******** Introduction to FEA and Ansys*******

- Introduction to FEA
- General Working of FEA
- Nodes, Elements, and Element Shapes
- General Procedure of Conducting Finite Element Analysis
- FEA through ANSYS
- Effective Utilization of FEA
- FEA Software
- Advantages and Limitations of FEA Software
- Key Assumptions in FEA
- Assumptions Related to Geometry
- Assumptions Related to Material Properties
- Assumptions Related to Boundary Conditions
- Assumptions Related to Fasteners
- Types of Analysis
- Structural Analysis
- Thermal Analysis
- Fluid Flow Analysis
- Electromagnetic Field Analysis
- Coupled Field Analysis
- Important Terms and Definitions
- Strength (Resistance to Deformation)
- Load
- Stress
- Strain
- Elastic Limit
- Ultimate Strength
- Factor of Safety
- Lateral Strain and Poisson's Ratio
- Bulk Modulus
- Creep
- Engineering Materials
- Introduction to ANSYS
- System Requirements
- Getting Started with ANSYS
- Interactive Mode
- Batch Mode
- Starting a New File Using the ANSYS Product Launcher window
- ANSYS Output Window
- ANSYS Metaphysics Utility Menu Window (ANSYS Session)
- Utility Menu
- Main Menu
- Graphics Area
- Standard Toolbar

Quickstep Computer Centre

- ANSYS Command Prompt
- Command Window Icon
- Raise Hidden Icon
- Reset Picking
- Contact Manager
- ANSYS Toolbar
- Model Control Toolbar
- User Prompt Information
- Current Settings
- Setting the Analysis Preferences
- Units in ANSYS
- Other Important Terms Related to ANSYS
- Dialog Boxes
- Graphics Display
- Panning, Zooming, and Rotating the Model
- Dividing the Graphics Area
- The Pan-Zoom-Rotate Dialog Box
- Graphics Picking
- Using Mouse Buttons for Picking
- ANSYS Database and Files
- Saving the File
- Resuming the File
- Clearing the Database
- Some Basic Steps in General Analysis Procedure
- Points to Remember while Performing an Analysis
- Exiting ANSYS
- Self-Evaluation Test

➢ Basic Solid Modeling

- Solid Modeling in ANSYS
- Solid Modeling and Direct Generation
- Solid Modeling Methods
- Bottom-up Construction
- Top-down Construction
- Considerations before Creating a Model for Analysis
- Details Required
- Symmetry
- Creating Geometric Entities
- Creating Lines
- Creating Arcs
- Creating B-Spines
- Creating Fillets between Intersecting Lines
- Creating Areas
- Creating and Modifying Work planes
- Display Working Plane
- Show WP Status
- WP Settings

CubicStep Computer Centre

- Offset WP by Increments
- Offset WP to
- Align WP with
- Coordinate Systems in ANSYS
- Global Coordinate System
- Local Coordinate System
- Active Coordinate System
- Display Coordinate System
- Nodal Coordinate System
- Element Coordinate System
- Results Coordinate System
- Creating New Coordinate Systems
- Deleting Existing Coordinate

➢ Advanced Solid Modeling

- Advanced Solid Modeling
- Creating Volumes
- Extruding Entities
- Extending the Line
- Creating Complex Solid Models by Performing Boolean Operations
- Modifying the Solid Model
- Scale
- Move
- Copy
- Reflect
- Deleting Solid Model Entities
- Importing Solid Models
- Importing the IGES File
- Importing Models from Pro/ENGINEER
- Importing the Model from Unigraphics

➢ Finite Element Modeling (FEM) – I

- An Overview of the Finite Element Modeling
- Element Attributes
- Element Types
- Reasons Why ANSYS has a Large Element Library
- Real Constants
- Material Properties
- Multiple Attributes
- Assigning Multiple Attributes before Meshing
- Assigning Default Attributes before Meshing
- Modifying Attributes after Meshing
- Verifying Assigned Attributes
- Element Attributes Table

