

CFD with Gambit and Fluent

This course is intended for people who have little or no experience with FLUENT and GAMBIT, but who have a basic understanding of fluid mechanics. Attendees will be given the opportunity to solve a number of tutorials during class or bring their own problems to work with. After the course the participants are ready to start working with the software and they will have a clear sense of the modeling possibilities in Gambit and Fluent.

The following topics are discussed:

- The CFD process
- Overview of Gambit and Fluent
- File management
- Geometry creation
- Topology theory
- Meshing techniques
- Boundary conditions
- Material properties
- Solver control
- Convergence and accuracy
- Adaption
- Post processing
- Modeling heat transfer
- Modeling turbulence
- Modeling reacting flows
- Multiphase modeling
- Modeling with moving zones
- User defined functions