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

B

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

Answer All Questions

PART - A (5x 2 = 10 Marks)

		CO	Blooms
1.	Why trade futures over CFDs? One of the most important ones is that futures have a set date for completion which will, ultimately, come with a set price. On one hand, this means that trading in futures can be more transparent than CFDs. On the other hand, this means that CFDs can be more flexible.	CO4	Und
2.	What is the central difference scheme in CFD? The central difference scheme is used for discretization of both the advection and the diffusion term, and the Gauss–Seidel method is used for solution of the resulting finite difference equations.	CO4	Rem
3.	Are CFD over-the-counter derivatives? CFDs trade over-the-counter (OTC) through a network of brokers that organize the market demand and supply for CFDs and make prices accordingly. In other words, CFDs are not traded on major exchanges such as the New York Stock Exchange (NYSE).	CO4	App
4.	Why is LES better than RANS? Steady models (RANS turbulence model) simulate the same result but are averaged in time. Transient turbulent structure can only be simulated by LES. Indeed, we remark that, except in cases of very simple flow situations, RANS models are never able to accurately reproduce an entire flow field.	CO5	Und
5.	How to choose a turbulence model? The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation.	CO5	App

PART – B (13+13+14 =40 Marks)

				CO	Blooms
6.	(a)	Describe the idea of CFD's Lax-Wendroff time stepping.	13	CO4	Und

<table>
<tr><td></td><td></td><td>

Lax-Wendroff Method

Lax step

$$u_{j+1/2}^{n+1/2} = \frac{1}{2}(u_{j+1}^n + u_j^n) - \frac{\Delta t}{2\Delta x}(F_{j+1}^n - F_j^n)$$

Compute Fluxes at n+1/2 and then:

$$u_j^{n+1} = u_j^n - \frac{\Delta t}{\Delta x}\left(F_{j+1/2}^{n+1/2} - F_{j-1/2}^{n+1/2}\right)$$

- Stable if CFL-condition fulfilled.
- Still diffusive, but here this is only 4th order in k, compared to 2th order for Lax method.

=> Much smaller effect.

</td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td>(or)</td><td></td><td></td><td></td></tr>
<tr><td></td><td>(b)</td><td>

Using multi-stage stepping, describe the idea of CFD time stepping.

In transient CFD simulations, we simulate the flow in real-time. This is solved by starting at \(t = 0 \) and using time increments \(\Delta t \) to calculate the next time step.

$$ \ t_{n+1} = \ t_{n} + \Delta t \tag{1} $$

Where:

- \(t_{n+1} \) is the next time step
- \(t_{n} \) is the current time step
- \(\Delta t \) is the time increment

Constant Delta T

The first option is to set \(\Delta t \) to a constant value, which means that \(\Delta t \) will not change throughout the transient simulation. To set up \(\Delta t \) as constant, open the simulation control panel and set *Adjustable time step* to *False*.

A
G

</td><td>13</td><td>CO4</td><td>App</td></tr>
<tr><td>7.</td><td>(a)</td><td>

Provide an explanation of the pressure correction equation for CFD turbulence models.

In Computational Fluid Dynamics (CFD), turbulence models are essential for simulating turbulent flow, which occurs in many practical engineering applications. The pressure correction equation is a key component in some turbulence models, particularly those based on the Reynolds-averaged Navier-Stokes (RANS)

Continuity $$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0$$

Momentum $$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}$$

Energy $$\frac{\partial}{\partial t}(\rho h_{tot}) + \frac{\partial}{\partial x_j}(\rho h_{tot} u_j) = \frac{\partial P}{\partial t} + \frac{\partial}{\partial x_j}(u_i \tau_{ij} + \lambda \frac{\partial T}{\partial x_j})$$

where $\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} + \frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_j}\right)$ $\quad h_{tot} = h + \frac{1}{2}u_i^2$

</td><td>13</td><td>CO5</td><td>Und</td></tr>
<tr><td></td><td></td><td>(or)</td><td></td><td></td><td></td></tr>
</table>

