# COMPUTATIONAL FLUID DYNAMICS
## (MECH 3221)

**Time Allotted : 2½ hrs** **Full Marks : 60**

***Figures out of the right margin indicate full marks.***

***Candidates are required to answer Group A and any 4 (four) from Group B to E, taking one from each group.***

***Candidates are required to give answer in their own words as far as practicable.***

## Group – A

1. Answer any twelve: **12 × 1 = 12**

*Choose the correct alternative for the following*

(i) A control volume is defined as a __________ region in space.
(a) Fixed (b) Moving (c) Deformable (d) Expanding

(ii) Which of the following is true for an unsteady flow?
(a) Velocity field does not change with time
(b) Pressure field is independent of time
(c) Flow properties change with time at a given point
(d) Flow properties remain constant with respect to time at a given point.

(iii) The Peclet number (Pe) is defined as the ratio of
(a) Diffusive transport to convective transport
(b) Convective transport to diffusive transport
(c) Momentum flux to energy flux
(d) Inertia force to viscous force

(iv) The diffusion term in the Finite volume method is discretized using the
(a) Upwind differencing scheme (b) Central differencing scheme
(c) SIMPLE algorithm (d) None of the above

(v) The Finite volume method ensures conservation of mass, momentum, and energy because
(a) It directly solves the Navier-Stokes equations
(b) It integrates the governing equations over a control volume
(c) It uses the central differencing scheme for convection
(d) All of the above

(vi) The under-relaxation factors used in the SIMPLE algorithm
(a) Ensure convergence of the iterative solution
(b) Reduce computational cost
(c) Improve accuracy of the pressure field
(d) Directly solve the momentum equation

(vii) The Tri-Diagonal Matrix Algorithm (TDMA) is also known as
(a) Newton-Raphson method (b) Gauss-Seidel method
(c) Thomas Algorithm (d) Jacobi method

(viii) In a CFD simulation, what is the main role of specifying initial conditions?
(a) To provide boundary conditions at the inlet and outlet
(b) To define the fluid properties
(c) To give the starting guess for the solver's iterative process
(d) To set up the computational grid.

(ix) Which of the following is not a typical input for boundary conditions in a CFD simulation?
(a) Velocity (b) Pressure (c) Mesh size (d) Temperature

(x) Which of the following tasks is not performed by the preprocessor in CFD software?
(a) Grid generation
(b) Boundary condition specification
(c) Numerical solution of governing equations
(d) Selection of fluid properties

*Fill in the blanks with the correct word*

(xi) The continuity equation is derived from which fundamental principle? ___________.

(xii) The substantial derivative consists of two terms: the __________ and the __________.

(xiii) The staggered grid method is commonly used in CFD to prevent the occurrence of _________ oscillations in the velocity field.

(xiv) The Tri-Diagonal Matrix Algorithm (TDMA) is an efficient method used for solving _________ systems of equations.

(xv) The finite volume method converts a differential equation into a set of _________ equations.

## Group - B

2. (a) What is the general form of the transport equation in fluid mechanics? What physical phenomena does it model? *[(CO2)(Explain/LOCQ)]*
(b) Discuss three important assumptions and limitations of the standard navier-Stokes equations for fluid motion. *[(CO1)(Identify/LOCQ)]*

**(4 + 2) + 6 = 12**

3. (a) Calculate the value of substantial derivative $\left(\frac{D\vec{V}}{Dt}\right)_{t=2s}$ of the velocity field represented by $\vec{V} = (-2xt)\hat{\imath} + (-3y)\hat{\jmath} + (4zt^2)\hat{k}\left(\frac{m}{s}\right)$ at a point P (1, -3, 4). Write down the values of the temporal and convective components of acceleration. *[(CO1)(Solve/IOCQ)]*

(b) How is the 'source term' in the general transport equation typically handled in CFD simulations? *[(CO2)(Implement/IOCQ)]*

**(5 + 2 + 2) + 3 = 12**

## Group - C

4. (a) In a CFD simulation, how is momentum conservation typically handled in an incompressible flow problem using the finite volume method? *[(CO3)(Implement/IOCQ)]*

(b) Explain the significance of boundary conditions when analyzing a fluid flow using the control volume approach. Provide an example of a boundary condition. *[(CO3)(Implement/IOCQ)]*

(c) What challenges arise when solving an initial value problem for nonlinear fluid flow? *[(CO3)(Implement/IOCQ)]*

**4 + 4 + 4 = 12**

5. A property Ø per unit mass is transported by convection and diffusion through an 1D domain. The governing equation is $\frac{d}{dx}(\rho u\emptyset) = \frac{d}{dx}\left(D_{\emptyset}\frac{d\emptyset}{dx}\right)$, and the boundary conditions are $\emptyset_{x=0} = 1$; $\emptyset_{x=L} = 0$. Consider L = 1.0 m. Using five equally spaced cells ($\delta x = 0.2\ m$) and the central differencing scheme for both convection and diffusion, formulate the linear algebraic equations for intermediate values of Ø. The following data apply: density $\rho = 1 kg/m^3$, diffusion coefficient $D_{\emptyset} = 0.1$ kg/ms, and velocity u = 0.1m/s. *[(CO4)(Solve/IOCQ)]*

**12**

## Group - D

6. (a) State the concept of 'staggered grid' in CFD. What is the main advantage of using a staggered grid in solution of fluid flow problems? *[(CO5)(Understand/LOCQ)]*

(b) State the concept of 'pressure-velocity coupling' in CFD analysis. Why is 'pressure-velocity coupling an important issue in solution of incompressible flow problems? *[(CO5)(Understand/LOCQ)]*

**(3 + 3) + (3 + 3) = 12**

7. (a) How does the SIMPLE algorithm solve the 'pressure-velocity coupling' problem? Mention the main steps involved in the SIMPLE algorithm. *[(CO5)(Solve/IOCQ)]*

(b) List five limitations of the Tri-diagonal Matrix Algorithm (TDMA) when applied to solving larger or non-linear systems. *[(CO5)(Explain/LOCQ)]*

**(2 + 5) + 5 = 12**

## Group - E

8. (a) Take into account a rectangular pipe of specified width, height and axial length, and water is flowing internally through the pipe. A constant heat flux is applied at the bottom wall by flush-mounted heaters. Regard all other portions of the pipe walls as adiabatic. Outline the procedural steps to solve this problem using a Computational Fluid Dynamics (CFD) software, considering laminar flow regimes. *[(CO6)(Analyze/IOCQ)]*

(b) For the above problem, write down the steps needed to obtain the bottom wall-averaged Nusselt number and the Friction factor. *[(CO6)(Apply/IOCQ)]*

**8 + 4 = 12**

9. (a) Take into account a rectangular 2D cavity, where the left and right walls are cold and the top wall is adiabatic. The bottom wall is symmetrically heated with constant heat flux to one-quarter of the total span. The non-heated portions of the bottom wall is also adiabatic. List the selection procedure of the flow regime. Outline the procedural steps for solving this problem using a Computational Fluid Dynamics (CFD) software, assuming laminar flow regimes. *[(CO6)(Analyze/IOCQ)]*

(b) For the above problem, write down the steps needed to obtain the average Nusselt Number for the partially-heated portion of the bottom wall. *[(CO6)(Apply/IOCQ)]*

**(3 + 5) + 4 = 12**

---

| Cognition Level | LOCQ | IOCQ | HOCQ |
|---|---|---|---|
| Percentage distribution | 30.21 | 69.79 | 0 |