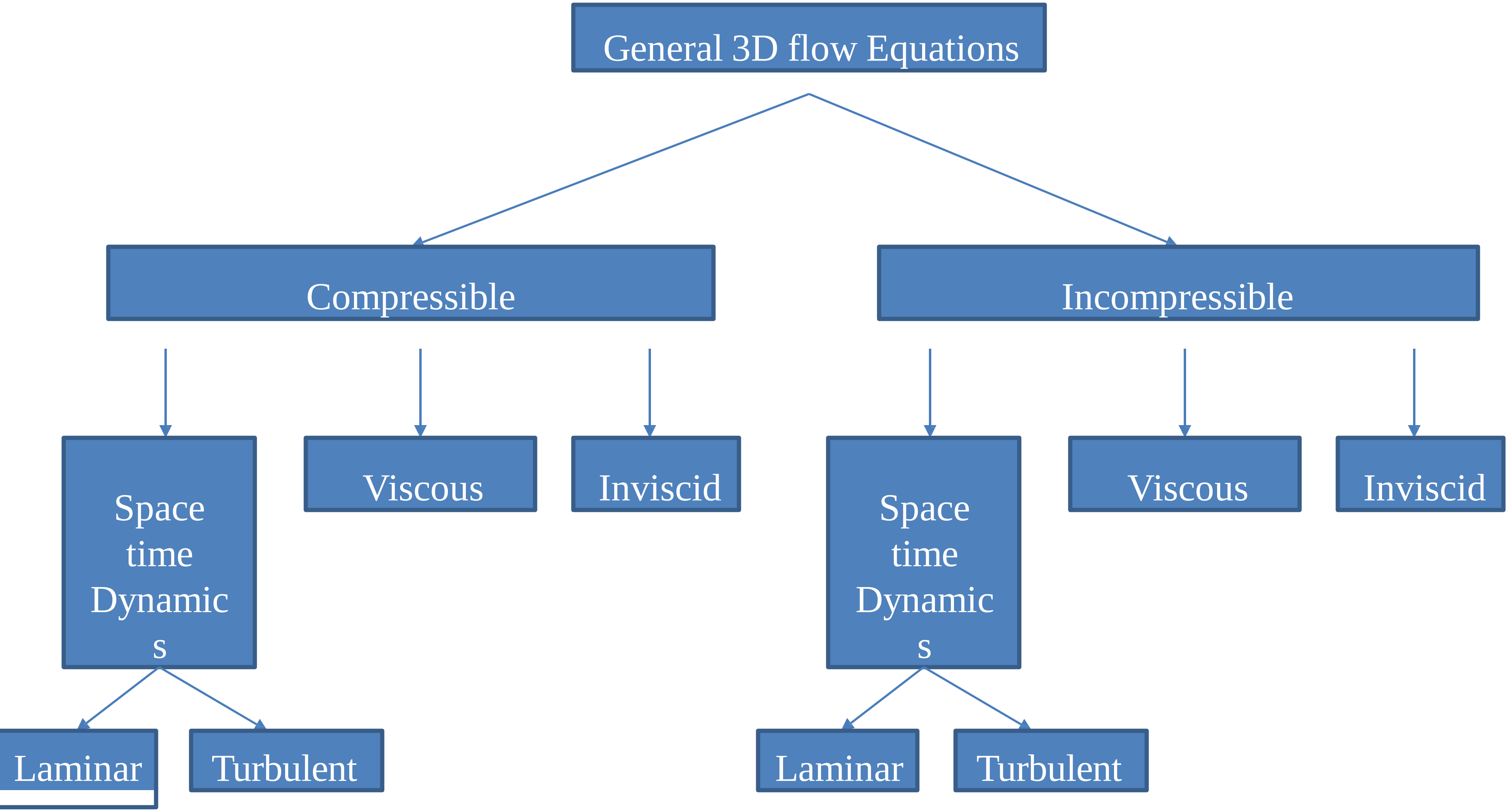
- In the absence of any imposed time scale , $t_S = \frac{L_S}{U_S}$

$$\frac{\partial \rho^* \vec{V}^*}{\partial t^*} + \vec{\nabla}.\left[\rho \vec{V}^* \vec{V}^* - \left(\frac{\mu_S}{\rho_S U_S L_S}\right)\mu^*\left(\nabla \vec{V}^* + \nabla \vec{V}^{*T}\right)\right]$$

$$+\frac{2}{3}\left(\frac{\mu_S}{\rho_S U_S L_S}\right)\nabla\left(\mu^*\nabla\vec{V}^*\right) = -\nabla p^* + \frac{g L_S}{U_S^2}\rho^* \hat{e}_g$$

- $Re = \frac{\rho_S U_S L_S}{\;} = \frac{Inertia\ force}{\;}$

Simplified Models



General 3D flow Equations

Compressible

Incompressible

Space time Dynamics

Viscous

Inviscid

Space time Dynamics

Viscous

Inviscid

Laminar

Turbulent

Laminar

Turbulent

Finite Difference Method

- Equivalent Differential Eqn
- Local Truncation Error
- Consistency
- Stability
- Dispersion

UNIT- II
MATHEMATICAL BEHAVIOR OF PARTIAL DIFFERENTIAL EQUATIONS AND THEIR IMPACT ON COMPUTATIONAL AERODYNAMICS

Classifications

- Classification of partial differential equations
- • A general partial differential equation in coordinates x and y:
- • Characterization depends on the roots of the higher order (here
- Second order) terms:
- **b^2-4ac> 0 then the equation is called hyperbolic.**
- **b^2-4ac = 0 then the equation is called parabolic.**
- **b^2-4ac< 0 then the equation is called elliptic.**
- Note: if a, b, and c themselves depend on x and y, the equations may be of different type, depending on the location in x-y space.

ELLIPTIC PROBLEMS

- Elliptic equations are characteristic of equilibrium problems, this includes many
- (but not all) steady state flows.
- • Examples are potential flow, the steady state temperature distribution in a rod of
- Solid material and equilibrium stress distributions in solid objects under applied
- Loads.
- • For potential flows the velocity is expressed in terms of a velocity potential:
- u=∇φ. Because the flow is incompressible, ∇.u=0, which results in ∇2φ=0. This is
- also known as Laplace's equation
- The solution depends solely on the boundary conditions. This is also known as a
- boundary value problem.
- • A disturbance in the interior of the solution affects the solution everywhere else.
- The disturbance signals travel in all directions.
- • As a result, solutions are always smooth, even when boundary conditions are
- Discontinuous. This makes numerical solution easier!

PARABOLIC PROBLEMS

- Parabolic equations describe marching problems. This includes time dependent problems which involve significant amounts of dissipation. Examples are unsteady viscous flows and unsteady heat conduction. Steady viscous boundary layer flow is also parabolic (march along streamline, not in time).
- An example is the transient temperature distribution in a cooling down rod: The temperature depends on both the initial and boundary conditions. This is also called an initial-boundary-value problem.
- Disturbances can only affect solutions at a later time.
- Dissipation ensures that the solution is always smooth.

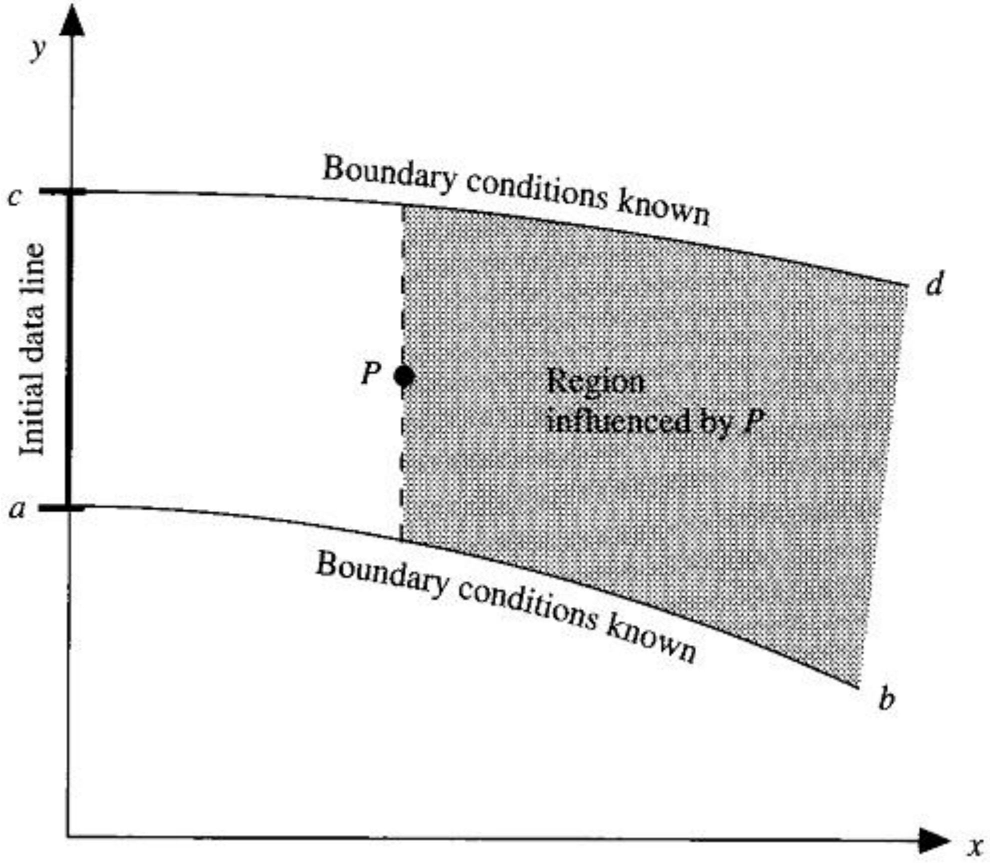


Fig 2.1 Region of influence for parabolic problems

HYPERBOLIC PROBLEMS

- Hyperbolic equations are typical of marching problems with negligible
- dissipation.
- An example is the wave equation:
- This describes the transverse displacement of a string during small amplitude vibrations. If y is the displacement, x the coordinate along the string, and a the initial amplitude, the solution is: Note that the amplitude is independent of time, i.e. there is no dissipation. Hyperbolic problems can have discontinuous solutions.
- Disturbances may affect only a limited region in space. This is called the zone of influence. Disturbances propagate at the wave speed c.
- Local solutions may only depend on initial conditions in the domain of dependence.

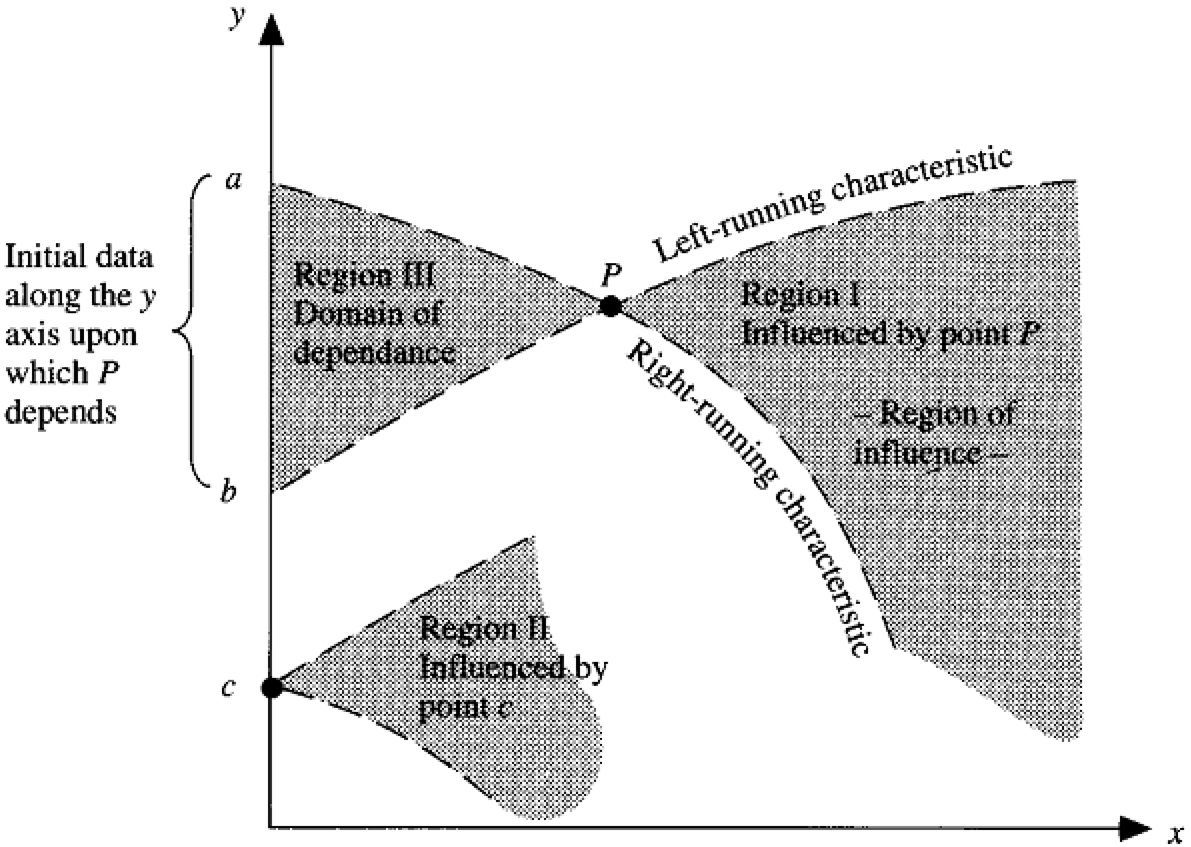


Fig 2.2 Region of influence for hyperbolic problems

UNIT- III
BASIC ASPECTS OF DISCRETIZATION

OUTLINE OF DISCRETIZATION

- Approximations to partial derivatives
- Finite difference representation of Partial Differential Equations
- Discretization

– Consistency

– Stability

– Convergence

- Explicit and implicit approaches
- The finite volume technique
- Boundary conditions
- Stability analysis

THE WAYS TO OBTAIN FINITE DIFFERENCE REPRESENTATIONS OF DERIVATIVES

- Forward difference
- Backward difference
- Central difference

ON THE SELECTION OF A FINITE DIFFERENCE APPROXIMATION

- Depends on the physics of the problem being studied
- Any scheme that fails to represents the physics correctly willfail when you attempt to obtain a solution

- **Steps of Numerical Solution**
- Discretization
  - Consistency
  - Stability
  - Convergence

- **Discretization**
- This is the process of replacing derivatives by finite difference approximations.
- This introduces an error due to the truncation error arising from the finite difference approximation and any errors due to treatment of BC’s.
- The size of the truncation error will depend locally on the solution. In most cases we expect the discretization error to be larger than round-off error.

