

In-house internship program on

" Computational Fluid Dynamics
Simulation using ANSYS"

June 19-30,2023

Organized by

Department of Mechanical Engineering



**SAMRAT ASHOK TECHNOLOGICAL INSTITUTE,
VIDISHA (MP)-464001**

About Institute

Samrat Ashok Technological Institute, a premier institute of the region was established by Late Maharaja Jiwaji Rao Scindia on November 1, 1960 with a donation from Gangajali Trust fund. The Institute started with B.E. in Civil Engineering, Mechanical Engineering and Electrical Engineering. Now the Institute offers 10 full time Under Graduate leading to degree in Bachelor of Engineering and 10 full time Post Graduate Courses in the areas of Engineering, Science and Management. It is an autonomous institute, which is Grant in Aid by Government of Madhya Pradesh and managed by the Maharaja Jiwaji Rao Education Society chaired by Hon'ble Shrimant Jyotiraditya M. Scindia.

About Department

The Department of Mechanical Engineering is one of the oldest and well-developed department of the Institute. It has been established in 1960. It offers One UG Programme in Mechanical Engineering and one PG programme in Advanced Production System.

Chairman

Dr. R. K. Pandit (Director)

Convenor

Dr. Sanjay Katare (HOD, MED)

Program Coordinator(s):

Dr. Mangal Singh Lodhi

Dr. Kamlesh Sharma

Prof. Nikhil Mohan Vyas

Address for Correspondence:

Mr. Saurabh Dwivedi

Mob: +917828088778

Mr. Kuljeet Singh

Mob : +919691329129

Objective of Program

The primary objective of this In-house Internship Program (IIP) is to provide necessary training on Computational Fluid Dynamics (CFD) software that can be utilized for simulation of various fluid and thermal systems. The program will facilitate the students to understand the essential concepts of CFD tool, methodology through a hands-on training.

Content of Program

Following topics will be covered during the training program:

1. Introduction to CFD: Computational approach to Fluid Dynamics and its comparison with experimental and analytical methods
2. Governing differential equations and boundary conditions
3. Solution Methodology: Domain discretization using FDM, FVM and FEM, Stability, Convergence and Accuracy.
4. Finite Volume Method: Domain discretization, Pressure velocity coupling, with SIMPLE method.
5. Introduction CFD software: Geometry; grid, generation, solver, post processing.
6. Lab session on Simulation of Microchannel Heat Exchanger.
7. Lab session on Simulation of Natural Circulation Loop.
8. Lab session on Simulation of Vapour Compression Refrigeration System.