	(b)	Write about CFD models using algebraic mixing length models. A turbulent flow field is characterized by velocity fluctuations in all directions and has an infinite number of scales (degrees of freedom). Solving the NS equations for a turbulent flow is impossible because the equations are elliptic, non-linear, coupled (pressure-velocity, temperature-velocity). The flow is three dimensional, chaotic, diffusive, dissipative, and intermittent. The most important characteristic of a turbulent flow is the infinite number of scales so that a full numerical resolution of the flow requires the construction of a grid with a number of nodes that is proportional to $Re^{9/4}$. The governing equations for a Newtonian fluid are Conservation of Mass $$\frac{\partial \rho}{\partial t} + \frac{\partial \rho \tilde{u}_i}{\partial x_i} = 0 \qquad (1)$$ Conservation of momentum $$\frac{\partial \rho \tilde{u}_i}{\partial \tau} + \frac{\partial \rho \tilde{u}_j \tilde{u}_i}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \tilde{u}_i}{\partial x_j}\right) - \frac{\partial \tilde{p}}{\partial x_i} + \rho g_i + \tilde{s}_{ui} \qquad (2)$$ Conservation of passive scalars (given a scalar $\tilde{T}$) $$\frac{\partial \rho c_p \tilde{T}}{\partial \tau} + \frac{\partial \rho c_p \tilde{u}_j \tilde{T}}{\partial x_j} = \frac{\partial}{\partial x_j}\left(k \frac{\partial \tilde{T}}{\partial x_j}\right) + \tilde{s}_t \qquad (3)$$	13	CO5	App
8.	(a)	Describe the discretization of the central and upwind types in CFD. **Central Difference Scheme:** The central difference scheme is a widely used method for discretizing derivatives in CFD. It approximates derivatives by using the function values at points symmetrically located around the point where the derivative is being evaluated. **Discretization of First-Order Derivative:** For a function $f(x)$, the central difference scheme for the first-order derivative $\frac{\partial f}{\partial x}$ at a point x_i can be expressed as: $$\frac{\partial f}{\partial x} \approx \frac{f(x_{i+1}) - f(x_{i-1})}{2\Delta x}$$ where Δx is the grid spacing.	14	CO4	Cre

		Discretization of First-Order Derivative: For the first-order derivative $\frac{\partial f}{\partial x}$, the upwind scheme considers the flow direction. If the flow is in the positive direction (e.g., $u > 0$), the upwind difference scheme at point x_i can be expressed as: $\frac{\partial f}{\partial x} \approx \frac{f(x_i)-f(x_{i-1})}{\Delta x}$ Similarly, if the flow is in the negative direction (e.g., $u < 0$), the scheme becomes: $\frac{\partial f}{\partial x} \approx \frac{f(x_{i+1})-f(x_i)}{\Delta x}$ **Discretization of Second-Order Derivative:** The upwind scheme for the second-order derivative $\frac{\partial^2 f}{\partial x^2}$ incorporates a similar approach, considering the flow direction while approximating the derivative.			
		(or)			
	(b)	Write a case study using CFD for the stage separation aerodynamics of upco **Methodology:** 1. **Geometry Generation**: The first step involves creating detailed 3D geometries of the launch vehicle and its stages using CAD software. This includes the main rocket body, payload, and individual stages to be separated. 2. **Mesh Generation**: A high-quality computational mesh is generated around the entire geometry, focusing on regions of interest such as the separation interfaces and flow field around the vehicles. Special attention is paid to capturing boundary layer effects and resolving shock waves accurately. 3. **CFD Simulation Setup**: CFD simulations are performed using software packages capable of solving the compressible Navier-Stokes equations. The simulations consider various operating conditions, such as different separation velocities, angles, and altitudes. 4. **Analysis of Aerodynamic Forces**: The CFD results are analyzed to quantify aerodynamic forces acting on the vehicles during the separation process, including drag, lift, and side forces. These forces are compared against design limits to ensure structural integrity. 5. **Trajectory Prediction**: The CFD simulations are used to predict the trajectory of the separating stages, taking into account aerodynamic forces, gravity, and other external factors. Trajectory deviations and stability are assessed to optimize separation parameters. 6. **Optimization**: Parametric studies are conducted to explore the effects of various design parameters, such as stage separation angles, separation mechanisms, and aerodynamic configurations.	14	CO5	Cre

		Optimization algorithms may be employed to find the optimal combination of parameters that minimize aerodynamic loads and ensure stable separation.			

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