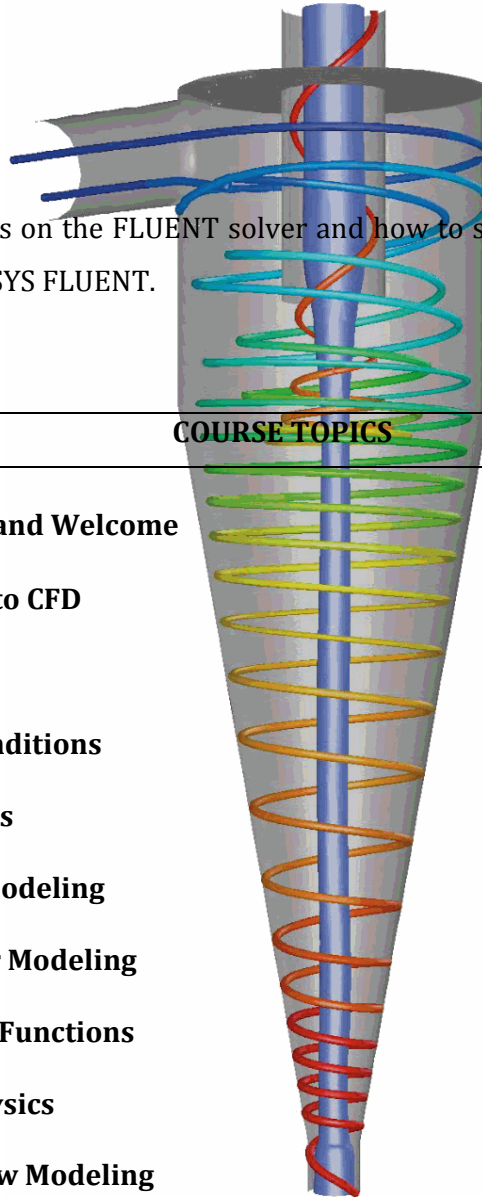


## ANSYS FLUENT Introductory Course

**Length:** 3 Days

### Course Overview

The training will focus on the FLUENT solver and how to setup, solve and post-process your results from ANSYS FLUENT.



### COURSE TOPICS

1. Introduction and Welcome
2. Introduction to CFD
3. Solver Basics
4. Boundary Conditions
5. Solver Settings
6. Turbulence Modeling
7. Heat Transfer Modeling
8. User-Defined Functions
9. Advanced Physics
10. Transient Flow Modeling
11. Postprocessing with FLUENT and CFD-Post