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

Question Paper Code: 1017237

B.E. / B.Tech. DEGREE EXAMINATIONS, NOV/ DEC 2024
Seventh Semester
Aeronautical Engineering
U20AE702 - COMPUTATIONAL FLUID DYNAMICS
(Regulation 2020)

Time: Three Hours Maximum: 100 Marks

Answer ALL questions

PART – A (10 x 2 = 20 Marks)

1. How does mathematical modeling of fluid flow benefit engineers?

2. Write the complete Navier–Stokes equations in the conservation form.

3. What are the limitations of structured grid in computational simulations?

4. State grid control functions.

5. Compare the between implicit and explicit solution methods in CFD?

6. What is a numerical dissipation error?

7. What do you mean by upwind scheme?

8. State applications of finite volume method .

9. How do you validate CFD solutions?

10. How do you chose the right turbulence model for the given CFD problem?

PART – B (5 x 16 = 80 Marks)

11. (a) Distinguish between elliptic, parabolic and hyperbolic system of PDEs applied to fluid flows. What are the types of boundary condition specific for flow problems? (16)

(OR)

(b) Derive the generalized transport equation involving the basic principles of fluid mechanics. (16)

12. (a) Explain the different grid generation methods commonly used in CFD solutions and describe their respective applications. (16)

(OR)

(b) Explain the grid control functions in Computational Fluid Dynamics. (16)

13. (a) When are explicit time-dependent methods preferred? Explain one such method with a suitable example. (16)

(OR)

(b) Explain the necessity of implicit methods in CFD solutions and the associated stability issues. (16)

14. (a) Explain the different flux evaluation schemes used in FVM. (16)

(OR)

(b) Explain the concept of SIMPLE algorithm with pressure correction and velocity correction equations for incompressible fluid and give the importance of SIMPLER algorithm. (16)

15. (a) Why boundary conditions are needed? Explain common thermal and flow boundary conditions used in CFD with suitable examples. (16)

(OR)

(b) Explain the steps involved in analyzing a simple fluid flow problem using commercial CFD simulation software. (16)

---------xxxx--------