Organize a
Student Development program
On
Heat Transfer and Fluid Flow simulation Using Ansys Fluent
Date: 22 June-3 July 2020

Course Coordinator: Prof. Chandan D. Chaudhari
                              Dr. Kaustubh V. Chavan
                              Prof. Ajay S. Hundiwale

Prerequisite: Basics of FEA

Objectives: The objective of this course is to......
- ✓ Explain the ANSYS capabilities, terminology, and ANSYS Interactive (GUI).
- ✓ Build Finite Element Models in ANSYS.
- ✓ Run the basic steps of performing a complete ANSYS analysis.
- ✓ Apply and test meshing Techniques
- ✓ Interpret the results of post processing

Content:

- ✓ Introduction to CAE
- ✓ General FEM Procedure involved in FEA
- ✓ GUI of Ansys Workbench
- ✓ Working on Live Project: Methodologies used to solve the FEA problem
- ✓ CAD Modelling using Ansys Workbench/ 3D Modeler/Space Claim
- ✓ Pre-processing Part: Defining and Assigning of the Materials
- ✓ Methods of Discretization/ Meshing and optimizing the model using refine mesh
- ✓ Working with various boundary conditions
- ✓ Surface and Line Modelling
- ✓ Post Processing and result validations
- ✓ Basics of CFD
- ✓ Concept of entry length and determination using CFD tool
- ✓ Mixing flow in elbow
- ✓ Heat transfer Transient flow analysis
- ✓ Laminar model and Turbulent model analysis
- ✓ VOF model analysis

Internship Duration: Two weeks

Requirements: ANSYS software

Any students from third year and final year of engineering can register the course.Certificate will be provided after successful completion of the course.