- **Consistency**

  A finite-difference representation of a PDE is consistent if the difference between the PDE and its difference representation vanishes as the mesh is refined

- **Stability**

  A stable numerical scheme is one for which errors from any source (round-off, truncation) are not permitted to grow in the sequence of numerical procedures as the calculation proceeds from one marching step, or iteration, to the next, thus: errors grow→ unstable errors decay → stable

  • Stability is normally thought of of as being associated with marching problems

  • Stability requirements often dictate allowable step sizes

  • In many cases a stability analysis can be made to define the stability requirements.

- **Convergence**

  The solution of the FDE’s should approach the solution of the PDE as the mesh is refined.

  Lax Equivalence Theorem (linear, initial value

  • For a properly posed problem, with a consistent finite difference representation, stability is the necessary and sufficient condition for convergence.

  In practice, numerical experiments must be conducted to determine if the solution appears to be converged with respect to mesh size.

- **Two Different Approaches**
- • There are many difference techniques used in CFD, you will find that any technique falls into One or the other of following two different general approaches:

  – Explicit approach

  – Implicit approach

- **Advantages of explicit**

  • Relative simple to set up and program

  • This scheme is easily vectorized and a natural formassively parallel computation

- **Disadvantage of explicit**

  • Stability requirements require very small steps sizes

- **Advantage of implicit**

  – Stability requirements allow a large step size

- **Disadvantages of implicit**

  – More complicated to set up and program

  – This scheme is harder to vectorize or parallelize

  – Since the solution of a system of equations is required at each step, the computer time per step is much larger than in the explicit approach.

Solution schemes

- Because of the large number of mesh points, it is generally not practical to solve the system of equations
- Instead, an iterative procedure is usually employed.

  - Initial guess for the solution is made and then each mesh point in the flow field is updated repeatedly until the values satisfy the governing equation.
- This iterative procedure can be thought of as having a time-like quality

GRIDS

Different Types Of Grids

- Structured Grid
- Unstructured Grid

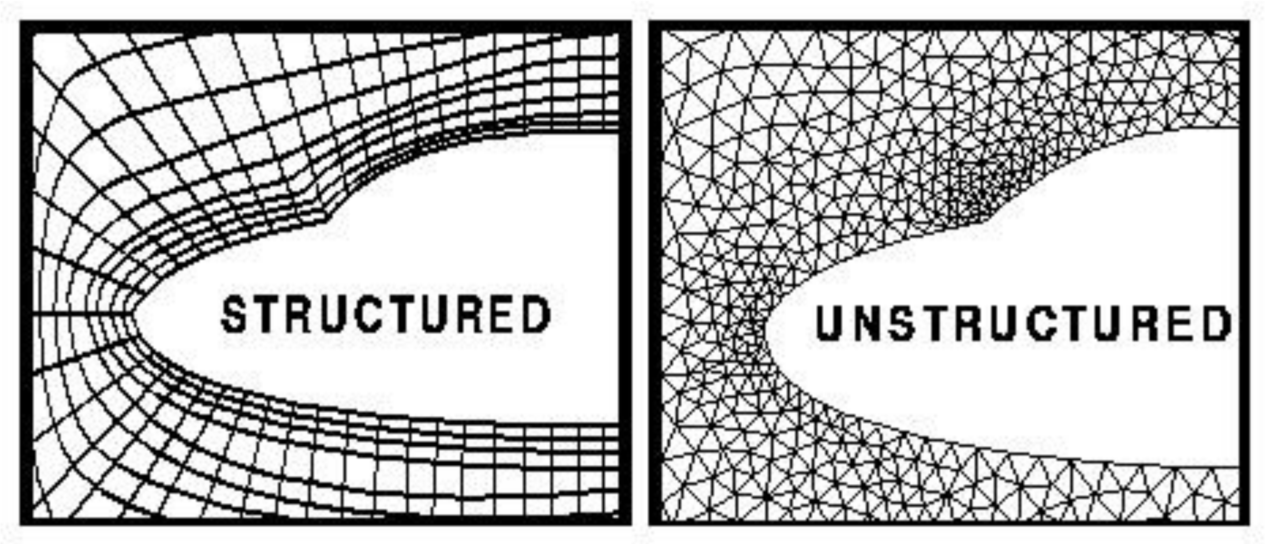


Fig 3.1 Structured and Unstructured grids

Structured Grid

- Cartesian Grids

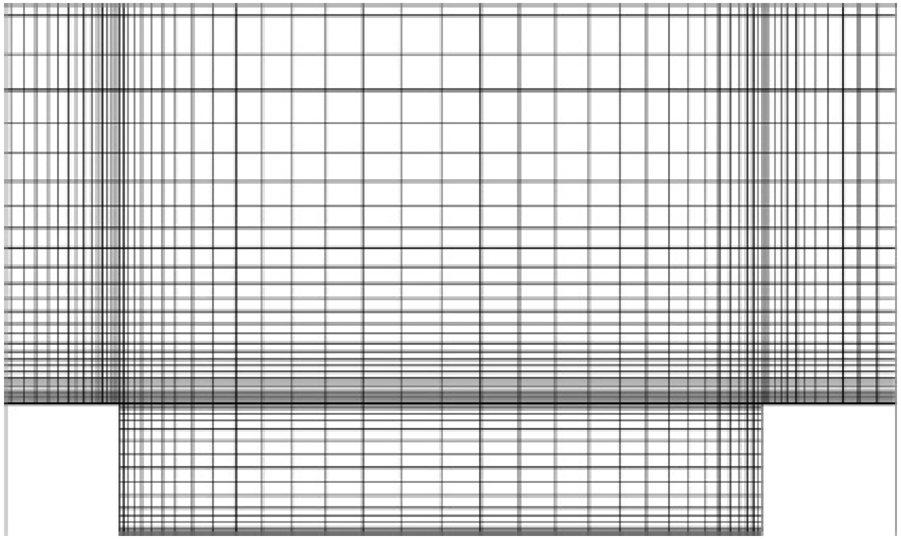


Fig 3.2 Cartesian grids

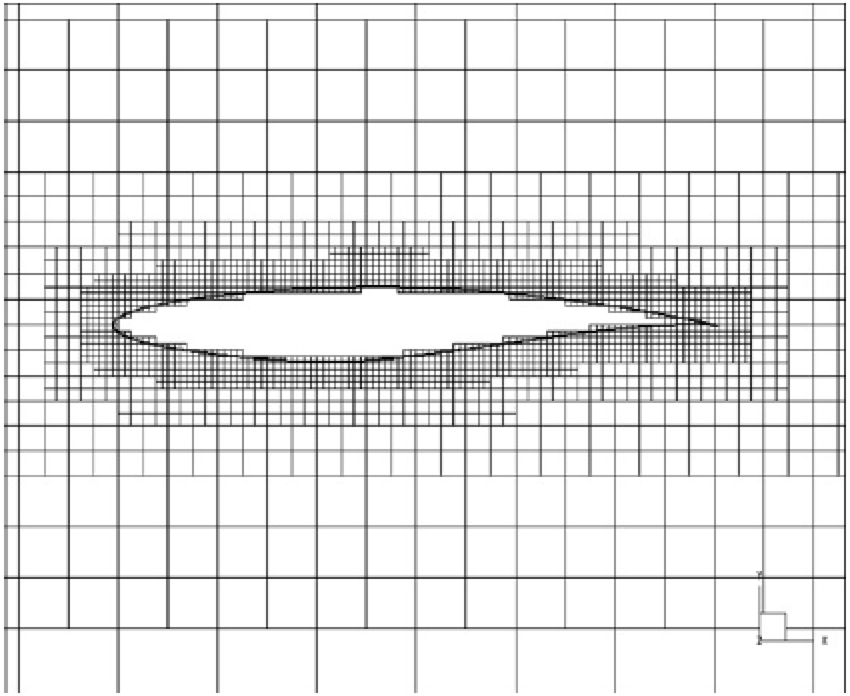
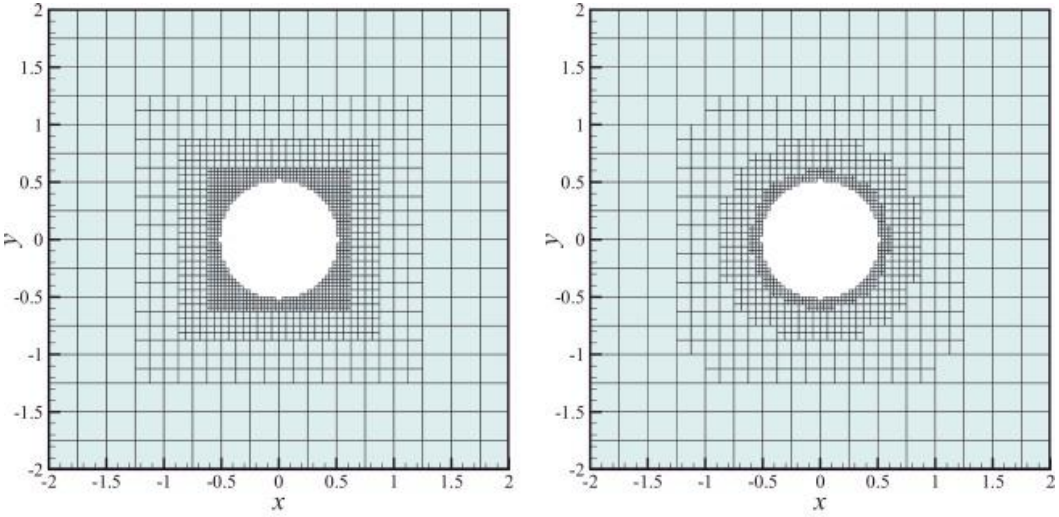


Fig 3.3 Types of cartesian grids



(a) Block-based refinement (1480 cells)

(b) Cell-based refinement (1096 cells)

