Computational Fluid Dynamics: An Insight & Applications

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. The fundamental basis of almost all CFD problems is the Navier–Stokes equations, which define many single-phase (gas or liquid, but not both) fluid flows. The most popular method is called the Finite Volume Method wherein the governing partial differential equations (typically the Navier-Stokes equations, the mass and energy conservation equations, and the turbulence equations) are recast in a conservative form, and then solved over discrete control volumes. This discretization guarantees the conservation of fluxes through a particular control volume. The tutorial/workshop would involve CFD Case Studies involving internal pipe flows and Heat transfer within solids and provide the participants an exposure to the various steps involved in CFD simulation such as geometry creation, meshing, discretizing the governing equations, solution algorithm and post processing.

Faculty: Dr V J Lakhera, Professor, Mechanical Engg Deptt., IT, NU