Reg.No: | | | | | | | |

SNS College of Technology, Coimbatore-35.

(Autonomous)

B.E/B.Tech- Internal Assessment -III

Academic Year 2023-2024 (Even Semester)

Sixth Semester

Aerospace Engineering

19ASB304 Computational Fluid Dynamics for Aerospace Application

A

Time: $1^{1/2}$ Hours **Maximum Marks: 50**

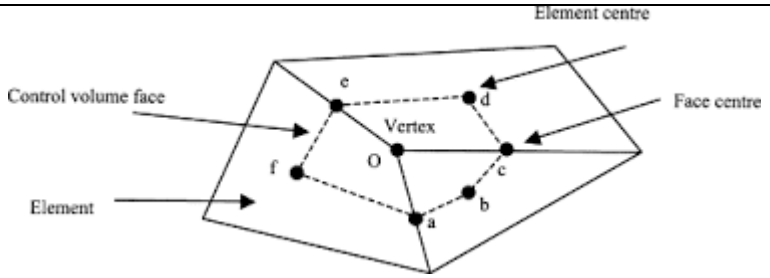
Answer All Questions

PART - A (5x 2 = 10 Marks)

		CO	Blooms
1.	What is the finite volume method in CFD? The finite volume method is a method for solving partial differential equations like the Navier-Stokes equations in the form of algebraic equations. The physical parameters are approximated at discrete nodes surrounded by finite volumes within the problem domain.	CO 4	Rem
2.	Differentiate between cell-centered and vertex centered method. For the cell centered method, the (finite) volumes used to satisfy the integral form of the equation are the mesh elements itself. For the vertex centered approach, the finite volumes are elements of the mesh dual to the computational mesh.	CO 4	App
3.	What is Lax-Wendroff time-stepping? The Lax-Wendroff method is a single step method for evolving time dependent solutions governed by partial differential equations, in contrast to Runge-Kutta methods that need multiple stages per time step.	CO 4	Und
4.	When to use LES model? LES prioritizes larger eddies influenced by flow geometry, using a subgrid-scale model for smaller, more universal scales. LES is ideal for capturing transient turbulent structures, unlike time-averaged RANS models.	CO 5	App
5.	Is K-Omega a RANS model? The k-omega ( ) turbulence model is one of the most commonly used models to capture the effect of turbulent flow conditions. It belongs to the Reynolds-averaged Navier-Stokes (RANS) family of turbulence models where all the effects of turbulence are modeled.	CO 5	App

PART – B (13+13+14 =40 Marks)

				CO	Blooms
6.	(a)	What does cell-centered formulation mean in terms of finite volume techniques?	13	CO 4	Rem



Finite Volume Method (FVM)

The finite volume method is a numerical technique used to solve PDEs. It involves dividing the computational domain into a finite number of small control volumes (or cells) and then integrating the governing equations over each control volume. The key idea is to ensure the conservation of fluxes across the boundaries of each control volume.

Cell-Centered Formulation

In the cell-centered formulation of the finite volume method, the primary variables (such as velocity, pressure, temperature, etc.) are stored at the geometric centers of the control volumes. Here's how it works:

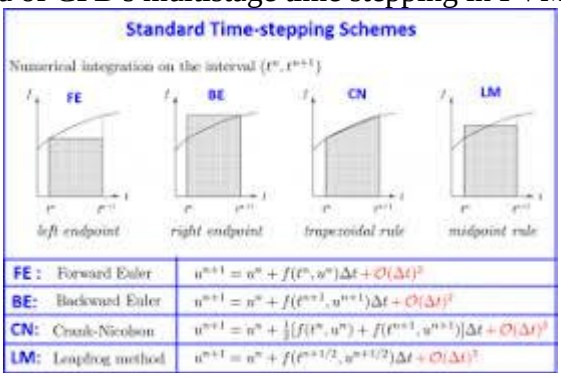
1. **Domain Discretization**: The computational domain is divided into a grid of non-overlapping control volumes (cells). These cells can be of various shapes, such as quadrilateral (in 2D), hexahedral (in 3D), or more complex polyhedral shapes.
2. **Variable Storage**: Each cell has a representative point, usually its geometric center, where the primary variables are defined. These variables are averaged values over the cell.
3. **Flux Calculation**: To compute the fluxes of quantities like mass, momentum, or energy across the cell faces, the values at the cell centers are used. The fluxes are typically calculated at the cell faces, which involves interpolating or averaging the values from the adjacent cell centers.
4. **Conservation Equations**: The integral form of the conservation equations is applied to each control volume. This involves summing the fluxes across the boundaries of the cell and ensuring that the net flux satisfies the conservation laws.

