

# COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics (CFD) is an efficient computerized method of studying Fluid Mechanics and Heat Transfer based on numerical analysis. CFD simplifies the complications related to experimental methods and provides detailed characterization of three-dimensional flow fields. CFD is applied to a wide range of Research and Engineering problems i.e. Aerodynamics and Aerospace Analysis, Weather Simulation, Natural Science and Environmental Engineering, Industrial System Design and Analysis, Biological Engineering, Fluid Flows and Heat Transfer, Engine and Combustion Analysis.

Facilities in CFD Lab:

1. Computers with high computation capability (20 Nos.)
2. ANSYS V 19 Software
3. MATLAB Software
4. HTRI Exchanger Suite Educational Software