Course Structure & Syllabus
CFD – Domain

Code	Subject	Course Type	Credit
CUCF2180	Introduction to CFD	T-P-Pj(2+0+1)	3
CUCF2181	Grid Generation	T-P-Pj(0+2+0)	2
CUCF2182	Flow Solver Techniques-Simulia	T-P-Pj(0+3+1)	4
CUCF2183	Simulation and Validation	T-P-Pj(0+5+0)	5
CUCF2184	Industry Specific Project	T-P-Pj(0+0+6)	6
	TOTAL CREDITS	T-P-Pj(2+10+8)	20

Domain Track Title: Computational Fluid Dynamics

Track Total Credits-20 (2+10+8)

Course Division (List all divisions):

1. CUCF2180 Introduction to CFD (2+0+1)
2. CUCF2181 Grid Generation (0+2+0)
3. CUCF2182 Flow Solver Techniques-Simulia (0+3+1)
4. CUCF2183 Simulation and Validation (0+5+0)
5. CUCF2184 Industry Specific Project (0+0+6)

Domain Track Objectives:

- To provide the knowledge of CFD in the Industrial Level.
- To apply CFD methods as a tool for design, analysis and engineering applications.

Domain Track Course outcomes:

- To impart knowledge about various computational methods for fluid flow and heat transfer problems so as to enable the students to write computer programs for solving elementary fluid dynamics/heat transfer problems.
- To execute the industry projects so as to produce Quality products to the Clients.

Domain Syllabus:

1. Introduction to CFD (2-0-1)

Module-1

Fluid, Properties of Fluid (Temperature, Vapour Pressure, Viscosity, Specific Gravity, Compressibility, Speed of Sound, Density, Energy, Specific Heat, Newtonian and Non-Newtonian Fluids

Module-2

Lagrangian and Eulerian Approaches, Classification of Fluid Flow (Inviscid vs. Viscous, Laminar vs. Turbulent, Incompressible vs. Compressible, Internal vs. External, Steady vs. Unsteady, Rotational vs. Irrotational, 1D, 2D and 3D Flows)

Project 1. Case studies in industrial pipe flows.

Module-3

Governing Equations of Flow Field (Conservation of Mass, Momentum (Navier-Stoke Equation) and Energy)- Differential Form, Integral Form.

Project 2. Case studies and generation of drag and lift for flow over bodies.

Module-4

Flow Features (Stagnation, Boundary Layer (Laminar to Transition to Turbulent), Flow Separation), Drag & Lift forces, Types of Boundary Conditions.

Project 3. Case studies in different types of aerofoil and its applications.

Module-5

Heat Transfer in Fluid (Conduction, Convection (Natural, Forced), Radiation), Non-dimensional Quantities, Grid Generation, Flow Similarity between Prototype and Model.

Project 4. Case studies for various non-dimensional quantities and its importance in CFD.

Module-6

CFD and its applications, Discretization methods for the CFD (FDM, FVM, FEM, Hybrid Methods).

Project 5. Case studies for the various applications of CFD and its significance.

Module-7

Turbulence Modelling, CFD Solution Tool Chain

Project 6. Case studies for the different turbulence modelling in CFD.

2. Grid Generation (0-2-0)

2.1 Introduction to Meshes- CFD Meshing Basics

2.2 Different Element Shapes- Creating, Managing & Updating Meshes

2.3 CFD Meshing- 2D Mesh, 3D Mesh, Mapped Face Meshing,

Practice 1. 2D mapped Mesh for rectangular pipe

Practice2. 2D mapped Meshing for Aerofoil.

2.4 Structured Meshing, Un-Structured Mesh, Sweep 3D Mesh

Practice 3. 3D structure mesh of Circular Cylinder

Practice 4. 3D unstructured mesh with prims layers for Aerofoil

Practice 5. 3D coarse/ medium/ fine sweep mesh for pipe

Practice 6.3D coarse/ medium/ fine unstructured Octree Tetrahedron mesh for Aerofoil.

Practice 7. 3D hex- dominant mesh for rectangular Duct.

2.5 Visualization the Mesh- Visualization Management, Mesh Visualization Options, Section, Clipping Box, Mesh colour, Element shrink

2.6 Reviewing the Mesh- Quality Analysis, free Edges, Interfaces, Duplicate Checker, Isolate Node Checker

Practice 8. 3D hex-dominant with surface mesh, Boundary prim mesh for DS car.

Practice 9. 3D Tetrahedron filler mesh Narrowing pipe.

Practice 10. 3D Tetrahedron, surface, for Electronics Module.

Practice 11. 3D Sweep mesh for circular cylinder.

Practice 12. 3D mesh generation for Subsonic Converging-diverging Nozzel.

Practice 13. 3D Sweep mesh generation for U-Bend pipe.

Practice 14. 3D mesh generation of Dimple Ball.

Practice 15. 3D mesh generation of a wedge body.

3. Flow Solver Techniques-SIMULIA (0-2-1)

3.1 Overview-Fluid Dynamics Engineer Essentials

- Connecting to the platform, Assigning roles and Apps, Platform Interface, Importing 3D XML file, Simulation Conventions in the 3DExperience Platform

Practice 1. Getting Started with the 3DExperience Platform.

3.2 Import an assembly

- Explore the imported assembly, Renamed the assembly, Search for parts/assemblies in the database, open parts/ assemblies found through search, duplicate, delete and save entities, Import a 3D XML file containing fluid materials, Create and save a new material.

