Course Plan (Spring-2025)

School: Engineering

Department: Mechanical Engineering

Course Code: ME316

Course Name: Computer-Aided Engineering

Course type: Core

Credit Structure: L-T-P-Cr-CH: 1-0-2-3-5

Instructor: Dr. Sushen Kirtania and Dr. Shikha Bhuyan

1. Abstract: Computation plays an important role in engineering. Therefore, computational knowledge is an essential requirement for Mechanical Engineering students. Many of the engineering problems cannot be solved without the aid of computer. The aim of this course is to make the B. Tech. students learn different modelling and analysis software along with the numerical solution techniques of some engineering problems with the help of computer programming. Emphasis is on learning the applications of different software packages and programming languages through hands-on sessions. In the teaching-learning process, basic knowledge of advanced subjects like finite element method (FEM) and computational fluid dynamics (CFD) will be covered. Finally, assignments will be given for submission.

2. Course objectives: The main objectives of this course are –

(i) To impart knowledge to the students related to solid modeling and system modeling of thermo-fluid systems.

(ii) To train the students on modern techniques for solving real-life-engineering problems using commercial-software packages.

(iii) To train the students for writing and presenting a technical report by giving assignments and mini projects.

(iv) To encourage students for higher education.

3. Prerequisites of the course: None.

4. Course outline: Introduction to computer-aided modelling, Introduction to structural analysis, Introduction to CFD, Modelling of fluid systems, and use of commercial fluid-flow solvers to solve different problems.

5(a). Time plan:

Sl.	Topic	Contents	L	P
1.	Introduction to computer-aided modelling	Basic drafting, Modelling of parts and assembly drawing using standard software packages.	2	6
2.	Introduction to structural analysis	Introduction to finite element analysis (FEA): Basic engineering analysis of Beams, Trusses, Plates; Stress analysis of structure with individual and combined loading under Mechanical, Thermal and Thermo-Mechanical loading.	5	5
3.	Introduction to CFD	Mathematical nature the governing partial differential equations (PDEs) for fluid flow and heat transfer, Introduction to finite difference method (FDM) and finite volume method (FVM),	5	2

		Preprocessor, Solver and postprocessor of a commercial CFD package, SIMPLE algorithm, RANS based turbulence models, Shear stress transport model, Near wall treatment		
4.	Modelling of fluid systems	Geometry modelling using a standard commercial package, Specification of boundary conditions, Free-stream conditions and flow properties, User-defined functions.	2	2
5.	Use of commercial fluid-flow solver (ANSYS Fluent and programming package) to solve the following problems	(i) Lid-driven cavity flow problem (ii) Natural and mixed convection in a lid-driven cavity (iii) Forced convective flow and heat transfer in a channel for laminar and turbulent flow conditions (iv) Unsteady flow through a channel with pulsatile inlet velocity profile (v) Steady and unsteady-state temperature profiles in solids under heat conduction (vi) Conjugate heat transfer in a duct with constant heat flux and axially varying heat flux (vii) Solution of viscous, laminar, incompressible flow over immersed bodies – airfoils.	-	13

Total number of classes: (14+28) = 42

5(b). Evaluation plan:

There will be one End-Semester (Practical) Examination of 50 marks. The components of this examination are Practical Examination (20 marks), Viva (10 marks) and Report submission (10 mark) and Assignments (10 marks).

6. Pedagogy: Lecture and discussion, Practical sessions, Assignments, Viva, and Report.

7. Course outcomes: Towards the end of the course the student would be able to

CO1: Model physical systems involving structural and thermal applications.

CO2: Use commercial-software packages to simulate engineering problems involving structural loading, fluid flow and heat transfer.

CO3: Write and present their assignments as a technical report or document.

CO4: Pursue research in the field of FEM and CFD.

Textbooks:

1. Dixit US. Finite Element Methods for Engineers, Cengage Learning, 2009.

2. Versteeg, H. K. and Malalasekera, W., An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Pearson, 2nd Edition, 2009.

3. Anderson, D. A., Tannehill, J. C., and Pletcher, R. H., Computational Fluid Mechanics and Heat Transfer, CRC Press, 3rd Edition, 2012.

References:

1. Bhat, N. D. and Panchal V. M., Machine Drawing, Charotar Publishing House, Court Road, Anand, India, 48th Edition, 2013.

2. Srinivas, P., K., Sambana, C. and Datti, R. J., Finite Element Analysis using ANSYS@ 11.0, PHI, New Delhi, 2012.

3. Munford, P. and Normand, P. Mastering Autodesk Inventor 2016 and Autodesk Inventor LT, John Wiley Sons, 2016.

4. Kent, L.L. ANSYS Workbench Tutorial Release 14, SDC Publications, 2012.

5. ANSYS FLUENT Tutorial guide Release 15.0, ANSYS Inc., 2013.