ANSYS Fluent Syllabus

Total duration: 40 hours (Theory 20 Hours + Lab 20 Hours)

ANSYS Fluent Total Duration : 40 Hours	
Session	**Topics**
Session 1	**Chapter 1: Introduction**
	What is CFD?
	Applications of CFD & Uses of CFD
	The Mathematics of CFD
	Fundamentals of Fluid MechanicsEQUATION OF STATE
	CFD Methodology
	Introduction to ANSYS Fluent
	Planning Your CFD Analysis with Fluent
	Chapter 2: Graphical User Interface (GUI)
	Menu Bar & Toolbars
	The Navigation Pane
	Task Pages
	The Console
	Boundary Conditions
	Fluent in Workbench
Session 2	**Chapter 3: Solid Modeling Fundamentals**
	Problem Description
	Creating a Fluent Fluid Flow Analysis System in ANSYS Workbench
	Creating the Geometry in ANSYS DesignModeler
	Meshing the Geometry in the ANSYS Meshing Application
	Setting Up the CFD Simulation in ANSYS Fluent
	Displaying Results in ANSYS Fluent and CFD-Post
	Duplicating the Fluent-Based Fluid Flow Analysis System
	Changing the Geometry in ANSYS DesignModeler
	Updating the Mesh in the ANSYS Meshing Application
	Calculating a New Solution in ANSYS Fluent
	Comparing the Results of Both Systems in CFD-Post
	Chapter 4: Transonic Flow–Externally Compressible
	Problem Description
	Turbulence Models
	Preparation
	Mesh & General Settings
	Models & Materials
	Boundary Conditions
	Operating Conditions
	Solution & Post processing
Session 3	**Chapter 5: Multiple Species**
	Simulation Physics & Boundary Conditions
	Set Boundary Conditions
	Set Operating Conditions
	Set Solution Methods
	Set Solution Controls
	Start the Calculation
	Add a Results System

ANSYS Fluent Total Duration : 40 Hours	
Session	**Topics**
Session 3	**Chapter 6: Turbulence Model in Fluent**
	Problem Specification
	Preliminary Analysis
	Geometry
	Mesh
	Mesh Refinement
	Physics Setup
	Numerical Solution
	Numerical Results
	Verification & Validation
Session 4	**Chapter 7: Modeling Periodic Flow and Heat Transfer**
	Introduction
	Problem Description
	Mesh
	General Settings
	Models
	Materials
	Cell Zone Conditions
	Periodic Conditions
	Boundary Conditions
	Solution
	Post processing
	Chapter 8: Modeling Radiation and Natural Convection
	Introduction
	Problem Description
	Reading and Checking the Mesh
	Specifying Solver and Analysis Type
	Specifying the Models
	Defining the Materials
	Specifying Boundary Conditions
	Obtaining the Solution
	Post processing
	Comparing the Contour Plots after Varying Radiating Surfaces
	S2S Definition, Solution, and Postprocessing with Partial enclosure
Session 5	**Chapter 9: Turbulent Flow in a Compact Heat Exchanger**
	Introduction
	Prerequisites
	Problem Description
	Setup and Solution
	Chapter 10: Siphoning Model
	Introduction
	Procedure
	Define the Materials
	Define the phases
	Define Phase Interactions
	Problem Setup > Boundary Conditions
	Problem Setup > Boundary Conditions
	Copy Boundary conditions from outlet to ambient
	Set Solution Methods
	Set Solution Controls
	Initialize the Flow Field
	Generate Animation