Hands-on Internship Program on CFD Simulation with Ansys Fluent

Objective and Program need:

The objective of the programme is to acquire knowledge in Computational Fluid Dynamics (CFD) and to find solutions for Engineering problems involving Fluid flow using software tool.

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process (that is, on a computer). The applications of CFD are Aerospace, Appliances, Automotive, Biomedical, Chemical Processing, HVAC&R, Hydraulics, Marine, Oil & Gas, Power Generation, Sports, Electronic Cooling etc. Many of the B Tech, M Tech and PhD students are doing projects in this area from Mechanical Engineering, Civil Engineering and Chemical engineering. This training will provide better vision for faculties to improve the output of the projects.

Topics:

Day1 (08.05.2023):

Introduction to Ansys Workbench: Overview of ANSYS Workbench. Component and custom systems in workbench.

2D and 3D modeling: How to Import generic CAD formats (native & Neutral)? Body Types & States (Frozen & Active) in DM, Use of Multi-body parts and Share topology, How to Decompose geometry into mesh able sections.

Introduction to Ansys Meshing: Overview on Meshing methods & Mesh controls ANSYS Meshing graphics user interface.

Meshing methods Tetrahedrons and sweep, Global Mesh controls: Difference between Patch dependent and Patch conformal mesh, Global inflation in tetrahedral meshing, Different methods for Hex Meshing Hexa dominant, sweep meshing.

Day2 (09.05.2023)::

Introduction to CFD, Physics for CFD: Overview of CFD: Development, Application and Analysis, Essentials of Fluid Dynamics and Heat Transfer for CFD, Governing equations, Boundary conditions.

Introduction to ANSYS Fluent and Solver settings: Launching ANSYS Fluent: Workbench system, GUI, Workflow, Text User Interface.

Solver Settings: Solver types, Pressure-Based Solver Inputs, Under Relaxation factors, Pressure-Based Coupled Solver, Spatial Discretization.

Day3 (10.05.2023)::

Introduction to Multiphase and its modeling: Overview and basic concept of multiphase Defining boundary condition for multiphase flow How to simulate multiphase flow?

Methods: Discrete Phase Model(DPM), Volume of Fluids (VOF), Eulerian Model, Mixture model.

Day4 (11.05.2023)::

Workshop on Simulation of Water Hammer: This workshop teaches the different steps involved in: Steady state and transient flow modeling including compressibility effects, How to prepare and monitor file outputs.

Open channel flow: This workshop teaches to model open channel flow with different types of boundary conditions.

Simulation of catalytic converter: This workshop teaches to model the species transport.

Day5 (12.05.2023)::

Combustor modeling: This workshop teaches to model non-premixed combustion

Modeling of Super Sonic Nozzle: This practice helps to get information about Compressible Fluid Flow