IFS ACADEMY

Training For The Future!!

Finite Element Analysis using MATLAB & ANSYS

(As per revised syllabus of Savitribai Phule Pune University Elective IV

(BE-Mechanical) 2012 Course)

Course Curriculum (70 Hrs.)

Chapter 1: Introduction to Finite Element Method

a. Introduction to the Finite Element Method
b. What is the Finite Element Method?
c. History
d. General Steps of the Finite Element Method
e. Explanation of 1D, 2D and 3D Elements
f. Need of FEM
g. Enlisting different FEM methods and detailed explanation of any one
h. Derivation of stiffness matrix equation
i. Steady State Heat Transfer
j. Dynamic Analysis

Chapter 2: Fundamentals of MATLAB

a. The MATLAB Windows
b. Reading data from files
c. Saving and loading variables
d. Plotting data
e. Customizing plots
f. Exporting graphics for use in other applications
g. Entering commands
h. Creating numeric variables
i. Creating character variables
j. Making and annotating plots
k. Getting help
l. Accessing and modifying values in variables
m. Performing calculations with vectors
n. Creating multiple plots
o. Using the Command History
p. Creating script files
q. Running scripts
r. Dividing code into sections
s. Publishing scripts
t. Creating and manipulating matrices
u. Performing calculations with matrices
v. Calculating statistics with matrix data
w. Visualizing matrix data
x. Logical operations and variables
y. Finding and counting
z. Logical indexing
aa. Importing from spreadsheets and delimited text files
bb. Dealing with missing data
cc. Plotting functions
dd. Customizing plots
ee. Creating functions

ff. Calling functions
gg. Setting the MATLAB path
hh. Debugging with the MATLAB Editor
ii. Using breakpoints
jj. Workshops / Assignments

Chapter 3: ANSYS Mechanical APDL

a. ANSYS Mechanical APDL GUI
b. Selection Logic
c. Solid Modeling
d. Meshing
e. Material Properties
f. Boundary Conditions
g. Solvers
h. Post Processing
i. Static Structural Analysis
j. Modal Analysis
k. Heat Transfer Analysis
l. Workshops & Assignments

Chapter 4: ANSYS Workbench

a. ANSYS Workbench Project Window
b. Design Modeler
c. Static Structural Analysis
d. Modal Analysis
e. Heat Transfer Analysis
f. Workshops & Assignments

Chapter 5: Workshops / Assignments

MATLAB Assignments:

a. Computer program for stress analysis 2-D truss subjected to plane forces
b. Computer program for modal analysis 1-D beam (simply supported or cantilever beams)
c. Computer program for frames subjected to transverse forces and moments
d. Computer program for 1-D temperature analysis

ANSYS Assignments:

e. Static stress concentration factor calculation for a plate with center hole subjected to axial loading in tension using FEA software.
f. 2D Forced convection problem using FEA software.
g. Modal analysis of any machine component using FEA software.
h. Stress and deflection analysis of any machine component consisting of 3-D elements using FEA software.

****

IFS Academy, Pune

Phone: +91-20-6400 7296, Email: training@ifsacademy.org,

Visit Us At: www.ifsacademy.org