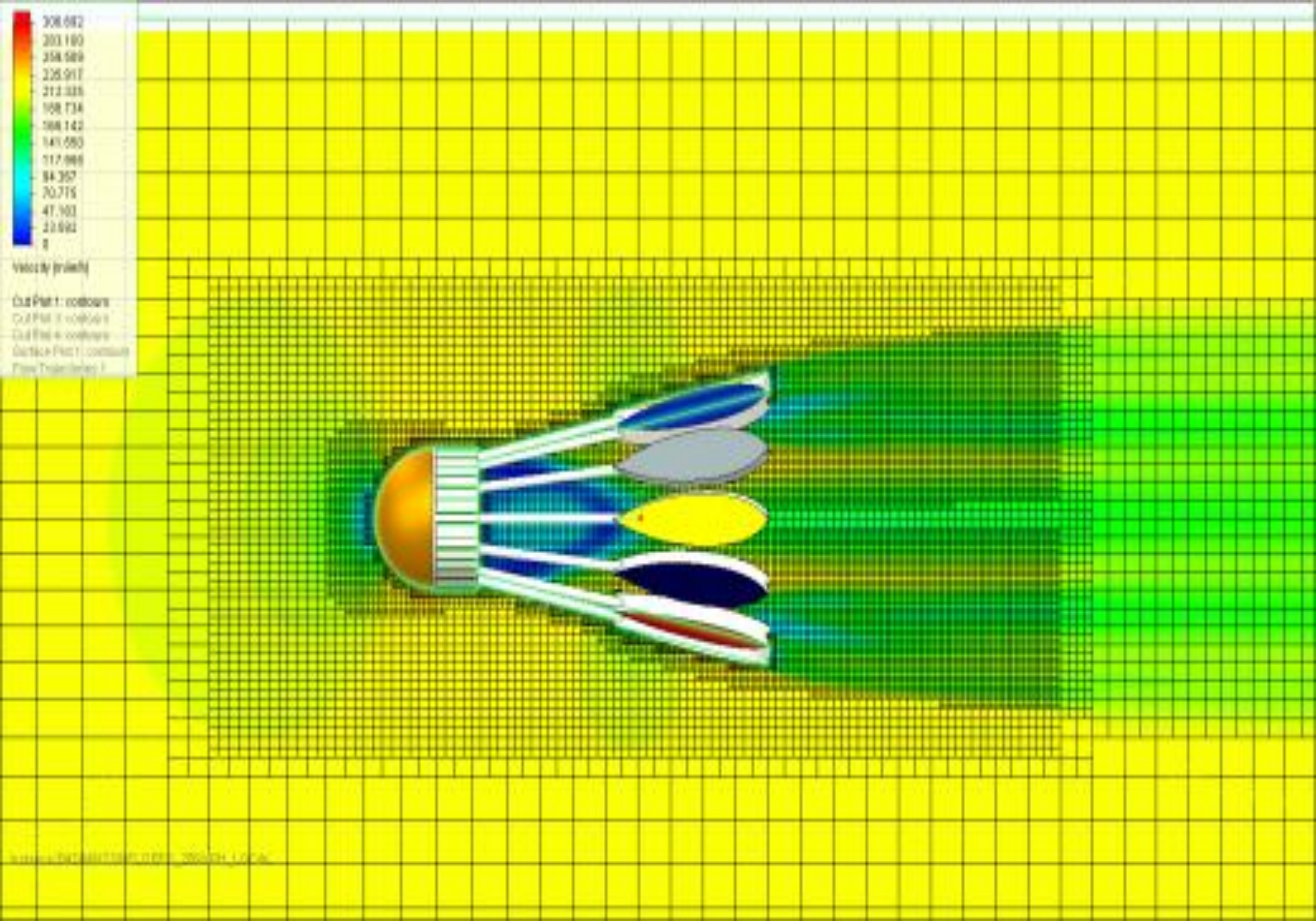


Fig 3.4 Cartesian grid over an Shuttle Cock

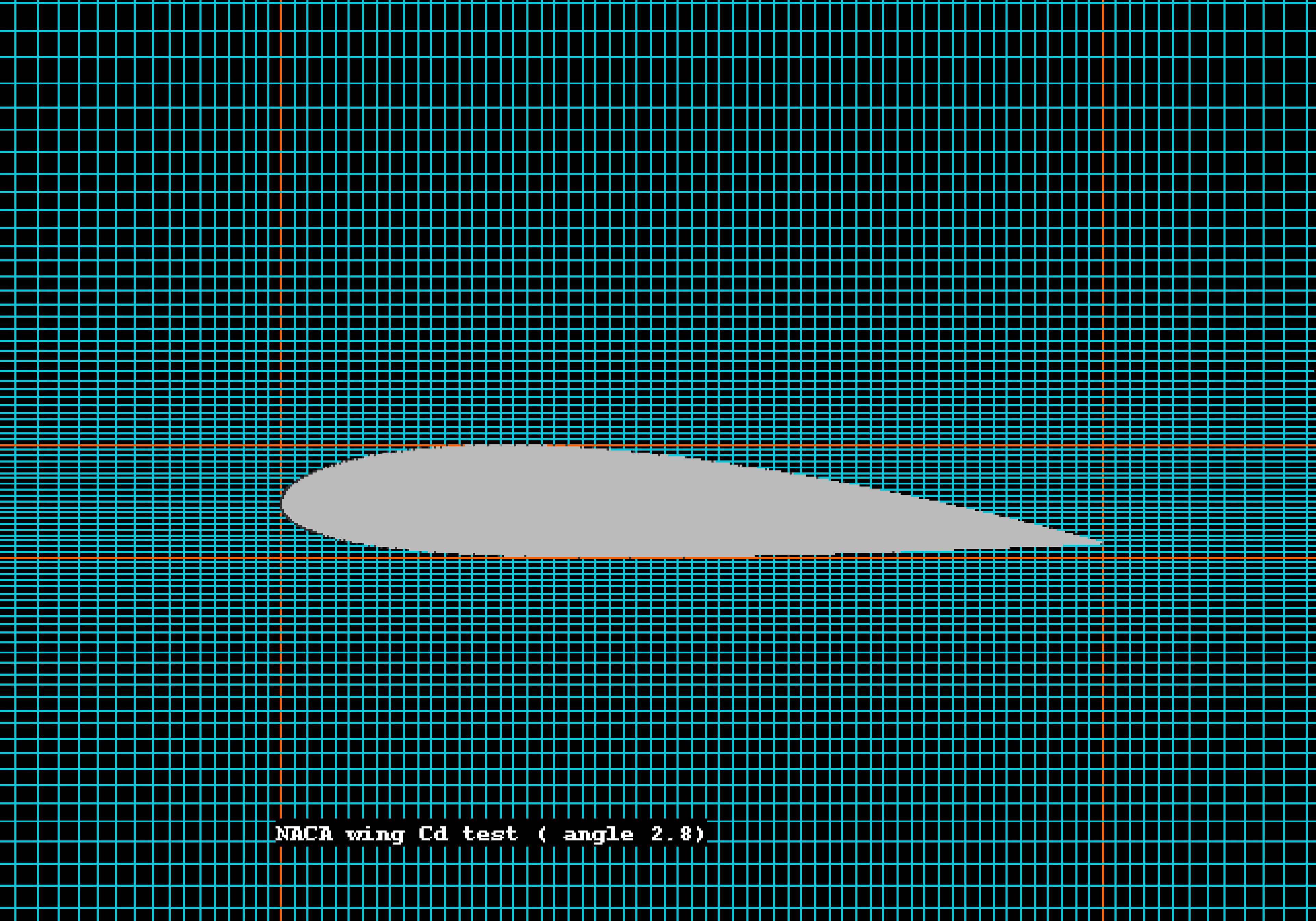


Fig 3.5 Cartesian grid over an airfoil

H Type

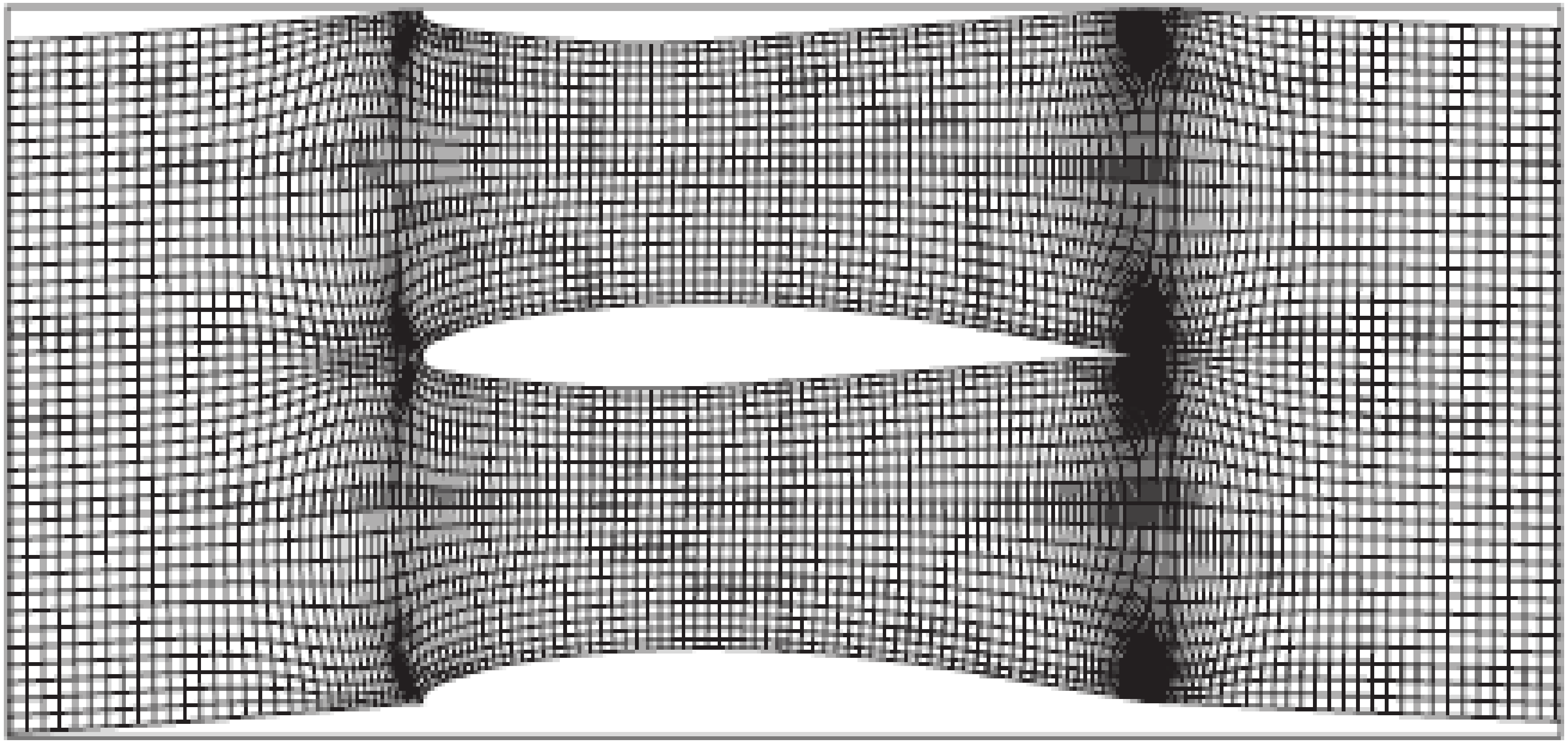


Fig: 3.6 H-Type mesh

C Type

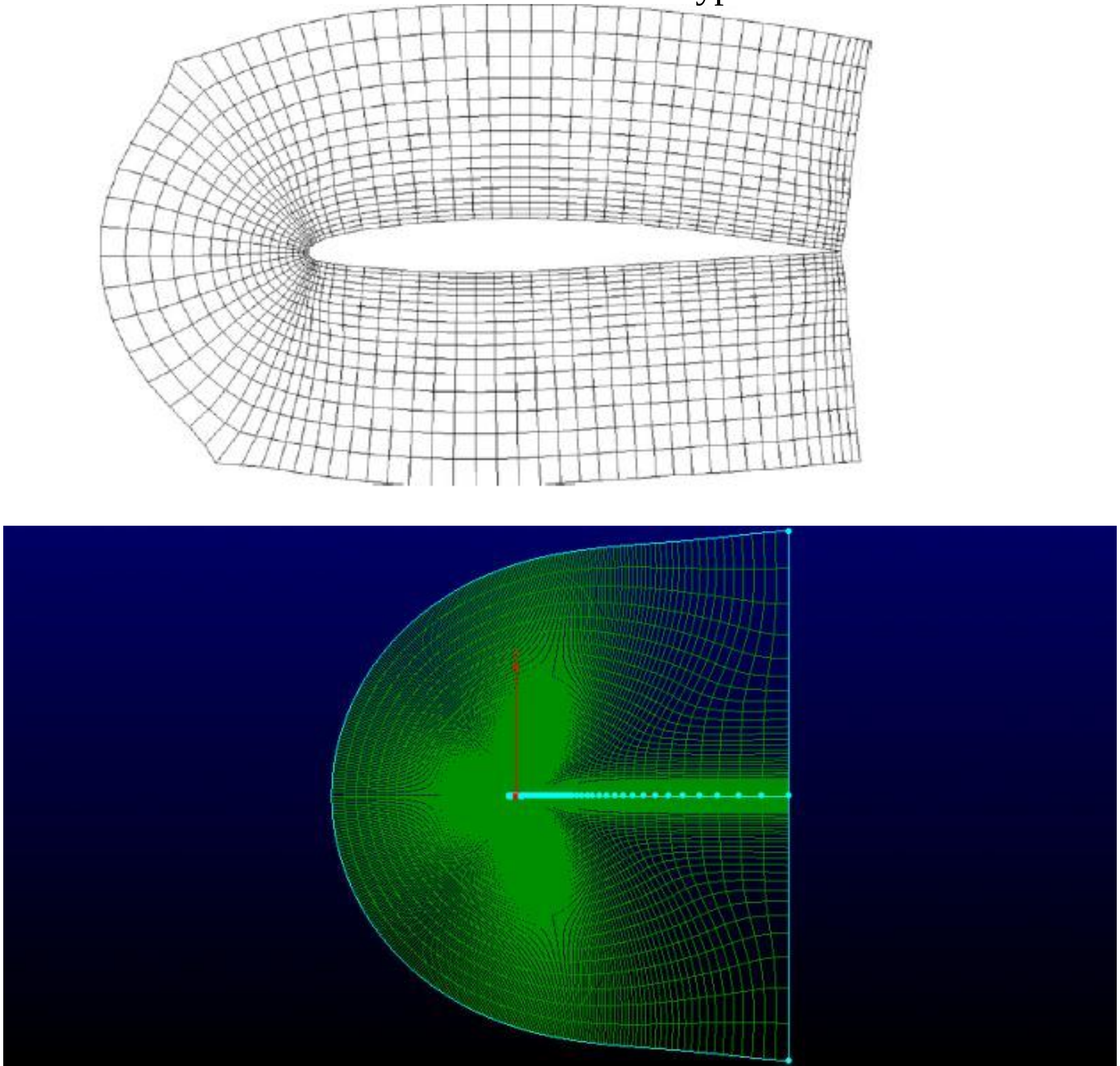


