



TECHHUB SOLUTIONS

...be step ahead in your field

Finite Element Analysis using ANSYS (ANSYS Mechanical APDL & Workbench with CFD)

Section I: ANSYS Mechanical APDL

Chapter 1: Before you start using ansys

- a. Introduction to FEA
- b. FEA through Ansys
- c. Preprocessor
- d. Solutions
- e. postprocessor
- f. Setting the analysis preferences
- g. h-method
- h. p-method
- i. Units in Ansys
- j. Panning, zooming and rotating the model
- k. Dividing the graphics areas
- l. Saving the file.
- m. Resuming the file
- n. Clearing and new file database
- o. Ansys help
- p. Existing ansys

Chapter 2: Solid modeling methods

- a. Bottom –up construction.
- b. Top –down construction

Chapter 3: Creating geometric entities

- a. keypoints
- b. hard points
- c. lines
- d. arcs
- e. splines
- f. area

Chapter 3A : 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. Real Constants

- g. Defining Section Properties
- h. Assigning Element Attributes before meshing
- i. Mesh Controls
- j. The ANSYS MeshTool
- k. Smartsizing
- l. Meshing
- m. Free Meshing
- n. Mapped Meshing
- o. Hybrid meshing
- p. Mesh Extrusion
- q. 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. Path Operations
- c. Estimating Solution Error
- d. Time History Postprocessor (POST26)
- e. Report Generator

Chapter 9: Introduction to Non-Linear Analysis

Chapter 10: Static Structural Analysis

- a. Workshops, Exercises and Case Studies

Chapter 11: Modal Analysis

- a. Workshops, Exercises and Case Studies

Chapter 12: Thermal Analysis

- a. Workshops, Exercises and Case Studies

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

Section II: ANSYS Workbench

Chapter 1: Introduction to ANSYS Workbench

- a. ANSYS Workbench Overview
- b. Workbench File Management
- c. Working with Units

Chapter 2: Mechanical Basics

- a. Basic Analysis Procedure
- b. The Mechanical Interface
- c. Menus
- d. Toolbars
- e. Graphics Control and Selection
- f. Outline Tree and Details
- g. Graphics Window
- h. The Mechanical Application Wizard
- i. Scoping Loads and Supports
- j. The Engineering Data Application
- k. Workshop: ANSYS Mechanical Basics

Chapter 3: General Preprocessing

- a. Geometry Branch
- b. Contact
- c. Remote Boundary Conditions
- d. Coordinate Systems
- e. Named Selections
- f. Selection Information
- g. Workshop: Contact Control

Chapter 4: Meshing in Mechanical

- a. Global Meshing Controls
- b. Local Meshing Controls
- c. Meshing Troubleshooting
- d. Workshop: Meshing Control

Chapter 5: Static Structural Analysis

- a. Workshops, Exercises and Case Studies

Chapter 5: Modal Analysis

- a. Workshops, Exercises and Case Studies

Chapter 6: Thermal Analysis

- a. Workshops, Exercises and Case Studies

Chapter 7: CFD Analysis

- a. Workshops, Exercises and Case Studies

Chapter 7: Results and Postprocessing

- a. Viewing Results
- b. Exporting Results
- c. Coordinates Systems
- d. Convergence
- e. Stress Singularities
- f. Workshop: Results Processing