



Project Based CFD Training Program

Objectives:

- Platform for understanding Computational Fluid Dynamics (CFD) with well planned theoretical lectures and hands-on tutorials
- Usage of commercial tools such as ANSYS-FLUENT
- Facilitate academic projects execution at various stages: Project conceptualization; Literature survey; Model and parameters selection; Geometry development; Mesh generation; Simulation; Post-processing; Result, Data analysis, etc

Features:

- Fluid Dynamics and Heat Transfer in the context of CFD
- Hands-on tutorial sessions with commercial tool (ANSYS: Fluent, ICEM-CFD and CFD Post)
- Project Execution in group of 2 - 3 students or Bring your own project and training will help you in its execution
- Certification

Participants: SE/TE/BE/ME students

Organizer: Department of Mechanical Engineering
Army Institute of Technology, Pune 411015, INDIA

Resource Faculty:

Dr **Sanjay M Gaikwad**, Assistant Professor,

Contact No: +918007655827, Email ID: smgaikwad@aitpune.edu.in

Dr **P M Purohit**, Associate Professor,

Contact No: +919145231814, Email ID: pmpurohit@aitpune.edu.in

Dr **R B Gurav**, Assistant Professor,

Contact No: +917350057110, Email ID: rbgurav@aitpune.edu.in