Fig: 3.7 C-Type mesh

O Type

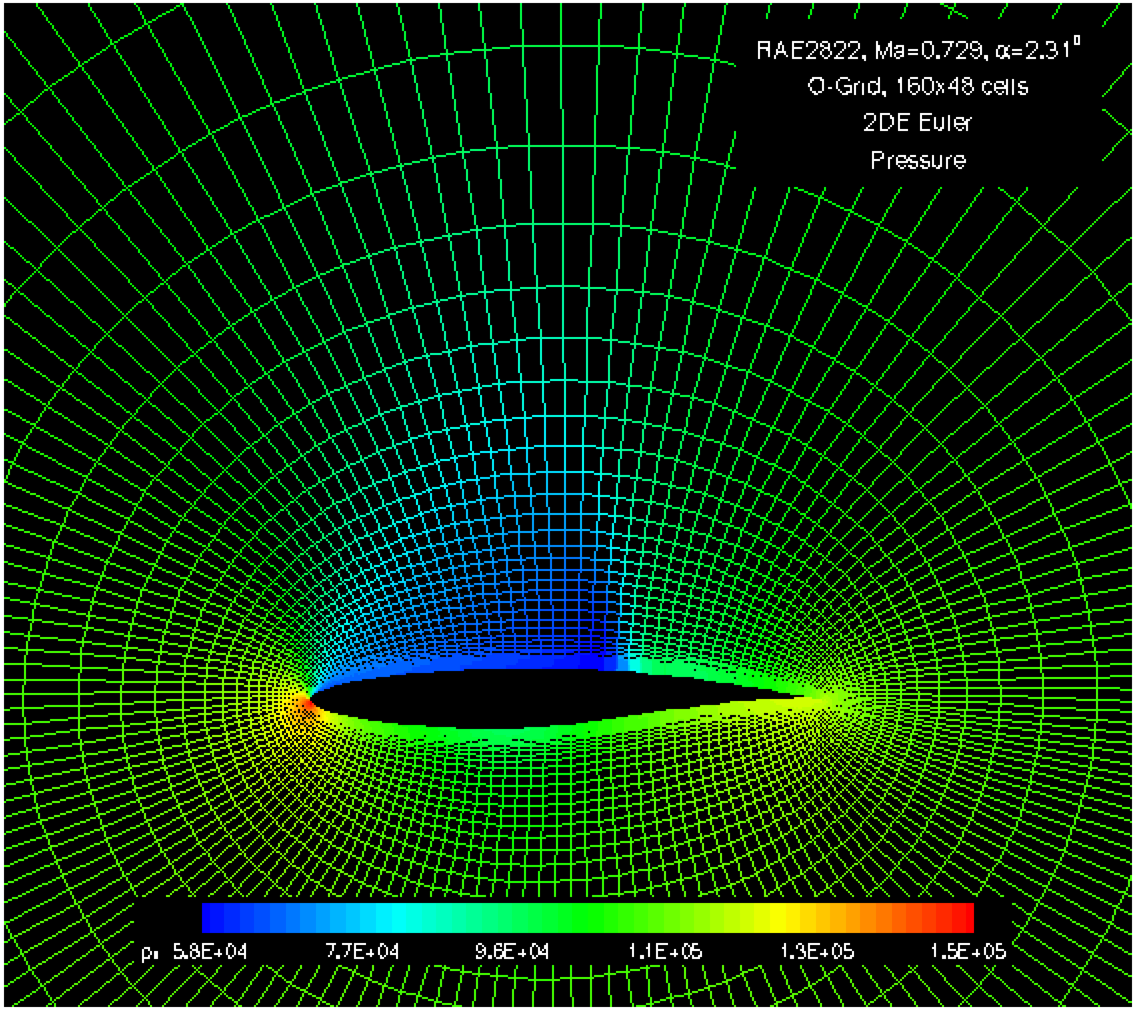


Fig: 3.8 O-Type mesh

I Type

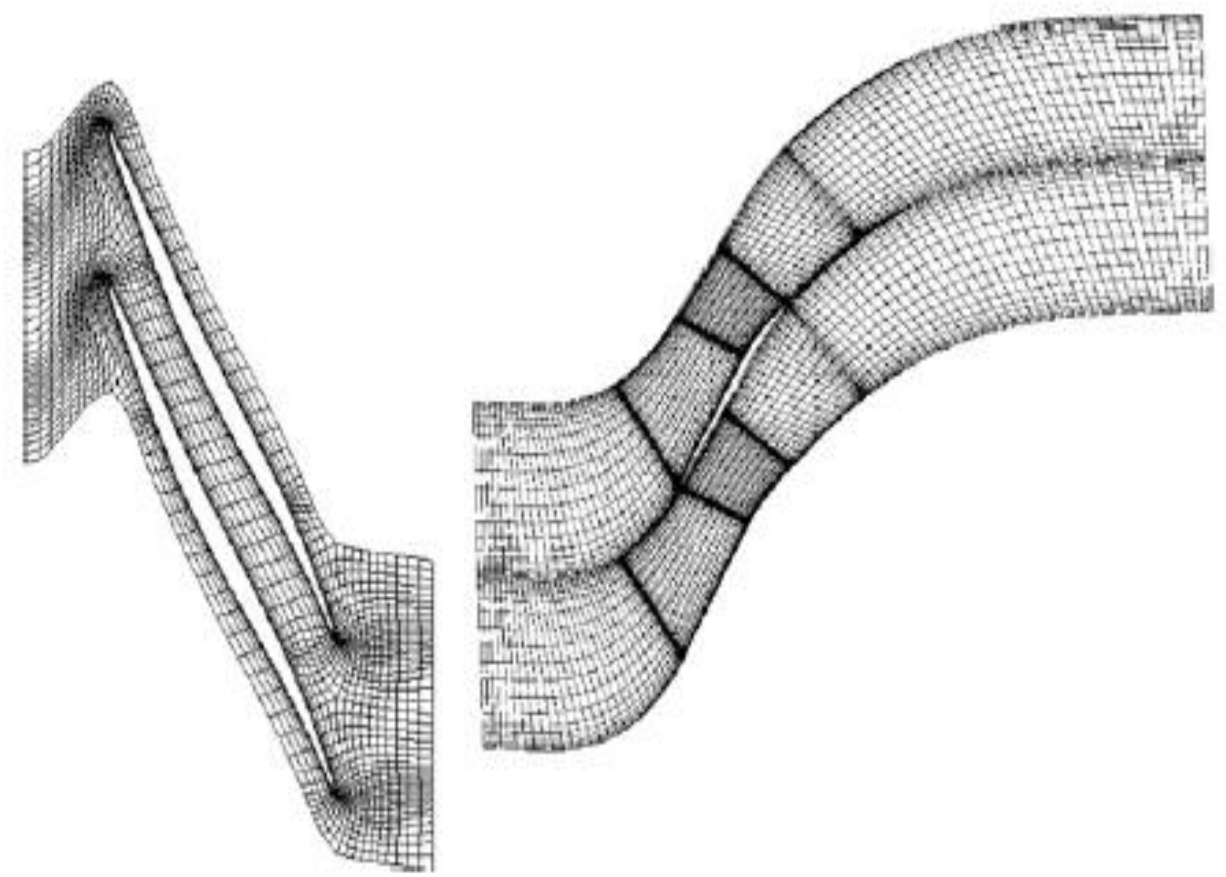


Fig: 3.9 I-Type mesh

C-H Grid

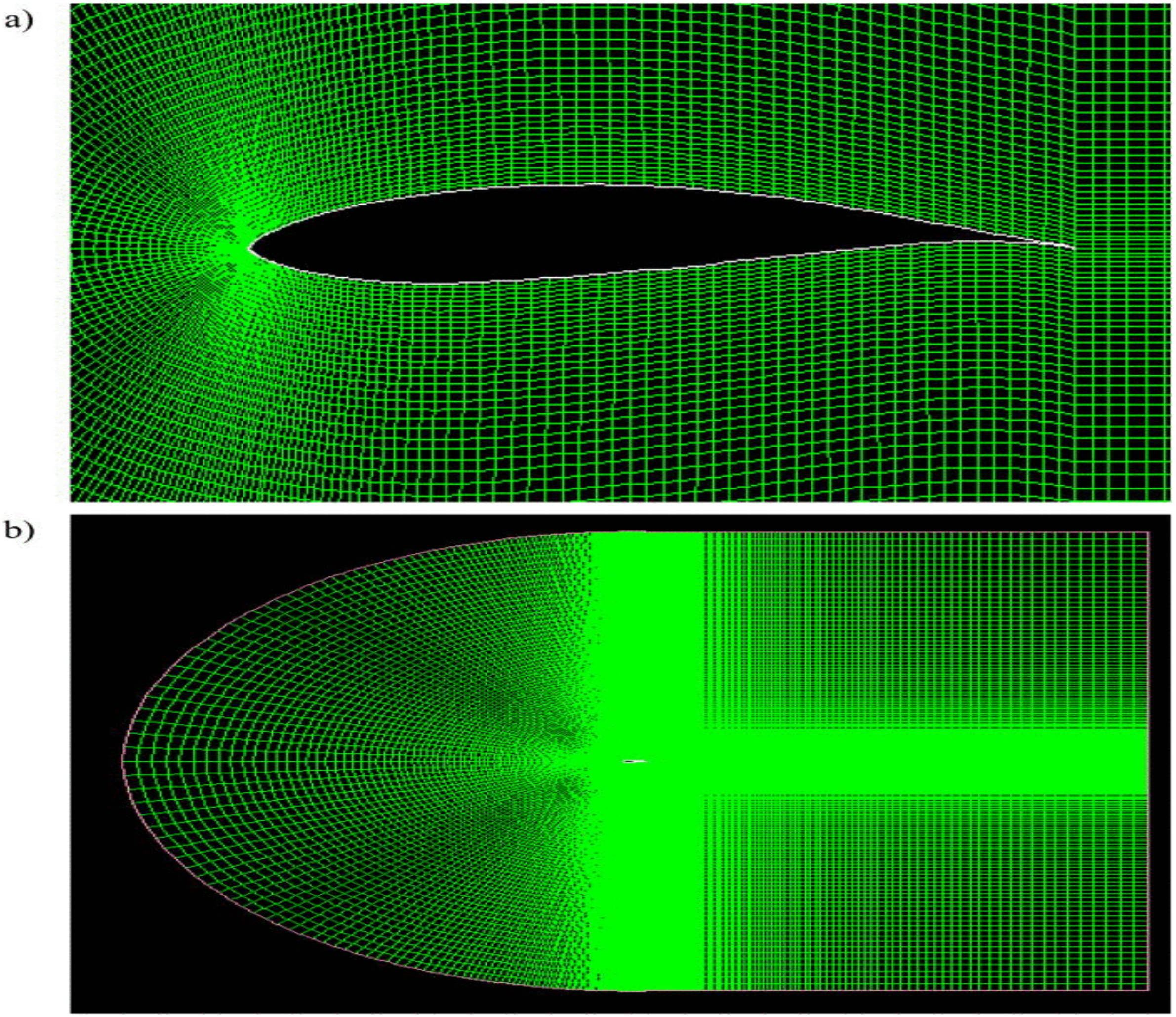
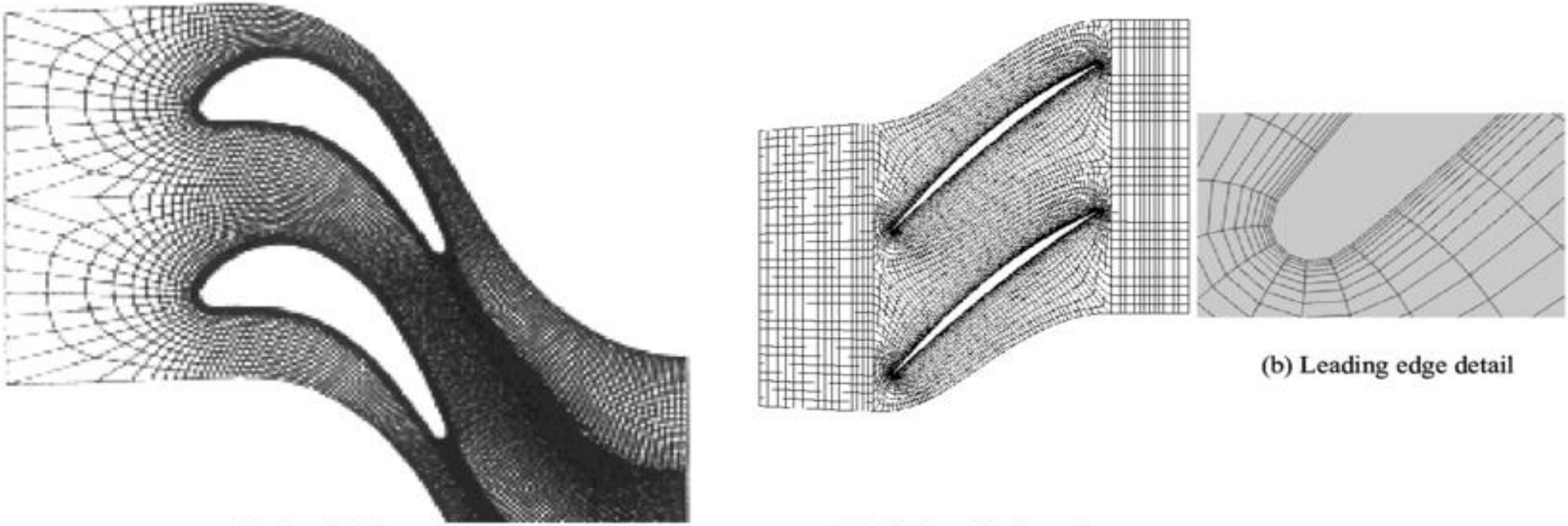