➢ Finite Element Modeling (FEM) – II

Quickstep Computer Centre

- Finite Element Modeling (FEM) - II
- Mesh Generation
- Mesh Density
- Meshing the Solid Model
- Setting Element Attributes
- Defining the Mesh
- Defining the Entity to be Meshed
- Defining the Meshing Type
- Meshing the Model
- Refining the Mesh Locally
- Extruding the Mesh
- Transitional Pyramid Elements
- Requirements for Creating Pyramid Elements
- Creating Transitional Pyramid Elements (Hex-to-Tet Meshing)
- Converting Degenerate Tetrahedral (20 nodes) Elements into
- Non-degenerate (10 nodes) Tetrahedral Elements
- Plotting Pyramid Elements
- Meshing the Beam with Orientation Nodes
- Creating the Beam Mesh with Orientation Nodes
- Creating the Beam Mesh with Two Orientation Nodes
- Improving the Tetrahedral Element Meshes
- Improving Tetrahedral Meshed Volumes by Using Volumes
- Improving Tetrahedral Meshed Volumes by Using Detached Elements
- Some Additional Tips while Meshing the Model
- Applying Loads
- The Nodal Coordinate System
- Loads in Different Disciplines
- Types of Loads in ANSYS
- Load Steps, Sub steps, and Time
- Applying Loads
- Deleting Loads
- Deleting DOF Constraints
- Deleting all Loads and Load Step Options
- Deleting all Loads Applied on Solid Model
- Deleting all Loads Applied on Finite Element Model

➢ Solution And Postprocessor

- Solution
- Defining the New Analysis Type
- Restarting the Analysis
- Setting Solution Controls
- Setting Analysis Options
- Solving the Analysis Problem
- Post processing the Result
- POST1 (General Postprocessor)
- POST26 (Time-history Postprocessor)
- Result Coordinate System (RSYS)

Quikstep Computer Centre

- Displaying the Deformed Shape of the Model
- Displaying the Minimum and Maximum Stresses
- Listing Reaction Forces
- Listing Stress Values at each Node
- Query Picking
- Path Operations
- Load Case Combinations

➢ Static Structural Analysis

- Effect of self-weight on a cantilever beam
- Analysis of a bicycle handle
- Analysis of a stud (pin)
- Analysis of a master

➢ Advanced Structural Analysis (Dynamic And Nonlinear)

- Advanced Structural Analysis
- Dynamic Analysis
- Performing the Modal Analysis
- Specifying the Analysis Type, Analysis Options, and Applying Loads
- Obtaining the Solution
- Reviewing Results
- Performing the Harmonic Analysis
- Specifying the Analysis Type, Analysis Options, and Applying Loads
- Obtaining the Solution
- Reviewing Results
- Performing the Transient Analysis
- Specifying the Analysis Type, Analysis Options, and Applying Loads
- Obtaining the Solution
- Reviewing Results
- Nonlinear Analysis
- Geometric Nonlinearity
- Material Nonlinearity
- Boundary Nonlinearity (Changing Status)
- Performing the Nonlinear Analysis
- Specifying the Analysis Type, Setting Solution Controls, and
- Applying Loads
- Obtaining the Solution

➢ Advanced Structural Analysis

- Steel tubes and springs structure
- Modal analysis of an airplane wing
- Nonlinear analysis (material nonlinearity)

➢ Thermal Analysis

- Thermal Analysis
- Important Terms Used in Thermal Analysis

Quintep Computer Centre

- Heat Transfer Modes
- Thermal Gradient
- Thermal Flux
- Bulk Temperature
- Film Coefficient
- Emissivity
- Stefan–Boltzmann Constant
- Thermal Conductivity
- Specific Heat
- Types of Thermal Analysis
- Steady-State Thermal Analysis
- Transient Thermal Analysis
- Performing Steady-State Thermal Analysis
- Setting the Analysis Preference
- Creating or Importing a Solid Model
- Defining Element Attributes
- Meshing the Solid Model
- Specifying the Analysis Type, Analysis Options, and Applying Loads
- Solving the Analysis Problem
- Post processing Results
- Performing Transient Thermal Analysis
- Specifying the Analysis Type and Setting Solution Controls

➢ Generating The Report Of Analysis

- Starting the ANSYS Report Generator
- Capturing Images for the Report
- Capturing Animations for the Report
- Capturing Data Tables for the Report
- Capturing Lists for the Report
- Compiling the Report
- Changing the Default Settings of the ANSYS Report Generator
- Error Estimation in Solution
- Percentage Error in Energy Norm (SEPC)
- Element Energy Error (SERR)
- Element Stress Deviations (SDSG)
- Maximum and Minimum Stress Bounds (SMXB and SMNB)

Quickstep Computer Centre