3.3 Fluid Dynamics Engineer Role Overview

- Exploring Fluid Dynamics Engineer Role Apps, CFD simulations work flow, Model preparations, Material definitions, Meshing, CFD analysis, Analysis convergence, co-simulation Analysis, Post processing results, CFD solver validations.

Practice 2. CFD analysis of steady state internal Laminar Pipe flow.

Project 1. Analysis of pipe flow at Re= 500.

3.4 Getting Started with CFD Simulations

- Fluid Model Creation, Fluid Scenario Creation App Interface, Model setup, applying meshing, Scenario Setup, Results visualizations, Reviewing Simulation Features,

Practice 3. CFD analysis Steady-state external flow over an Airfoil.

Project 2. Analysis of 2D cylinder in a rectangular domain with varying radius and height.

3.5 Geometry for CFD Simulations

- Geometry Preparation- Check and Repair, Defeature Idealize, Create, Healing, Join, Local Join, surface connection Checker, Face checker, Mid surface

Practice 4. Modeling of Air intake system.

Practice 5. Extracting Fluid volume for Engine Manifold.

3.6 Material and Section Properties of Fluid

- Understanding materials, working with materials, Creating a new Material, Applying a Material, Adding New Domains, Editing a Material Domain, Simulation Domain, Material Behaviors in a Simulation Domain, Section Properties, working with Imported Meshes

3.7 Defining Physics of Fluid

- Analysis Procedures- Enabling Temperature, Compressible, Coupled vs Segregated solver, Gravity effects.
- Turbulence Modelling- SST k-w, Realizable k-e, Spalart-Almaras, Radiation Modeling, Steady-state Analysis, Transient Analysis- Courant-Freidrichs-Levy (CFL) condition, Grid Independence Study, Bad cell Treatment

Practice 6. Grid Independence study for above cases (pipe / airfoil) using different solver schemes.

Project 3. Analysis of flow an over a circular cylinder at $Re=10^7$.

Practice 7. Conjugate Heat Transfer (CHT) Analysis of an Electronics Module.

Project 4. Analysis of temperature rise through cross flow heat exchanger.

3.8 Boundary and Initial Conditions

- Boundary Conditions, wall boundary conditions, Thermal wall boundary, Initial Conditions, Initializing Compressible flows, Turbulence Specifications at Boundaries, Surface- to – surface Radiation specification at Boundaries, Time- dependent Boundary Conditions, Spatially- Varying Boundary conditions, User defined Boundary conditions.

3.9 Turbulence Modeling & Modeling Techniques.

Practice 8. Aerodynamics analysis of DS car.

Practice 9. Unsteady Flow across a Circular Cylinder.

Practice 10. Transonic Flow over an Airfoil.

Project 5. Analysis and estimation of Drag lift coefficients flat plate at Re=10,000.

3.10 Solution Convergence

Practice 11. Cavitating Flow through a Narrowing Pipe.

Project 6. Analysis of compressible flow nozzle with atmospheric pressure at the nozzle exit.

3.11 Post-processing Results.

Practice 12. Creating Post processing reports for all the above cases.

4. Simulation and Validation (0-4-0)

4.1 Fluid flow in the rear duct of an automotive HVAC system.

4.2 CFD Analysis of an Air intake system.

4.3 CFD Steady-state External flow over a Drone in cruise.

4.4 DE featuring of a Lens Component.

4.5 CFD analysis for Conjugate Heat Transfer in a fan –cooled CPU Board.

4.6 CFD analysis Energy computations in a Contact Analysis.

4.7 Thermo-mechanical Analysis of a Laser Powder Bed Fusion Build.

4.8 CFD analysis in Turbulent pipe flow.

4.9 CFD Supersonic flow analysis for 3D cone.

4.10 CFD analysis over a Ahmed body.

5. Industry Specific Projects (0-0-6)

(Selected Projects will be carried out)

5.1 CFD Analysis of economizer in a tangential fired boiler.

5.2 Analysis and comparison of vertical tube with smooth tube.

5.3 A CFD-based analysis of the 14-bis aircraft aerodynamics and stability.

5.4 CFD analysis of gas flow behaviour in economizer duct.

5.5 Combined aerodynamic and structural optimization of a high-speed civil transport wing.

5.6 Fluid flow and temperature distribution in radiators used in automobiles.

5.7 Analysis of Cyclone dust collector air flow.

5.8 CFD analysis of shell and tube heat exchanger with fins for waste heat recovery application.

5.9 A theoretical analysis and CFD simulation on the on the ceramic monolith heat exchanger.

5.10 Analysis of water flow for Laminar & Turbulent Flow in Conventional Water Tap.

5.11 CFD Investigation of Airflow on a SANTRO Zing Car by using Fluent.

5.12 CFD analysis of rocket nozzle.

5.13 CFD analysis of supersonic exhaust in a scramjet engine.

5.14 Aerodynamic Design for Bus/Car Vehicle.

5.15 CFD analysis of exhaust manifold.

5.16 CFD analysis of centrifugal fan.

5.17 Analysis of intake manifold in SI engines.

5.18 CFD modelling of the automobile catalytic converter.

5.19 CFD analysis of fluid flow and heat transfer in a single tube-fin arrangement of an automotive radiator.

5.20 Computational flow field analysis of a vertical axis wind turbine.