● Innovation Project

Essential/recommended readings

- Theory of modeling and simulation, Zeigler B.P., Praehofer. H., Kim I. G., 2nd Edition. Academic press, 2000.
- Theory of Fractional Dynamic Systems, Lakshmikantham, V., Leela, S., Vasundhara Devi,
  J. Cambridge Academic Publishers, Cambridge, 2009.
- Fractional-order nonlinear systems: modeling, analysis and simulation, Petras, I., SpringerVerlag Berlin Heidelberg, Germany, 2011.
- Chaos: An Introduction to Dynamical Systems, K.T. Alligood, Sauer, Tim D., Yorke James
  Springer, 1996.
- Nonlinear Dynamics and Chaos, Strogatz, S. Reading, MA: Addison-Wesley, 1994.
- Optimization and Dynamical Systems, Helmke U., Moore J. B, SpringerVerlag, 1993.

DISCIPLINE SPECIFIC ELECTIVE COURSE -4 (DSE-4)
VI.5.2. Computational Fluid Dynamics (CFD)

CREDIT DISTRIBUTION, ELIGIBILITY AND PRE-REQUISITES OF THE COURSE

Course title & Code	Credits	Credit distribution of the course			Eligibility criteria	Pre-requisite of the course (if any)
		Lecture	Tutorial	Practical/ Practice		
Computational Fluid Dynamics# (CFD), DSE-4, VI.5.2	**4**	**0**	**0**	**4**	**12th Pass With Maths**	**Calculus, Linear Algebra & Differential Equations**

#This course will also be available to the students in semester IV

Learning Objectives

This interactive practical paper aims to enable the students to visualize different types of problems of flow and heat transfer in various fields. Blood flow within arteries, biological tissues, heat transfer within biological tissues, flow within circular pipes, flow within an aquifer are some of the important application of CFD. In this paper, students will visualize CFD models, mathematical analysis of these visualizations, simulate them numerically using mathematical softwares such as ANSYS, COMSOL and post process the obtained numerical results.

Learning Outcome

After completing this paper, students will be able;

- Visualise and implement mathematical models of flow and heat transfer problems in different applications.
- Implement existing CFD based modules in ANSYS/COMSOL for simulation purpose.
- Design of UDF based problem specific modules in ANSYS/COMSOL
- Validate their numerical results with experimental data (if available) for suggesting new designs.

Syllabus

Practicals - (120 Hours)

- Governing equations for CFD: The continuity, momentum and energy equations with their physical interpretation
- Interpretation of different set of flow conditions such as inflow, outflow, no slip boundary etc as per problem requirement.
- CFD mesh generations, structured and unstructured mesh, mesh refinement (local & global), adaptive mesh.
- Implementation of existing CFD based modules in ANSYS/COMSOL for simulation purpose.
- Design and modification of existing modules using UDF (User defined functions) in ANSYS/COMSOL as per the problem requirement.
- Post-processing of numerically simulated results

Essential/ Recommended Readings:

- Computational Fluid Dynamics: A practical Approach (2019) by Jiyuan Tu, Guan Yeoh, Chaoqun Liu, 2nd Edition, Publisher: Butterworth-Heinemann.
- Computational Fluid and Particle Dynamics in the Human Respiratory system (2012) by Jiyuan Tu, Kiao Inthavong, Goodarz Ahmadi, Biological and Medical Physics, Bio-medical Engineering, Publisher: Springer.
- Multiphysics Modelling using COMSOL: A First Principle Approach (2011), by Roger W. Pryor, Jones and Bartlett Publishers, London, Singapore.