COURSE TITLE	**PARAMETRIC ANALYSIS OF FLUID FLOW BEHAVIOUR USING ANSYS-FLUENT**
COURSE CODE	**01CH0718**
COURSE CREDITS	**2**

Objective:

1 The objective of this course is to make students familiar with CFD software and how to Analysis of Fluid Flow Behaviour of model using Ansys Fluent through hands-on sessions.

Course Outcomes: After completion of this course, student will be able to:

1 Understand basic design concepts, use of Standard design model.

2 Apply and analyze knowledge to practical design concepts used for modelling using Ansys Fluent.

3 Evaluate the design modelling criteria of process equipment.

4 Create the actual design of a model and drawing of process industry equipment.

Pre-requisite of course: Knowledge of Fluid Mechanics and Heat Transfer

Teaching and Examination Scheme

Theory Hours	Tutorial Hours	Practical Hours	ESE	IA	CSE	Viva	Term Work
0	2	0	0	0	0	50	50

Contents : Unit	Topics	Contact Hours
Total Hours		

Suggested List of Experiments:

Contents : Unit	Topics	Contact Hours
1	**Tutorial 1** Introduction: What is CFD?, Applications of CFD, The Mathematics of CFD	3
2	**Tutorial 2** Introduction to ANSYS Fluent	3
3	**Tutorial 3** Creating the geometry and mesh in the ANSYS design modular.	3
4	**Tutorial 4** Creating the Mesh, setting parameter and Result analysis in ANSYS Fluent.	3
5	**Tutorial 5** To simulate Fluid Flow in a Circular pipe	3

DR. DHARMESH HARESHKUMAR SUR
Digitally signed by (Name of HOD)

DR. RAJENDRASINH BAHADURSINH JADEJA
Digitally signed by (Name of Dean/ Principal)

Suggested List of Experiments:

Contents : Unit	Topics	Contact Hours
6	**Tutorial 6** To simulate Fluid Flow in a converging pipe	3
7	**Tutorial 7** To simulate Fluid Flow in a diverging pipe	3
8	**Tutorial 8** To study the thermal conduction through a composite wall.	3
9	**Tutorial 9** To simulate Heat Transfer in a Double pipe heat exchanger.	4
10	**Tutorial 10** Natural Convection Heat Transfer Analysis in Square Cavity	4
Total Hours		**32**

Textbook :

1 An Introduction to ANSYS Fluent , John E. Matsson., SDC publication., 2022

References:

1 Fluid Mechanics for Chemical Engineers: with Microfluidics, CFD, and COMSOL Multiphysics, Fluid Mechanics for Chemical Engineers: with Microfluidics, CFD, and COMSOL Multiphysics, James O. Wilkes,, Prentice Hall publication, 2017

Suggested Theory Distribution:

The suggested theory distribution as per Bloom's taxonomy is as follows. This distribution serves as guidelines for teachers and students to achieve effective teaching-learning process

Distribution of Theory for course delivery					
Remember / Knowledge	**Understand**	**Apply**	**Analyze**	**Evaluate**	**Higher order Thinking / Creative**
10.00	20.00	30.00	30.00	10.00	0.00

Instructional Method:

1 The course delivery method will depend upon the requirement of content and need of students. The teacher in addition to conventional teaching method by black board,may also use any of CFD Soft ware such as Ansys Fluent,COMSOL Multi physics etc.

2 The internal evaluation will be done on the basis of continuous evaluation of students in the Software laboratory.

3 Practical examination will be conducted at the end of semester for evaluation of performance of students in laboratory.

4 Students will use supplementary resources such as online videos, NPTELvideos,e-courses, Virtual Laboratory.

DR. DHARMESH HARESHKUMAR SUR
Digitally signed by (Name of HOD)

DR. RAJENDRASINH BAHADURSINH JADEJA
Digitally signed by (Name of Dean/ Principal)

Supplementary Resources:

1 https://ansyshelp.ansys.com/public/account/secured?returnurl=/Views/Secured/corp/v251/en/flu_tg/flu_tg_parametric_study.html