Ansys CFD

Mode: Classroom

Duration: 30 hours

Target Audience:

- Mechanical Designers
- Chemical Engineers
- Structural Analysts

Course Contents:

DAY-1 (3 HRS.)	Introduction to CFD, familiarization with ANSYS interface
DAY-2 (3 HRS.)	Geometry creation using Ansys Design Modeller (2D/3D/Complex geometries)
DAY-3 (3 HRS.)	Mesh creation using Ansys Meshing
DAY-4 (3 HRS.)	Problem Setup in Ansys Fluent – Material Properties, Adding & Changing Materials
DAY-5 (3 HRS.)	Problem Setup in Ansys Fluent - Models, Operating Conditions, Cell-zone conditions
DAY-6 (3 HRS.)	Problem Setup in Ansys Fluent – interfaces, Solution Procedure & Control
DAY-7 (3 HRS.)	Steady-state Solution
DAY-8 (3 HRS.)	Transient Solution
DAY-9 (3 HRS.)	Results visualization and Data writing
DAY-10 (3 HRS.)	User-Defined functions Summary & Wrap-up