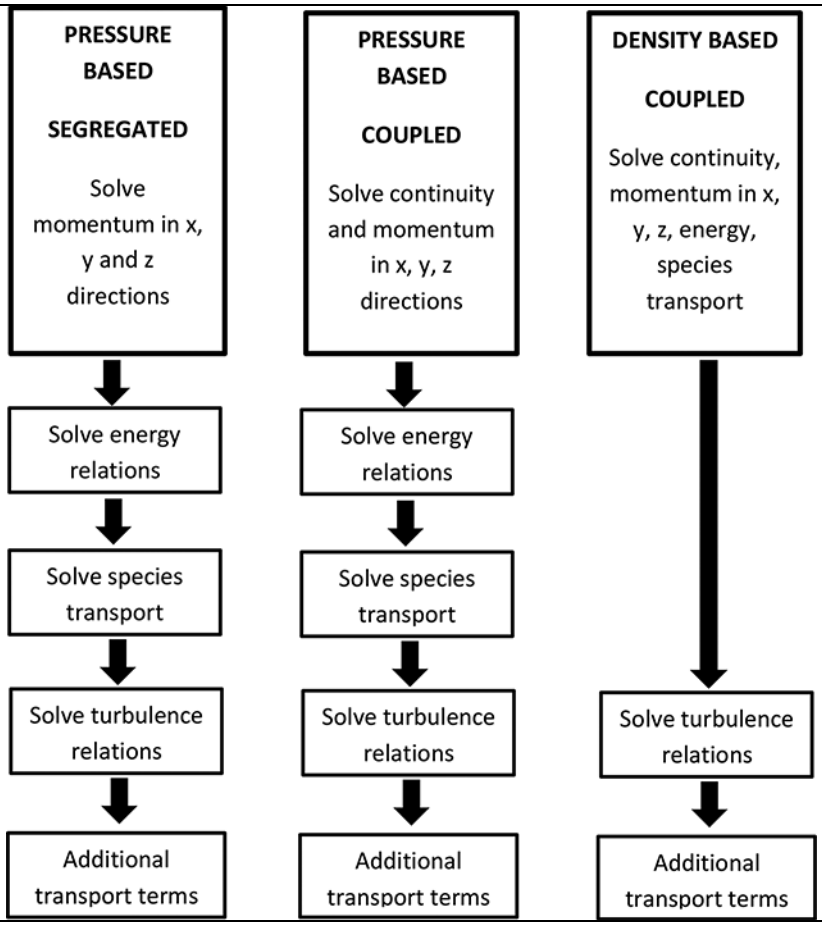
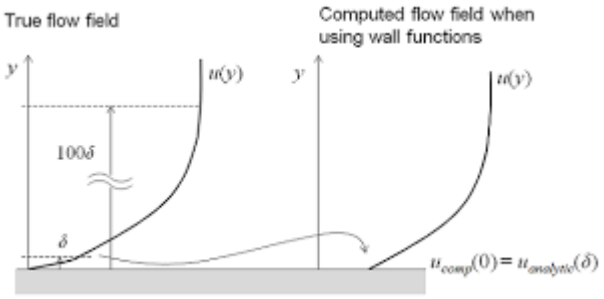
Advantages of Cell-Centered Formulation

- **Simplicity**: The cell-centered formulation is straightforward to implement since it uses a single set of variables located at the cell centers.
- **Conservation**: It inherently ensures conservation of quantities within each control volume, which is a key requirement for accurate physical simulations.
- **Flexibility**: It can handle complex geometries and unstructured meshes, making it suitable for a wide range of applications.

Comparison with Node-Centered Formulation

In contrast, a **node-centered formulation** (or vertex-centered formulation) places the primary variables at the vertices (nodes) of the control volumes rather than at the cell centers. While both formulations aim to solve the same governing equations, they differ in how they interpolate and calculate fluxes, as well as in their computational efficiency and ease of handling boundary conditions.

