VII Sem ME

Computational Fluid Dynamics

Assignment III

Unit - III

Due Date:

Note:

1. *Assignment should be hand-written on unrule sheet (A4 Size plain papers)*
2. *Write Questions (in black ink) and Answers (in blue). Figures, wherever required, should be drawn using pencil only.*
3. *Write Your Name, E.No, Course Name and Title, Date of Submission on Cover Page of Assignment. Cover Page can be typed. (see attached sample)*
4. *All pages of assignment should be properly stapled. You may also use a plastic folder.*
5. *Maximum marks for the entire assignment is 10.*
6. *No Marks will be awarded for incomplete / incorrect answers.Marks shall be deducted for illegible / untidy /rough Work.*
   Due weightage will be given for ***Neat work, good Handwriting and clear & labelled figures****.*
7. *Last Date of Submission is _____*

---

1. Describe the numerical approach using the finite difference method to solve for the temperature distribution in a one-dimensional composite wall with specified boundary temperatures.

2. Outline the steps involved in setting up a numerical model to determine the steady-state temperature profile in a cylindrical fin.

3. Explain how boundary conditions are incorporated when numerically solving for steady-state heat conduction in a spherical shell.

4. Explain the difference between explicit and implicit finite difference schemes for solving unsteady heat conduction problems.

5. Describe a numerical method to determine the transient temperature distribution in a two-dimensional plate subjected to a sudden change in boundary temperature.

6. How would you account for a heat generation term within a solid when numerically modelling unsteady-state heat conduction?

7. In a CFD simulation of flow through a heated pipe, how would you use empirical correlations to determine the convective heat transfer coefficient at the pipe wall?

8. Describe how you would select an appropriate Nusselt number correlation for external flow over a cylinder in a CFD simulation.

9. Explain the importance of dimensionless numbers like Reynolds and Prandtl numbers when applying convection heat transfer correlations in CFD.

10. Describe the post-processing steps in a CFD software (e.g., ANSYS) to visualise the temperature distribution in a simulated heat transfer problem.

11. Explain how you would use CFD software to calculate the total heat transfer rate from a heated surface to a flowing fluid.

12. Outline the procedure to extract temperature values at specific locations within the computational domain after a CFD simulation of a conduction problem.

13. Discuss how changing the boundary conditions (e.g., constant temperature vs. constant heat flux) affects the temperature distribution in a CFD simulation of heat conduction.

14. Explain how variations in fluid properties like thermal conductivity and viscosity impact the convective heat transfer predicted by a CFD simulation.

15. Describe how you would use CFD to investigate the effect of different inlet flow velocities on the heat transfer from a heated object.

Assignment I

of

Computational Fluid Dynamics

Paper Code: CADM-401P

VII Semester (BTech-ME)

Name of Student :

Enrollment Number :

Date of Submission :

Submitted on :

Grades :

Signature of Faculty :