Fig: 3.10 C-H -Type mesh

H-O-H Type



Blade to blade mesh

(a) Blade to blade mesh

(b) Leading edge detail

Fig: 3.11 H-O-H -Type mesh

Butterfly Grid

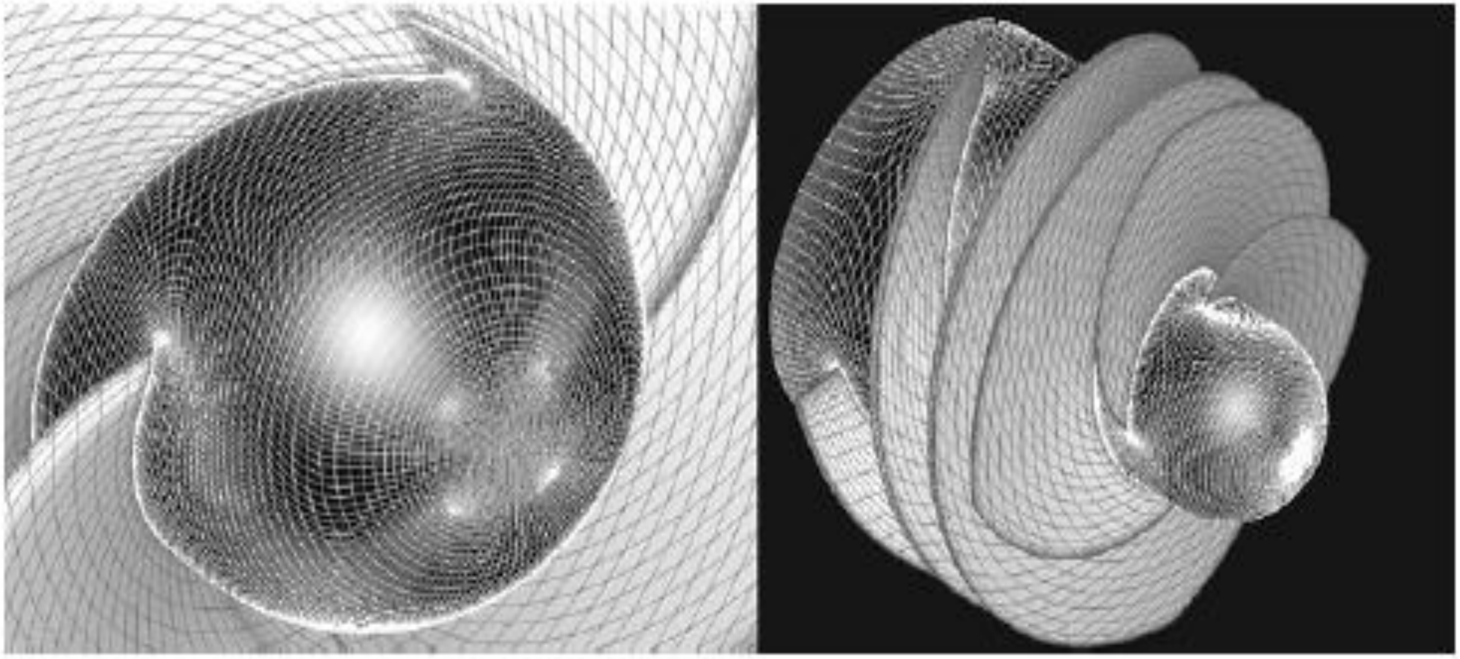


Fig: 3.12 Butterfly Grid

Overset Grid

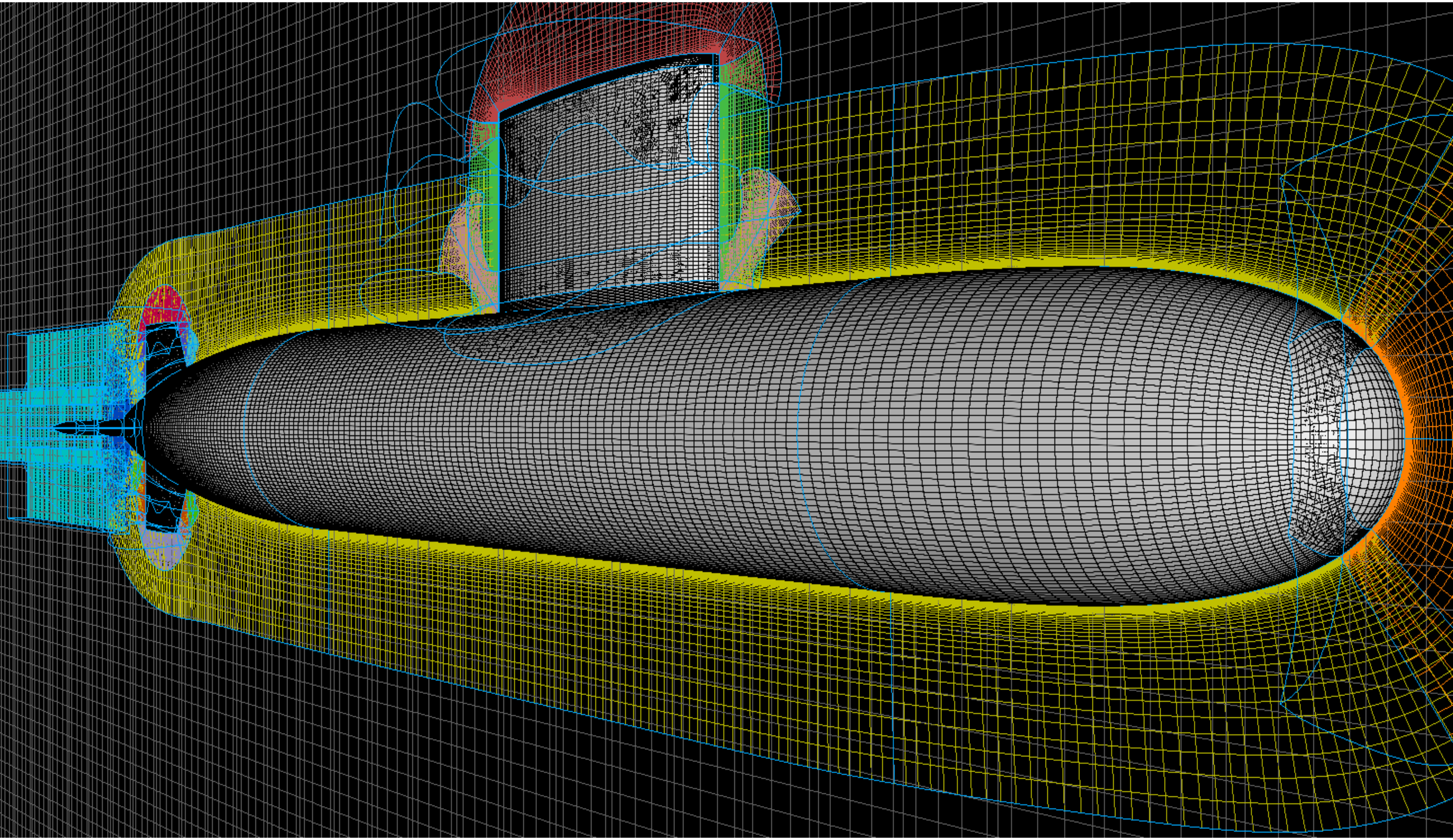


Fig: 3.13 Overset Grid

Unstructured grid

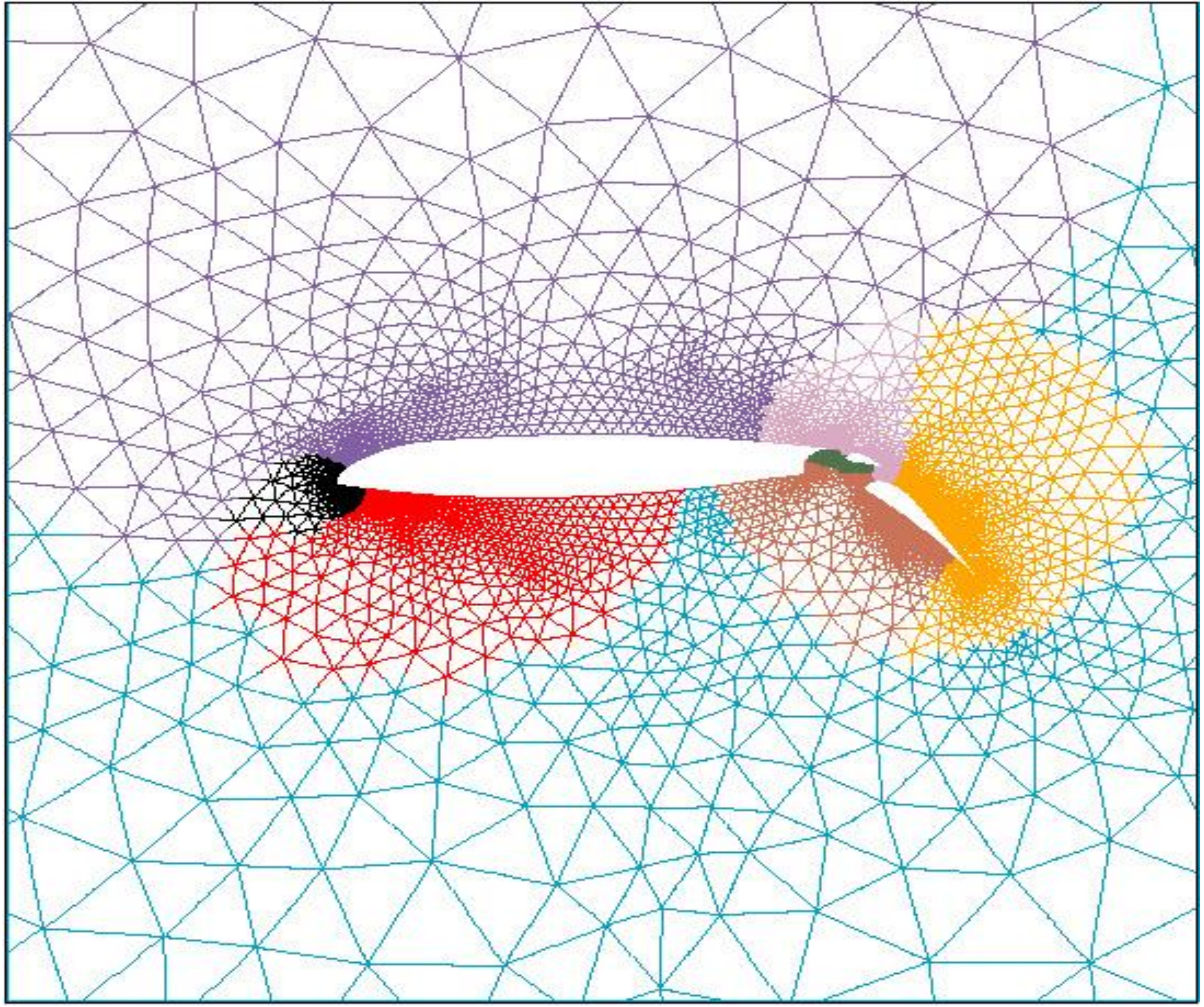


Fig: 3.14 Unstructured Grid

Tetrahedra grid

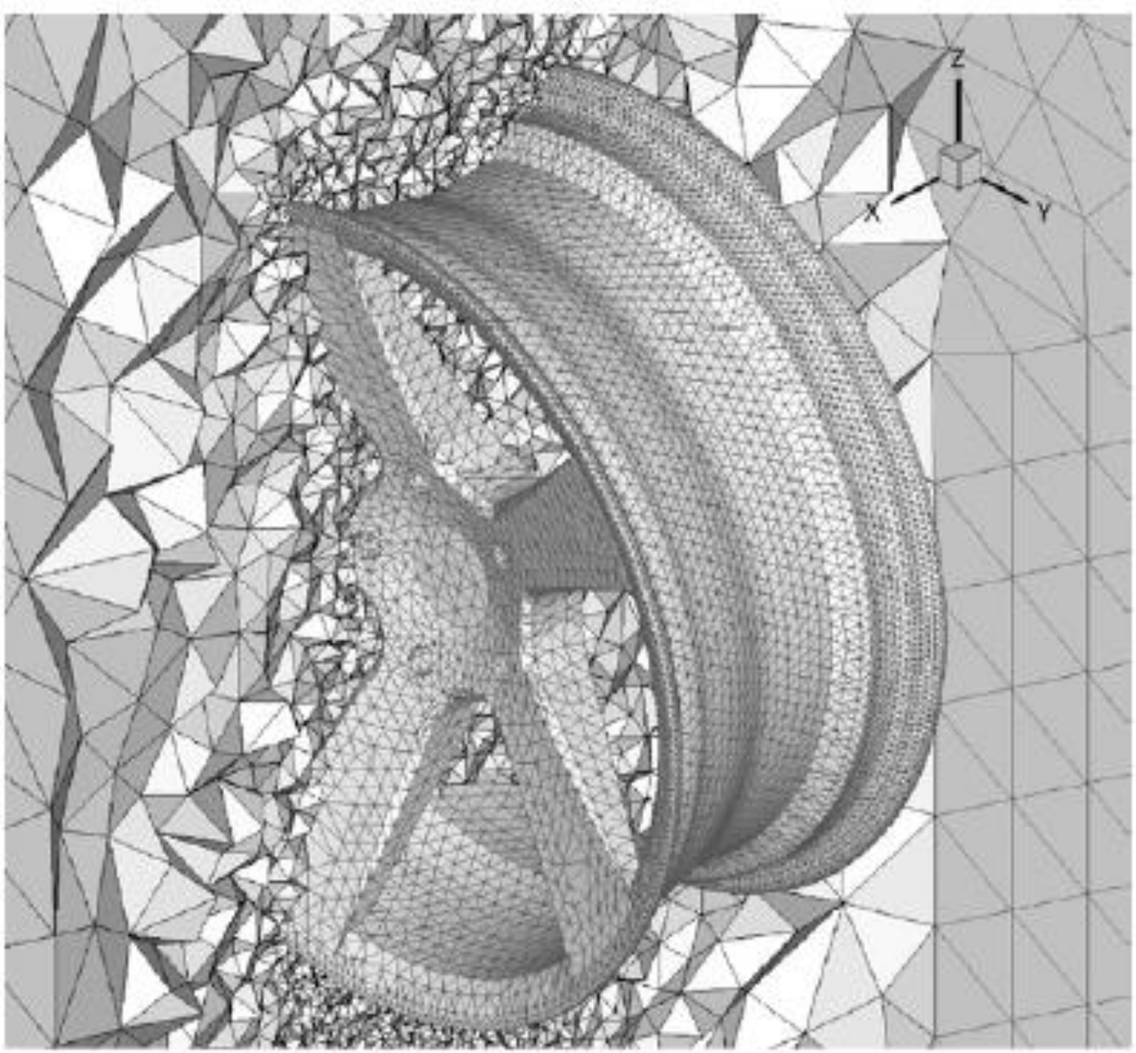


Fig: 3.15 Tetrahedra Grid

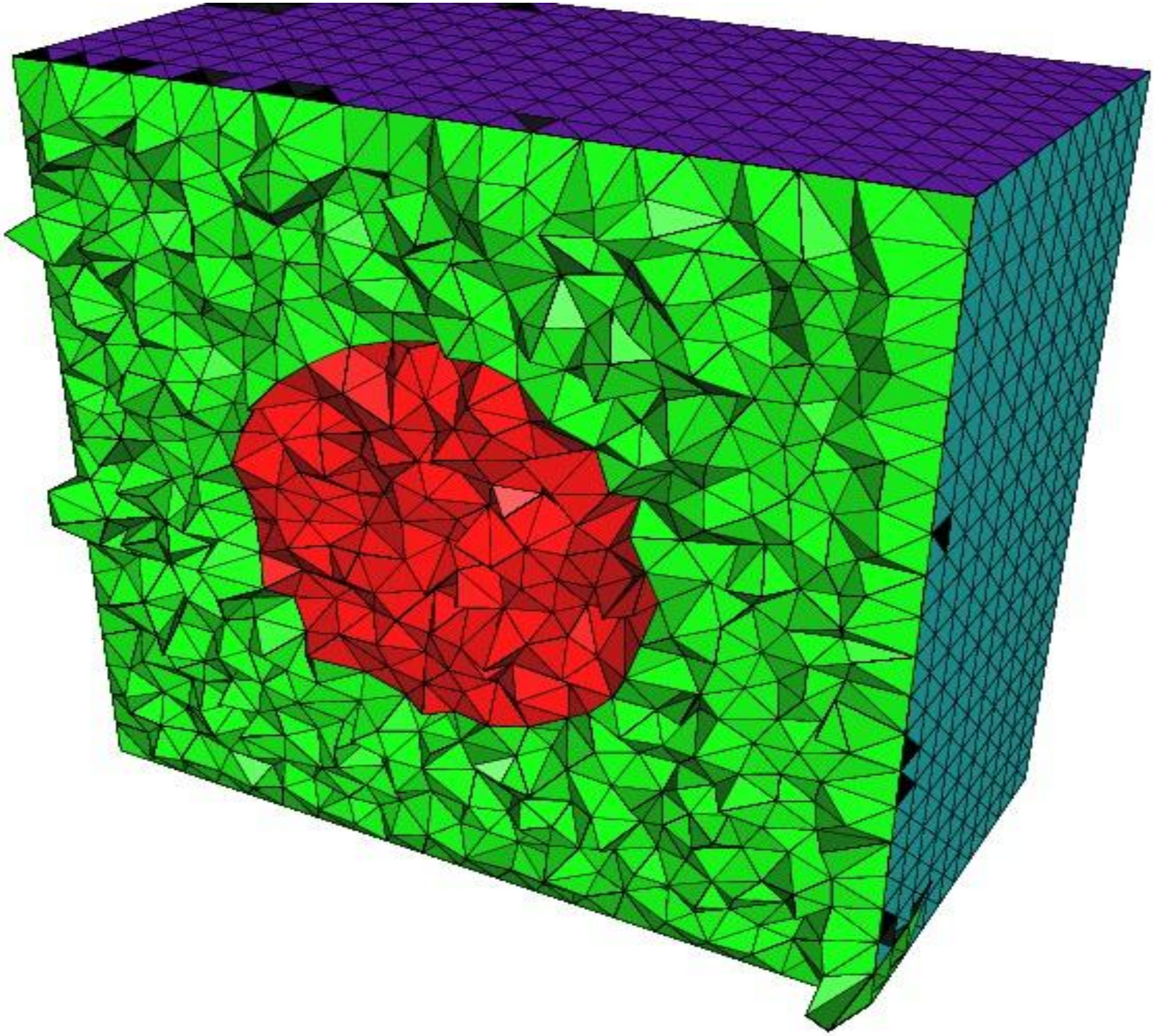


Fig: 3.16 Tetrahedra Grid

Hybrid Grid

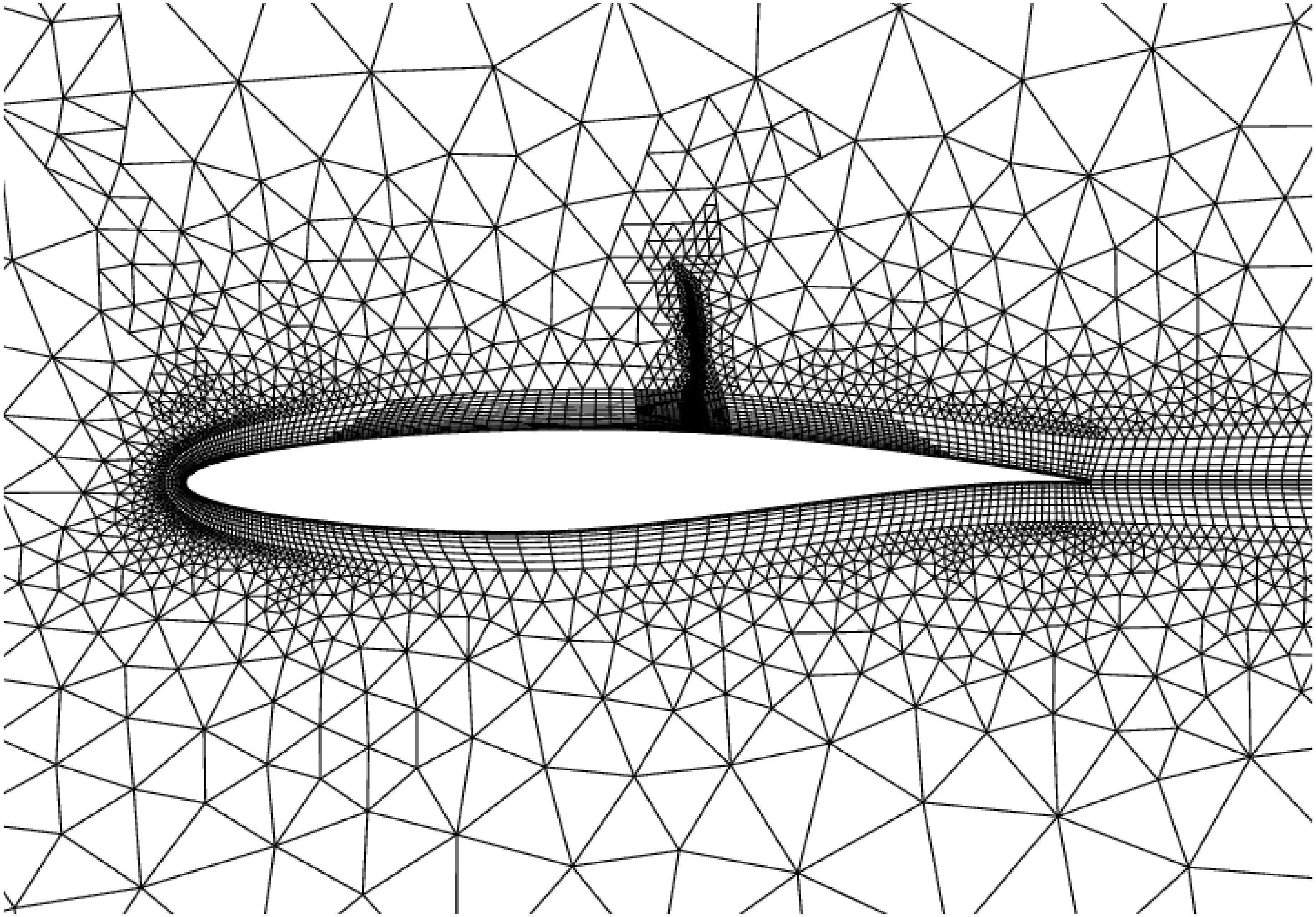


