

Introduction to Gambit and ANSYS Fluent.

Duration: (5 Days)

This course is designed for clients who have little or no experience with ANSYS FLUENT and GAMBIT computational fluid dynamics (CFD) software.

Course contents for Day 1 and Day 2: Gambit

GAMBIT is a pre processor and is used to meet the specific geometry and meshing needs of our users. This course includes both geometry modeling and mesh generation tools for structured, unstructured and hybrid meshing.

Course Topics Include:

- Introduction to Gambit
- Geometry creation
- Edge and Face meshing
- Volume meshing
- Boundary layers and size functions
- Meshing strategy
- CAD import and cleanup.

Course contents for Day 3 through Day 5: ANSYS Fluent.

The primary goal of this course is to cover the basics of working in the ANSYS Fluent software environment. All course material is designed to make users of ANSYS Fluent comfortable for calculating broad range of real-world CFD problems from start to finish. Users will have hands-on time to work through the entire simulation process, and learn about the range of physical models available along with the strengths and weaknesses of each option.

Course Topics Include:

- Solver Basics
- Boundary conditions
- Solver settings
- Heat Transfer
- Turbulence
- Multiphase
- User Defined Functions
- Moving zones.

Each course chapter is followed by "hands-on" workshops and exercises.

Fee: Rs 25000 /- + Applicable Taxes

For more information / registration contact:

Email: india-register@ansys.com

Phone: +91-20-66522545 ; **Fax :** +91-20-66522600