Engineers 3D
IT & ENGINEERING SERVICES
TRAINING   PROJECTS   PLACEMEMNTS

ANSYS Software Training
Course Curriculum

Chapter 1: Before you start using Ansys

    a. Introduction to the Finite Element Method

    b. What is the Finite Element Method?

    c. General Steps of the Finite Element Method

    d. Explanation of 1D, 2D and 3D Elements with examples of ANSYS Elements

    e. Need of FEM

    f. Types of analysis that can be done using ANSYS

    g. Advantages of the Finite Element Method

    h. Limitations of FEA

    i. About ANSYS Inc.

    j. ANSYS Family of products with their capabilities

    k. Types of analysis that can be done with ANSYS.

    l. Introduction to the Ansys GUI

    m. Operation Modes of Ansys

    n. Product Launcher

    o. Launcher Menu Options

    p. The ANSYS GUI

    q. The Icon Toolbar Menu

    r. Quitting Ansys

Chapter 2: Selection Logic

a. Pan-Zoom-Rotate
b. Picking
c. Coordinate Systems

Chapter 3: Solid Modeling

a. An Overview of Solid Modeling Operations
b. Working with Boolean operations
c. Working Plane
d. Importing of 3D models

Chapter 4: Meshing

a. Free meshing or Mapped meshing
b. Setting Element Attributes
c. Selecting Element Type
d. Shape Function
e. Defining Element Types
f. Defining Section Properties
g. Assigning Element Attributes before meshing
h. Mesh Controls
i. The ANSYS Mesh Tool
j. Smart sizing
k. Meshing
l. Free Meshing
m. Mapped Meshing
n. Hybrid meshing
o. Mesh Extrusion
p. Volume Sweeping

Chapter 5: Material Properties

a. Material Library
b. Specifying properties

Chapter 6: Boundary Conditions

    a.   Types of Loads

    b.   Applying loads

Chapter 7: Solvers

    a.   Types of Solvers

    b.   Solver Setup

    c.   Load Step Options

    d.   Solving Multiple Load Steps

Chapter 8: Post-processing

    a.   Contour Plot Viewing

    b.   Time History Postprocessor (POST26)

    c.   Report Generator

Chapter 9: Introduction to Non-Linear Analysis

Chapter 10: Sample Structural Analysis

    a. Workshop

Chapter 11: Sample Thermal Analysis

    a. Workshop

Chapter 12: Sample Modal Analysis

    a. Workshop

Chapter 13: Tips & Tricks

    a.   Using the Toolbar & Creating Abbreviations

    b.   Introduction to APDL

    c.   Using Parameters

    d.   Using the Start File

    e.   Using the Session Editor

    f.   Using Input Files

Chapter 14: ANSYS Workbench

    a.   Introduction to ANSYS Workbench

    b.   Graphical User Interface

    c.   Static Structural Analysis

    d.   Modal Analysis

    e.   Thermal Analysis

    f.   Contact Recognition

****