Fig: 3.17 Hybrid Grid

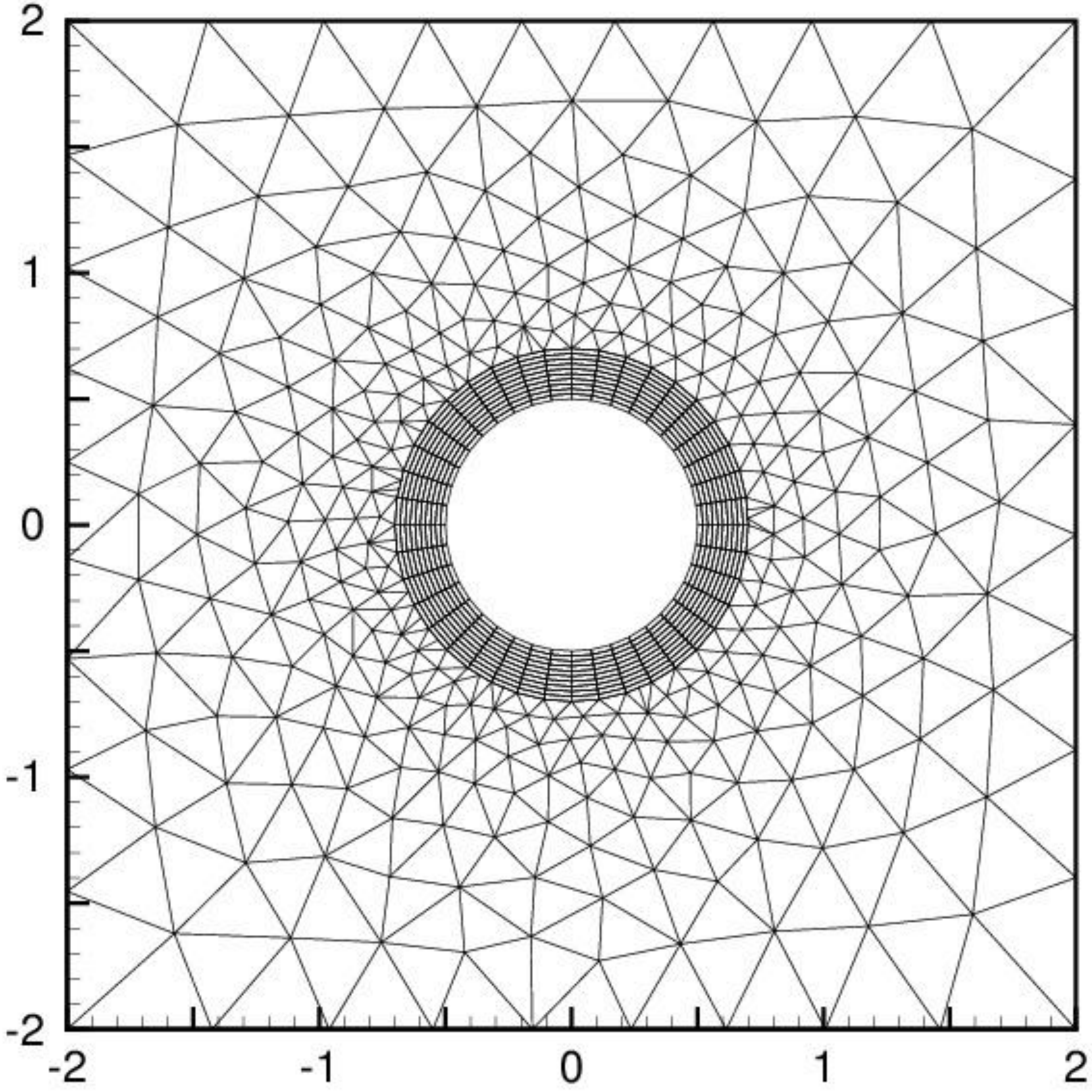


Fig: 3.18 Hybrid Grid

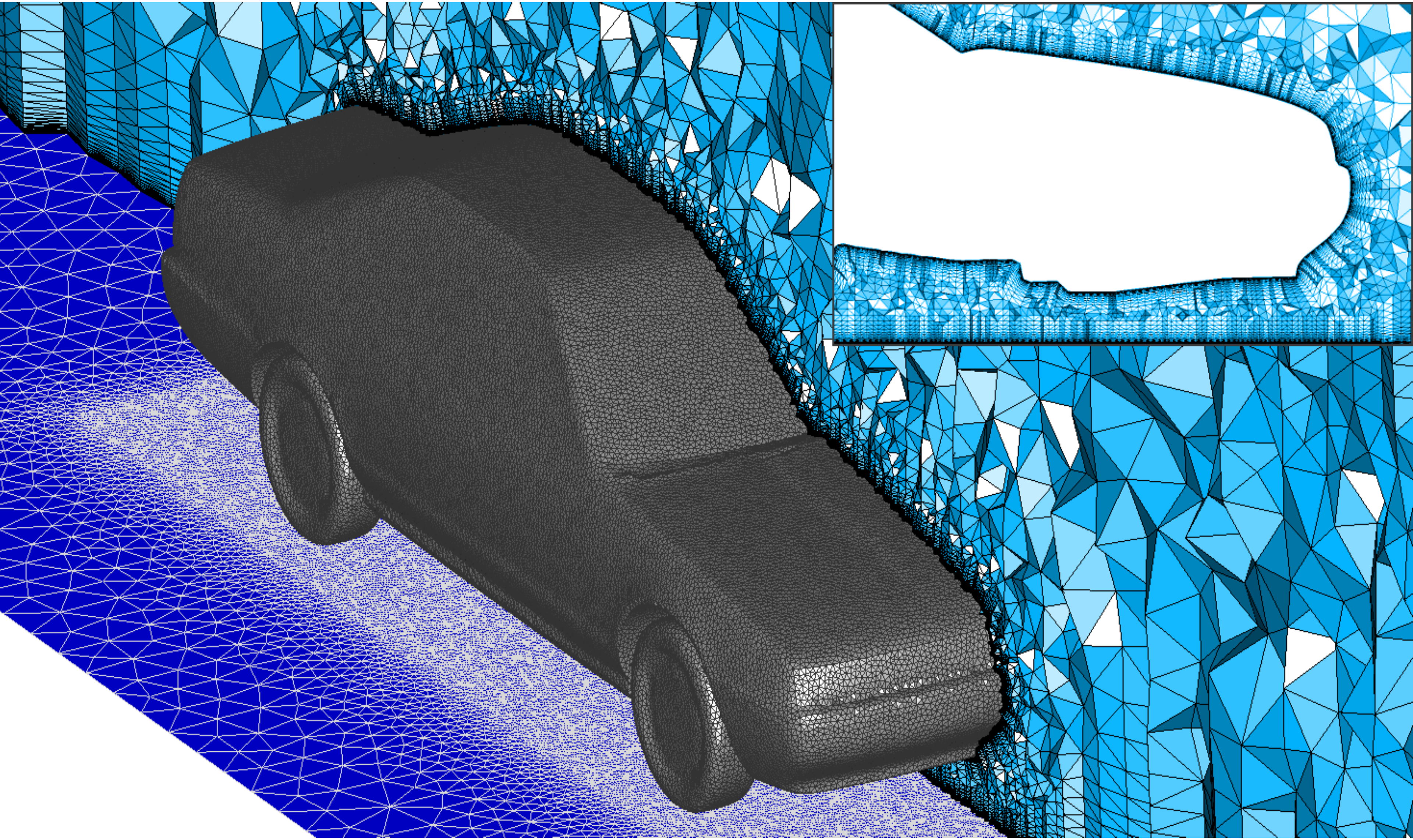


Fig: 3.19 Hybrid Grid over a car

Quadrilateral Grid

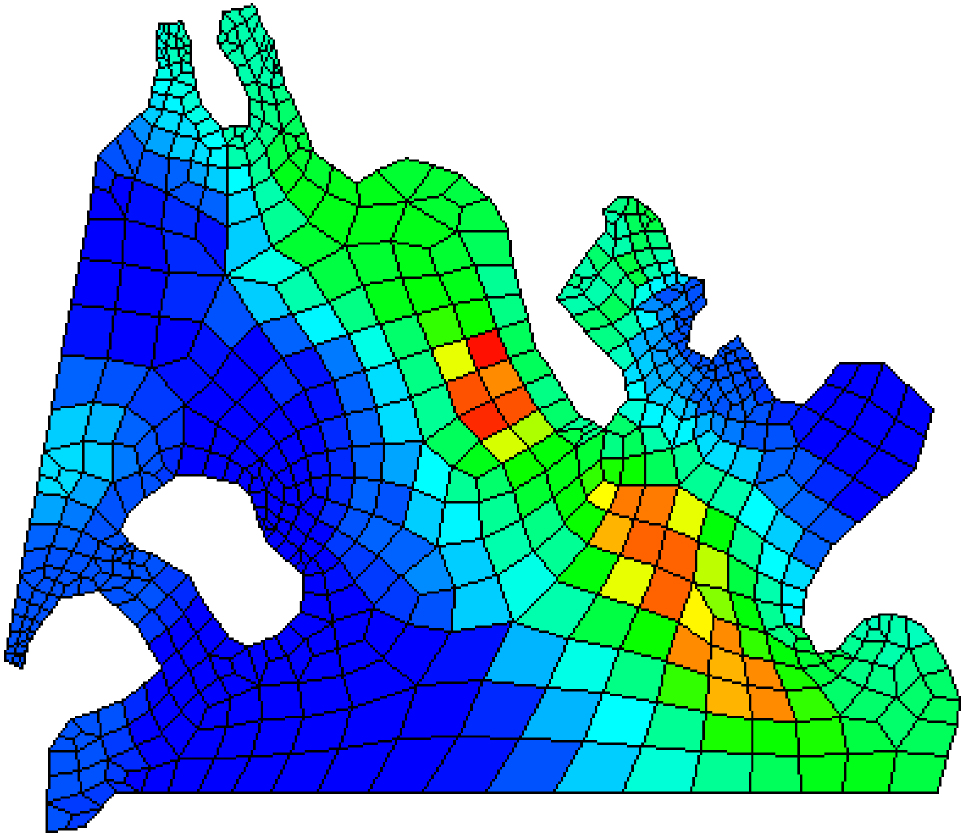


Fig: 3.20 Quadrilateral Grid

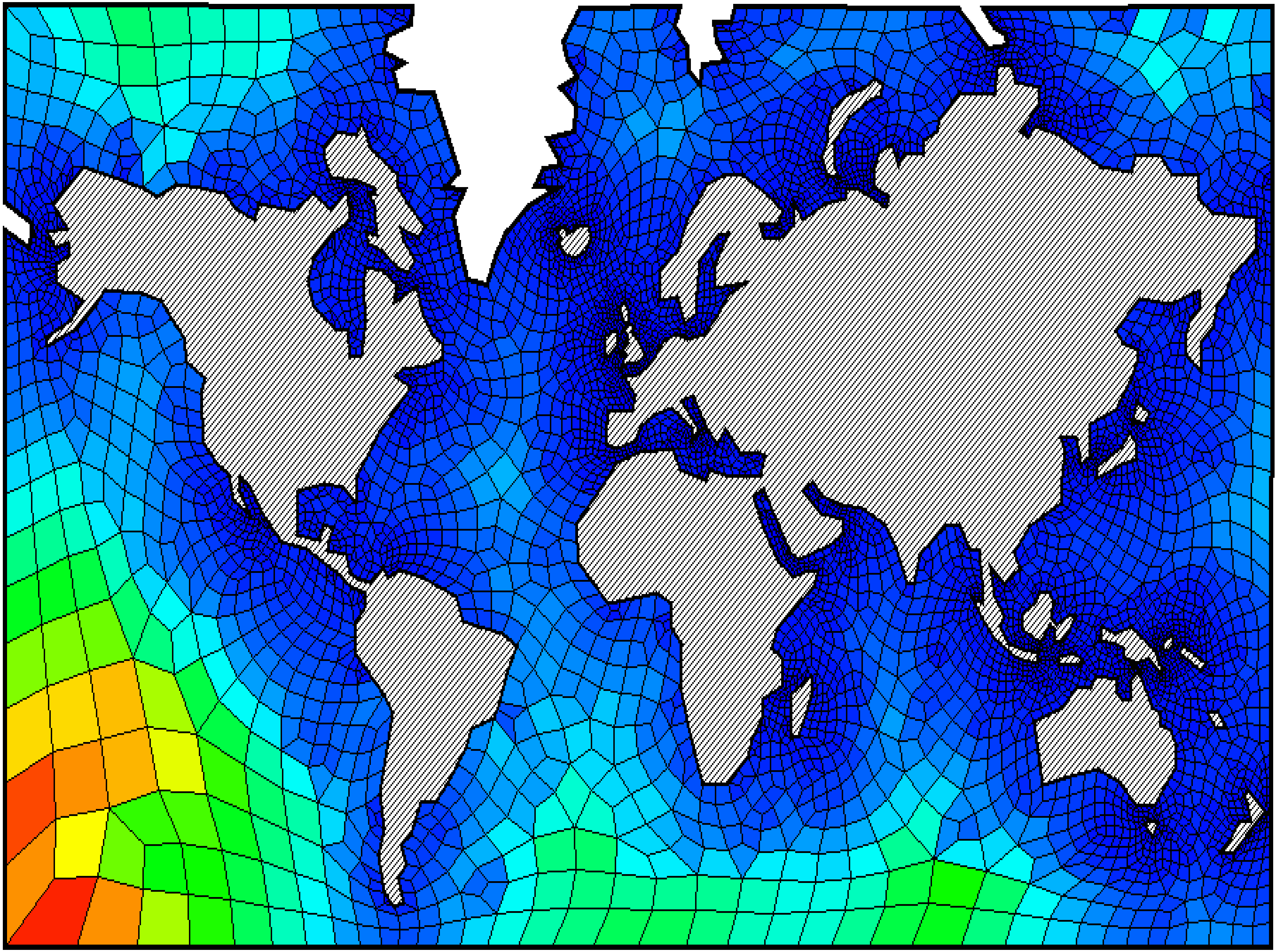


Fig: 3.20 Quadrilateral Grid generated for a world map

Hexahedral Cells

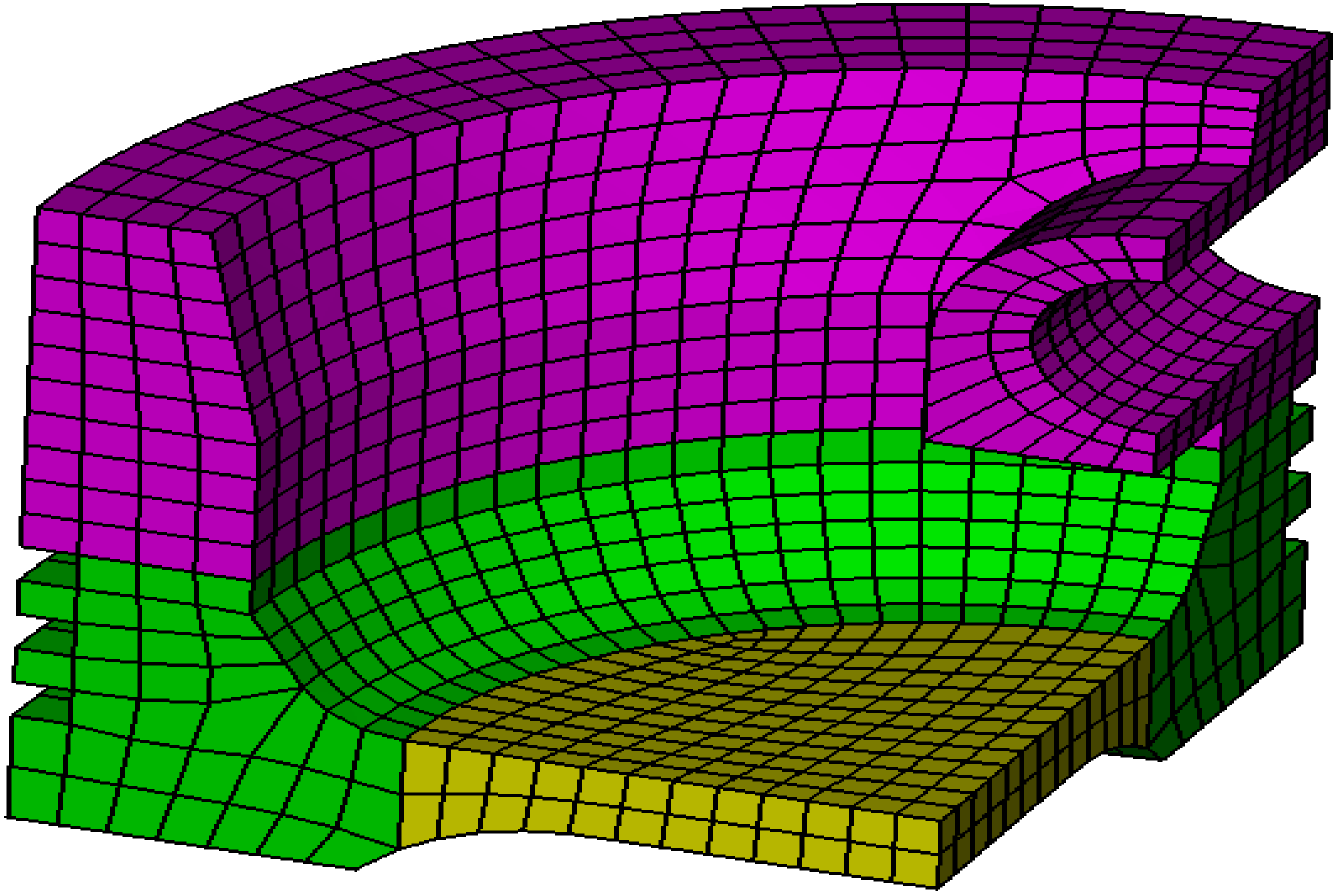


Fig: 3.21 Hexahedral Grid

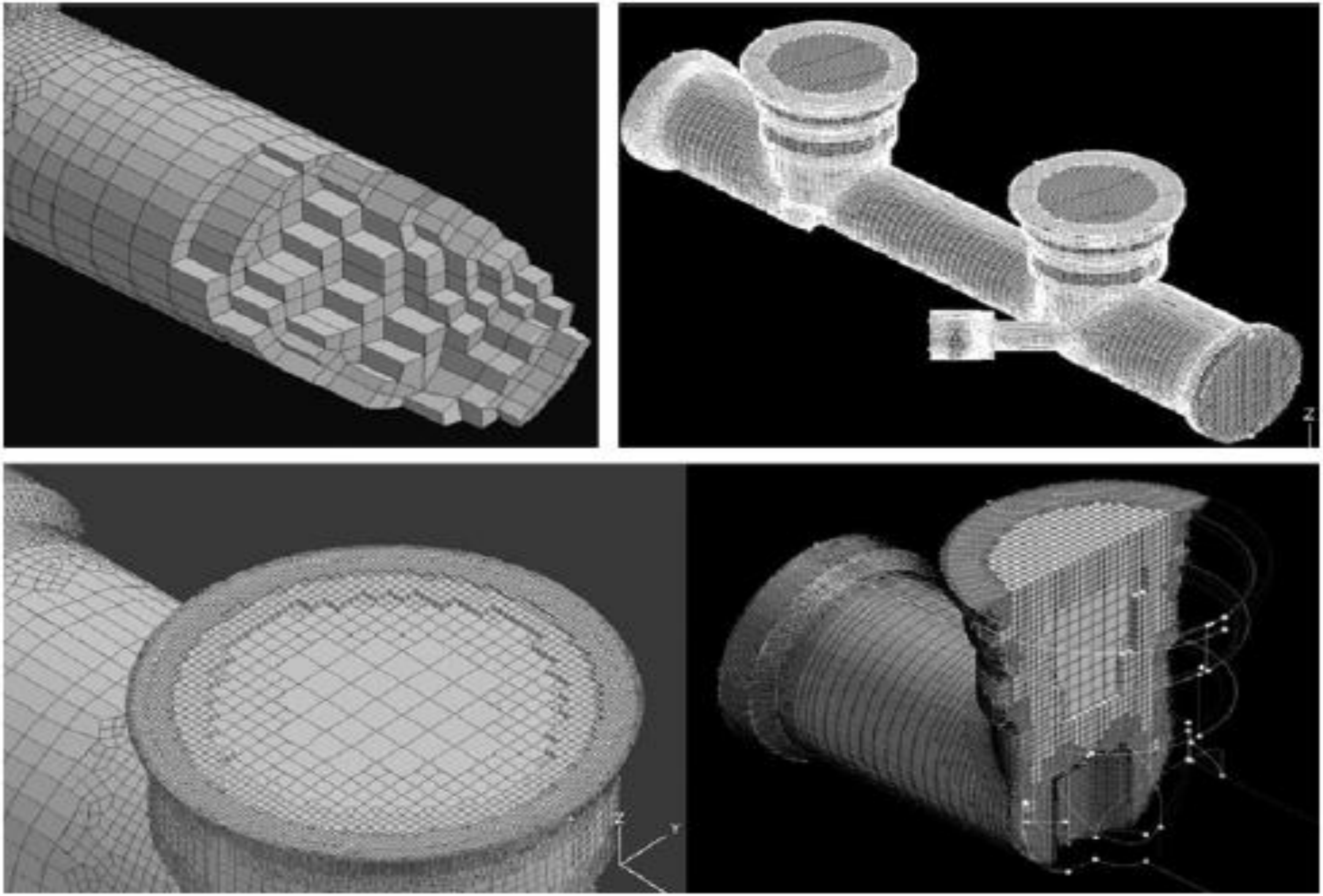


Fig: 3.22 Types of Hexahedral Grid

Structured and Hybrid Grids

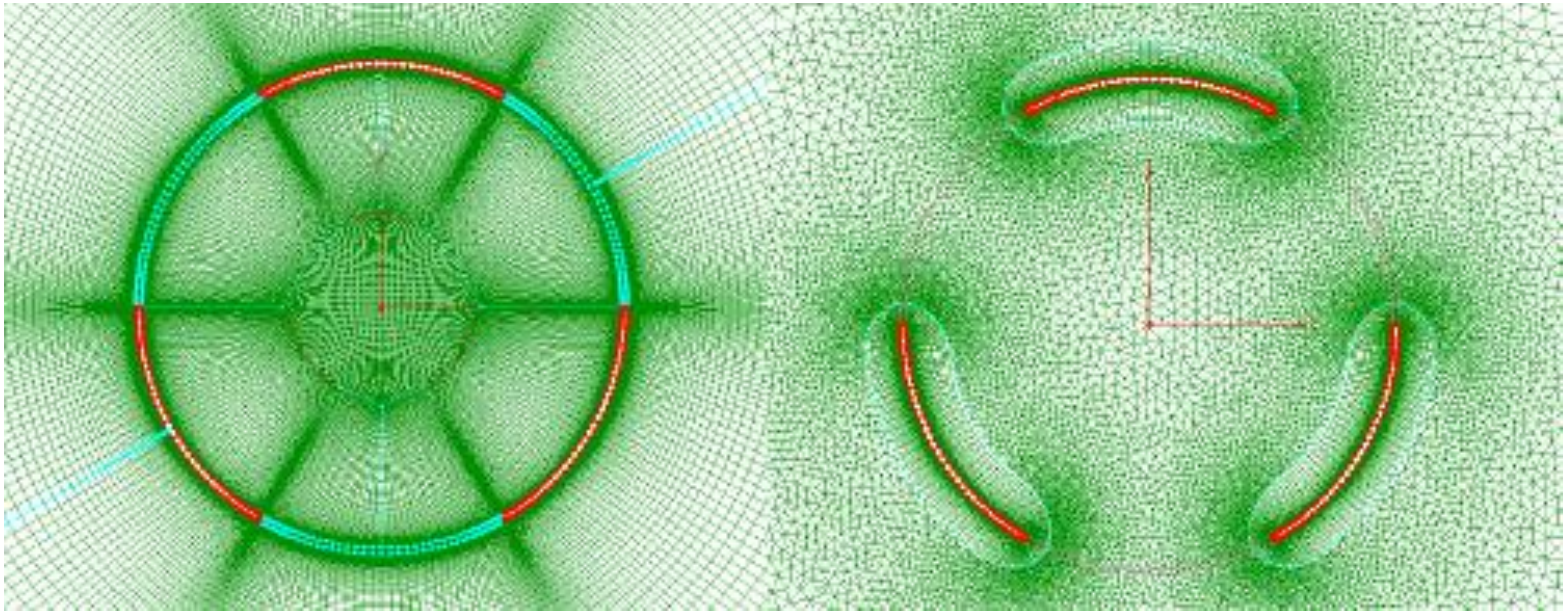


Fig: 3.23 Structured and Hybid grids

UNIT-IV

THE BASIS OF FINITE VOLUME METHODS

- The strength of the FVM is its direct connection to the physical flow properties. Indeed, the basis of the method relies on the direct discretization of the integral form of the conservation law. Integral form is the most general expression of a conservation law, as it does not require the fluxes to be continuous (property which is not satisfied for instance along shock waves or along free Surfaces). This is why we can state that the FVM is close to the physics of the flow system.

