COURSES OFFERED

- **Introduction to CFD using ANSYS Fluent**
- **MICROSOFT EXCEL**
- **Programmable Logic Controller**

DEPARTMENT OF MECHANICAL ENGINEERNG

VALUE ADDED COURSES

Seventh Semester

Introduction to CFD using ANSYS Fluent

Seventh Semester - Mechanical Engineering - 2018 Admission

Faculty Information

Instructor	Contact Information	Area of Interest
Mr.Renjith R	**renjithr@fisat.ac.in** **Mob:9496563100** **Linkedin: linkedin.com/in/renjith-r-a929a2143** **Research gate: https://www.researchgate.net/profile/Renith-Ravindranathan**	Finite Element Analysis, Computational Fluid Dynamics, Composite Materials, Open Source Numerical Modeling, Robotics and 3D printing

General Information

Course Objectives

a. Students should perform essential steps in setting up a CFD analysis

b. Students should Create high quality CFD Simulation meshes from imported CAD geometry

c. Perform all steps of a CFD simulation from CAD import to meshing to solution to results

Course Outcomes

Upon completion of the course, student will be able to effectively use Excel to

- Prepare CAD models for use in Ansys Fluent for CFD analysis
- Setup high quality meshes using fluent
- Do an introductory CFD analysis using ANSYS Fluent
- Use the parametric workflow with Ansys Fluent to execute parametric studies of multiple CFD design points.
- Post process and judge results from CFD analysis
- **Verify and validate CFD model with experimental analysisCourse Materials**

Reference Books and Tutorials

- Anderson, J. D., & Wendt, J. (1995). Computational fluid dynamics (Vol. 206, p. 332). New York: McGraw-Hill.
- https://cfd.ninja
- https://forum.ansys.com/

Course Contents

Module	Topic	Exercises
Module 1 **(5 Hours)**	Introduction to Computational Fluid Dynamics and Principles of Conservation, What, When, and Why?, CFD Applications, Numerical vs Analytical vs Experimental, Modeling vs Experimentation, Fundamental principles of conservation, Reynolds transport theorem, Conservation of mass, Conservation of linear momentum: Navier-Stokes equation, Conservation of Energy, General scalar transport equation	Exercises: Analytical simulation of simple flow problems, Comparison with CFD analysis outputs, Experimental data collection and verification
Module 2 **(7 Hours)**	The basics of CFD using ANSYS Fluent , Steps involved in a CFD , Meshing of water tight bodies in ANSYS, External file formats for Fluent, Familiarization with fluent mesher,Introduction to Fluent interface , Handling imported Geometries , Different meshing methods (local and Global). Introduction to Fluent workspace , How to uses ribben and tree , Introduction to set up models, Material properties • The different boundary condition types in Fluent, Define cell zone conditions in Fluent including solid zones and porous media, Simulation of well-posed boundary conditions in fluent	Exercises: ANSYS Fluent Guided Meshing Workflow for Watertight Geometry, Workshop on mixing Tee, Workshop on static mixer
Module 3 **(7 Hours)**	Flow field visualization and quantitative data analysis on CFD results , Producing contour plots and 2D graphs from results , Fluent meshing and connections with Workbench Geometry, Space claim and design modeler for Fluent, Parameters for a parametric study , Design points in fluent.	Exercises: Workshop on Flow Over Heated Obstacles and Parametric Study of Ball Valve, Preparation of Contour plots, 2 D graphs
Module 4 **(7 Hours)**	Solution, Different solution methods for CFD, Solution controls for faster convergence • Types of initialization in fluent, Report definitions to monitor and judge convergence, Four different types of errors , Strategies for minimizing error, Issues to consider during mesh creation (quality and cell type) , Best practices for mesh creation	Exercises: Workshop on Transonic Flow Over a NACA 0012 Airfoil, Practice sessions on Solution methods and Controls
Module 5 **(8 Hours)**	Turbulent flow determination, Introduction to turbulence model, Modeling flow near walls , Turbulence boundary conditions at	Exercises : Exercises on Vortex Shedding and Electronics Cooling with Natural

Module	Topic	Exercises
	inlets, Heat Transfer in CFD(conduction, convection (forced and natural) and radiation) , Wall thermal boundary conditions, Transient calculations in Fluent • Time step size and convergence , Post-process transient data and make animations	Convection and Radiation, Preparation of turbulence model, Post processing results and animations

Evaluation Schedule

Date	Details
	Assignments
	Mini Projects

Additional Information and Resources