



CAD-IT CONSULTANTS (ASIA) PTE LTD TRAINING COURSES

COURSE NAME : Introduction to ANSYS FLUENT 12.0
DURATION : 4 x 8hrs sessions

* Creditable for QUT Master program and Continuing Development (CPD) courses offered by the Professional Engineers Board (PEB) Singapore.

OBJECTIVES

The primary goal of this course is to cover the basics of working in the FLUENT software environment. All course material is designed to educate participants on the best practices for effectively calculating a broad range of real-world CFD problems from start to finish.

WHO SHOULD ATTEND

This training course is intended for clients who have little to no experience with the ANSYS FLUENT 12.0 computation fluid dynamics (CFD) software.

PRE-REQUISITES

Engineering knowledge is required. Familiarity with computer (PC or workstation) and knowledge of CFD is useful.

COURSE SYLLABUS

Day One: DesignModeler

- ANSYS Workbench Overview
- Introduction to DesignModeler
- Planes and Bodies
- CAD Geometry
- Sketching
- Modeling
- Negative Fluid Geometry
- Parameterization

Day Two: ANSYS Meshing

- Introduction to ANSYS Meshing
- Interface/ Contact Modeling
- Meshing Methods
- Common Mesh Controls
- Tetrahedral Meshing
- Sweep Meshing
- Multizone Meshing
- Meshing 2D Geometries
- Mesh Quality

Day Three: ANSYS FLUENT

- Introduction to CFD Analysis
- Solver Basics
- Boundary Conditions
- Solver Settings

Day Four: ANSYS FLUENT

- Turbulence Modeling
- Heat Transfer Modeling
- User Defined Functions
- Advanced Physics
- Transient Flow Modeling
- Post-processing With FLUENT/ CFD-Post

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

CAD-IT CONSULTANTS (ASIA) PTE LTD

159 SIN MING ROAD #03-05 AMTECH BUILDING SINGAPORE 575625

TEL : (65) 6508 7575 FAX : (65) 6454 3766 EMAIL : cadittraining@cadit.com.sg
<http://www.cadit.com.sg>