


# Introduction to ANSYS Fluent

16.0 Release




Fluid Dynamics



Structural Mechanics



Electromagnetics



Systems and Multiphysics

**FLUID**  
C O D E S



**ANSYS CHANNEL PARTNER FOR MIDDLE EAST**

## Agenda

# Agenda (Day 1). ANSYS Fluent

## Morning

Lecture – Chapter 1: Introduction to the CFD Methodology / Demo

### Workshop 1. Flow study in Pipe Intersection

Lecture – Chapter 2: Cell Zone and Boundary Conditions

Lecture – Chapter 3: Post Processing

## Afternoon

### Workshop 2. Multi Species Post Processing

Lecture – Chapter 4: Solver Settings

Lecture – Chapter 5. Turbulence Modeling

### Workshop 3. Study of Turbulent effects with sudden change in diameter of pipes

Lecture – Chapter 6. Summary / Review of Day 1

# Agenda (Day 2). ANSYS Fluent

## Morning

Lecture – Chapter 7: Introduction to Multiphase Flow Modeling

**Workshop 6. Bubble Column ( 2-phase gas-liquid flow in pipelines)**

Lecture – Chapter 8: Discrete Phase Model

**Workshop 4. Study of Solid Particle Flowing Through Pipes**

Lecture – Chapter 9: Free surface flows Modeling

**Workshop 5. Tank Flush**

## Afternoon

Lecture – Chapter 10: Best Practices

**Workshop 6. Study of Phase change in pipe line.**

Lecture – Chapter 11: Transient

Lecture – Chapter 12: Review