- The FVM requires setting up the following steps:

  • Subdivide the mesh, obtained from the space discretization, into finite (small) volumes, one
- Control volume being associated to each mesh point.

  • Apply the integral conservation law to each of these finite volumes.

Conditions on Finite Volume Selections

- **A cell-centered approach,** where the unknowns are at the centres of the mesh cells and the grid
- Lines define the finite volumes and surfaces. For the control volumes is indeed to make them coincide with the mesh cells. Here the variables are associated with a cell, as on Figure.1a and c. The flow variables are averaged values over the cell and can be considered as representative of some point inside the cell, for instance the central point of the cell.
- **A cell-vertex approach,** where the unknowns are defined at the corners of the mesh. Here the variables are attached to the mesh points, i.e. to the cell vertices, and d. A larger flexibility exists for the definition of the control volumes. Volumes for a consistent finite volume method have to be satisfied:
- (i) Their sum should cover the entire domain ΩJ
- (ii) The subdomains ΩJ are allowed to overlap with the conditions that each part of the surface ΩJ appears as part of an even number of different subdomains such that the overall integral conservation law holds for any combination of adjacent subdomains.
- (iii) Fluxes along a cell surface have to be computed by formulas independent of the cell in which they are considered.

Definition of the Finite Volume Discretization

- The integral conservation law is applied to each control volume ΩJ associated to mesh point J defining hereby the discretized equation for the unknowns UJ attached to that same vertex or cell. The advantage of this method, especially in absence of sources terms, is that the fluxes are calculated only on two-dimensional surfaces instead of in the three-dimensional space.
- Equation (1) is replaced by its discrete form, where the volume integrals are expressed as the averaged values over the cell and where the surface integral is replaced by a sum over all the bounding faces of the considered volume ΩJ

General Formulation of a Numerical Scheme

- A general and important interpretation of any numerical, conservative scheme is obtained directly from the integral conservation laws. The formulation that follows is valid for all possible cases, with structured grids or unstructured grids, either cell-centred or cell-vertices

Alternative formulation of the conservative condition

- Extending the subdivisions of equation (5.1.2) to an arbitrary number of cells, J =1–N, and summing over all the cells it is seen, after cancellation of the contributions from all the internal cell faces, that the sum will contain only contributions from the fluxes along the parts of the cells belonging to the boundaries of the domain and from the sources. Therefore,the conservative condition can be expressed as a requirement on the transient time evolution of the scheme. Note that for stationary sources and boundary fluxes, the right-hand side of this equation vanishes at convergence. Defining ΔUJ/Δt as the average value of ∂U/∂t over the cell ΩJ, conservation of the scheme requires that, at each time step, the following condition is to be satisfied.

UNIT-V

CFD TECHNIQUES

LAX–WENDROFF TECHNIQUE

- The **Lax–Wendroff method**, named after Peter Lax and Burton Wendroff, is a numerical method for the solution of hyperbolic partial differential equations, based on finite differences. It is second-order accurate in both space and time. This method is an example of explicit time integration where the function that defines governing equation is evaluated at the current time.
- Suppose one has an equation of the following form:
- Where *x* and *t* are independent variables, and the initial state, u(*x*, 0) is given.
- The first step in the Lax–Wendroff method calculates values for u(*x*, *t*) at half time steps, $t_{n + 1/2}$ and half grid points, $x_{i + 1/2}$. In the second step values at $t_{n + 1}$ are calculated using the data for t_n and $t_{n + 1/2}$.

- First (Lax) steps:

$$u_{i+1/2}^{n+1/2} = \frac{1}{2}(u_{i+1}^n + u_i^n) - \frac{\Delta t}{2\,\Delta x}(f(u_{i+1}^n) - f(u_i^n)),$$

$$u_{i-1/2}^{n+1/2} = \frac{1}{2}(u_i^n + u_{i-1}^n) - \frac{\Delta t}{2\,\Delta x}(f(u_i^n) - f(u_{i-1}^n)).$$

- Second step:

$$u_i^{n+1} = u_i^n - \frac{\Delta t}{\Delta x}\left[f(u_{i+1/2}^{n+1/2}) - f(u_{i-1/2}^{n+1/2})\right].$$

MAC-CORMACKS TECHNIQUE

- In computational fluid dynamics, the **MacCormack method** is a widely used discretization scheme for the numerical solution of hyperbolic partial differential equations. This second-order finite difference method was introduced by Robert W. MacCormack in 1969 The MacCormack method is elegant and easy to understand and program

$$\frac{\partial u}{\partial t} + a\frac{\partial u}{\partial x} = 0.$$

- The application of MacCormack method to the above equation proceeds in two steps; a predictor step which is followed by a corrector step.

- **Predictor step:** In the predictor step, a "provisional" value of at time level (denoted by) is estimated as follows

$$u_i^{\overline{n+1}} = u_i^n - a\frac{\Delta t}{\Delta x}\left(u_{i+1}^n - u_i^n\right)$$

- It may be noted that the above equation is obtained by replacing the spatial and temporal derivatives in the previous first order hyperbolic equation using forward differences.

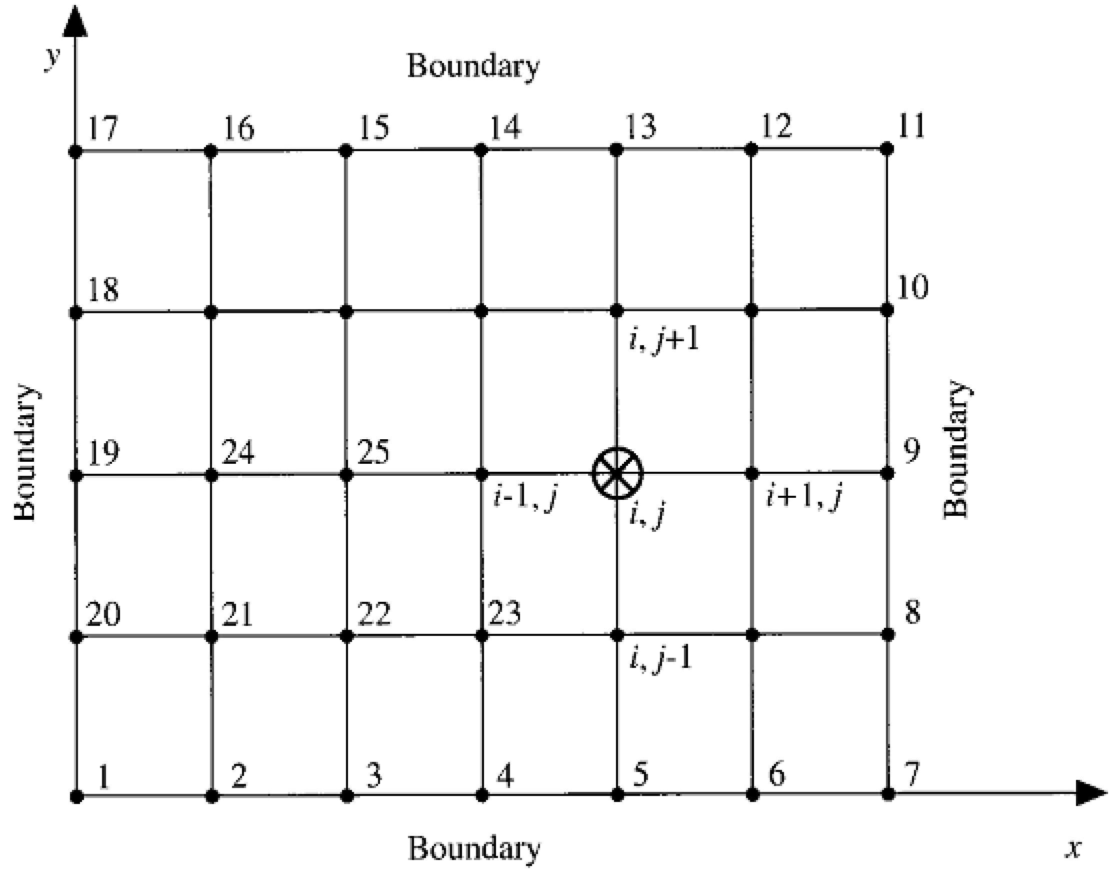
- **Corrector step:** In the corrector step, the predicted value   is corrected according to the equation

$$u_i^{n+1} = u_i^{n+1/2} - a\frac{\Delta t}{2\Delta x}\left(u_i^{\overline{n+1}} - u_{i-1}^{\overline{n+1}}\right)$$

- Note that the corrector step uses backward finite difference approximations for spatial derivative. Note also that the time-step used in the corrector step is   in contrast to the Δt used in the predictor step.
- Replacing the $u_i^{n+1/2}$ term by the temporal average
- to obtain the corrector step as

$$u_i^{n+1} = \frac{u_i^n + u_i^{\overline{n+1}}}{2} - a\frac{\Delta t}{2\Delta x}\left(u_i^{\overline{n+1}} - u_{i-1}^{\overline{n+1}}\right)$$

RELAXATION TECHNIQUE



$$\Phi_{i,j}^{n+1} = \frac{(\Delta x)^2(\Delta y)^2}{2(\Delta y)^2 + 2(\Delta x)^2}\left[\frac{\Phi_{i+1,j}^{n} + \Phi_{i-1,j}^{n}}{(\Delta x)^2} + \frac{\Phi_{i,j+1}^{n} + \Phi_{i,j-1}^{n}}{(\Delta y)^2}\right]$$

ALTERNATING-DIRECTION-IMPLICIT (ADI) TECHNIQUE

- In numerical analysis, the **Alternating Direction Implicit (ADI) method** is a finite difference method for solving parabolic, hyperbolic and elliptic partial differential equations. It is most notably used to solve the problem of heat conduction or solving the diffusion equation in two or more dimensions. It is an example of an operator splitting method

- Consider the linear diffusion equation in two dimensions,

$$\frac{\partial u}{\partial t} = \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) = (u_{xx} + u_{yy}) = \Delta u$$

- The implicit Crank–Nicolson method produces the following finite difference equation:

$$\frac{u_{ij}^{n+1} - u_{ij}^{n}}{\Delta t} = \frac{1}{2}\left(\delta_x^2 + \delta_y^2\right)\left(u_{ij}^{n+1} + u_{ij}^{n}\right)$$

- The idea behind the ADI method is to split the finite difference equations into two, one with the *x*-derivative taken implicitly and the next with the *y*-derivative taken implicitly,

$$\frac{u_{ij}^{n+1/2} - u_{ij}^{n}}{\Delta t/2} = \left(\delta_x^2 u_{ij}^{n+1/2} + \delta_y^2 u_{ij}^{n}\right)$$

$$\frac{u_{ij}^{n+1} - u_{ij}^{n+1/2}}{\Delta t/2} = \left(\delta_x^2 u_{ij}^{n+1/2} + \delta_y^2 u_{ij}^{n+1}\right).$$

PRESSURE CORRECTION TECHNIQUE

- **Pressure-correction method** is a class of methods used in computational fluid dynamics for numerically solving the Navier-Stokes equations normally for incompressible_flows.

$$\overbrace{\rho\left(\underbrace{\frac{\partial \mathbf{v}}{\partial t}}_{\substack{\text{Unsteady}\\\text{acceleration}}} + \underbrace{(\mathbf{v}\cdot\nabla)\,\mathbf{v}}_{\substack{\text{Convective}\\\text{acceleration}}}\right)}^{\text{Inertia}} = \underbrace{-\nabla p}_{\substack{\text{Pressure}\\\text{gradient}}} + \underbrace{\mu\nabla^2\mathbf{v}}_{\text{Viscosity}} + \underbrace{\mathbf{f}}_{\substack{\text{Other}\\\text{forces}}}$$

- The equations solved in this approach arise from the implicit time integration of the incompressible Navier–Stokes equations.

- **Numerical Procedures-SIMPLE, SIMPLER, SIMPLEC and PISO**

- In computational fluid dynamics (CFD), **SIMPLE algorithm** is a widely used numerical procedure to solve the Navier-Stokes equations. SIMPLE is an acronym for Semi-Implicit Method for Pressure Linked Equations.

SIMPLEC ALGORITHM

- Specify the boundary conditions and guess the initial values.
- Determine the velocity and pressure gradients.
- Calculate the pseudo velocities.
- Solve for the pressure equation and get the **p**.
- Set **p***=**p**.
- Using **p*** solve the discretized momentum equation and get **u*** and **v***.
- Solve the pressure correction equation.
- Get the pressure correction term and evaluate the corrected velocities and get **p, u, v, Φ***.
- Solve all other discretized transport equations.
- If Φ shows convergence, then STOP and if not, then set **p*=p, u*=u, v*=v, Φ*=Φ** and start the iteration again.

PISO ALGORITHM

set the boundary conditions.

- Solve the discretized momentum equation to compute an intermediate velocity field.
- Compute the mass fluxes at the cells faces.
- Solve the pressure equation.
- Correct the mass fluxes at the cell faces.
- Correct the velocities on the basis of the new pressure field.
- Update the boundary conditions.
- Repeat from 3 for the prescribed number of times.
- Increase the time step and repeat from 1.

SIMPLER ALGORITHM

- Guess velocity field
- Compute momentum coefficients and store.
- Compute pressure coefficients and store.
- Solve pressure equation and obtain pressure
- Solve momentum equations using stored momentum
- coefficients and just-computed pressure. Find u* andv*
- Find b term in pressure-correction equation using u*