		(or)			
	(b)	Describe the idea of CFD's multistage time stepping in FVM.  1. **Stage 1**: Compute the first intermediate solution using the initial state: $$U^{(1)} = U^n + \Delta t\, \mathcal{R}(U^n)$$ where U represents the solution vector, and $\mathcal{R}(U^n)$ represents the discretized residual (fluxes and source terms). 2. **Stage 2**: Compute the second intermediate solution using the result of the first stage: $$U^{(2)} = U^n + \frac{\Delta t}{2}\left(\mathcal{R}(U^n) + \mathcal{R}(U^{(1)})\right)$$ 3. **Final Update**: Combine the intermediate solutions to compute the final solution at the new time level: U^{n+1} = U^n + \frac{\Delta t	13	CO 4	Und
7.	(a)	Elucidate the CFD turbulence models' pressure and velocity corrections.	13	CO 5	Und

					
		(or)			
	(b)	Write about CFD models utilizing algebraic mixing length models. 1. **Mixing Length Hypothesis**: The mixing length l_m is analogous to the mean free path in molecular theory, representing the average distance a fluid particle travels before its turbulent energy is dissipated. In these models, the eddy viscosity ν_t is calculated as: $$\nu_t = l_m^2 \lvert \partial u / \partial y \rvert$$ where $\lvert \partial u / \partial y \rvert$ is the velocity gradient perpendicular to the flow direction. 2. **Eddy Viscosity**: The concept of eddy viscosity ν_t is introduced to model the turbulent stresses. The Reynolds stresses are modeled using the Boussinesq approximation: 	13	CO 5	App
8.	(a)	Explain the concept behind Runge-Kutta time stepping in CFD.	14	CO 4	Cre

Concept of Runge-Kutta Methods

Runge-Kutta methods are designed to approximate the solution of initial value problems of the form

$$\frac{dy}{dt} = f(t, y), \quad y(t_0) = y_0$$

Here, y is the dependent variable (such as velocity or pressure in a fluid), t is time, and $f(t, y)$ is a function that defines the rate of change of y.

General Approach

The idea behind Runge-Kutta methods is to compute the solution at the next time step (y_{n+1}) by taking a weighted average of several increment estimates within the current time step (t_n to t_{n+1}). These increments are based on the function $f(t, y)$ evaluated at different points within the time interval.

Example: The Classical Fourth-Order Runge-Kutta Method (RK4)

The classical fourth-order Runge-Kutta method (RK4) is one of the most commonly used due to its balance of accuracy and computational efficiency. The RK4 method uses four intermediate steps (or stages) to calculate the next value y_{n+1} from the current value y_n:

1. **Compute the first increment:**

$$k_1 = f(t_n, y_n)$$

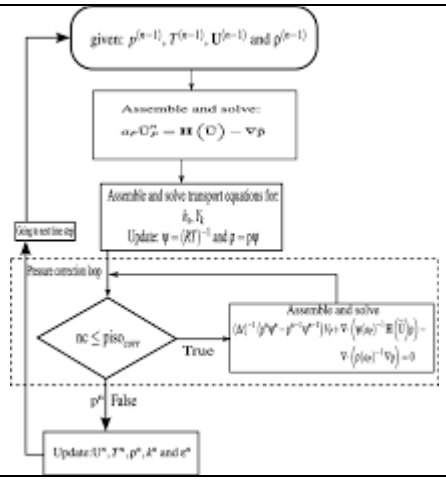
2. **Compute the second increment:**

$$k_2 = f\left(t_n + \frac{\Delta t}{2}, y_n + \frac{\Delta t}{2} k_1\right)$$

3. **Compute the third increment:**

$$k_3 = f\left(\downarrow + \frac{\Delta t}{2}, y_n + \frac{\Delta t}{2} k_2\right)$$

		(or)			
	(b)	Create the PISO algorithms used in CFD turbulence models.	14	CO 5	Cre

Abbreviations: Rem- Remembering Und-Understanding App-Applying Ana-Analyzing Eva-Evaluating Cre-Creating