

Table of Contents

Dedication	iii
Preface	xi

Chapter 1: Introduction to FEA

Design Validation Techniques	1-2
Analytical Method	1-2
Numerical Method	1-2
Experimental Method	1-2
Introduction to FEA	1-3
General Working of FEA	1-4
Types of Element	1-6
General Procedure to Conduct Finite Element Analysis	1-7
Coordinate Systems	1-8
FEA Software	1-9
Advantages and Limitations of FEA Software	1-9
Key Assumptions in FEA	1-9
Assumptions Related to Geometry	1-9
Assumptions Related to Material Properties	1-10
Assumptions Related to Boundary Conditions	1-10
Assumptions Related to Fasteners	1-10
Applications of FEA	1-10
Automobile Applications	1-10
Manufacturing Process Applications	1-10
Electromagnetics Applications	1-11
Aerospace Applications	1-11
Types of Engineering Analysis	1-11
Structural Analysis	1-11
Thermal Analysis	1-13
Fluid Flow Analysis	1-14
Electromagnetic Field Analysis	1-14
Coupled Field Analysis	1-14
Important Terms and Definitions	1-14
Strength	1-15
Load	1-14
Stress	1-15
Strain	1-15
Elastic Limit	1-15
Ultimate Strength	1-16
Factor of Safety	1-16
Lateral Strain	1-16
Poisson's Ratio	1-17
Stress Concentration	1-16
Bending	1-17

Bending Stress	1-17
Creep	1-17
Classifications of Materials	1-17
Aspect Ratio	1-17
Axisymmetry	1-18
Degrees of Freedom (DOF)	1-18
Self-Evaluation Test	1-20
Review Questions	1-20

Chapter 2: Introduction to ANSYS Workbench

Introduction to ANSYS Workbench	2-2
System Requirements	2-3
Starting ANSYS Workbench 2026 R1	2-3
Toolbox Window	2-4
Project Schematic Window	2-11
Menu bar	2-12
Main and Tab Toolbar	2-13
Shortcut Menu	2-12
Working on a New Project	2-13
Adding a System to a Project	2-13
Renaming a System	2-15
Deleting a System from a Project	2-15
Duplicating a System in a Project	2-17
Saving the Current Project	2-17
Opening a Project	2-18
Archiving the Project Data	2-19
Units in ANSYS Workbench	2-20
ANSYS Workbench Database and File Formats	2-21
Changing the Unit Systems	2-22
Components of a System	2-22
Engineering Data Cell	2-23
Geometry Cell	2-24
Model Cell	2-24
Mesh Cell	2-24
Setup Cell	2-25
Solution Cell	2-25
Results Cell	2-25
States of a cell in an Analysis System	2-26
Refreshing and Updating a Project	2-26
Adding Second System to a Project	2-27
Adding Connectors	2-28
Connector/Link Types	2-29
Deleting Connectors/Links	2-29
Specifying a Geometry for Analysis	2-29
Creating a Geometry	2-29
Using Help in ANSYS Workbench	2-31
ANSYS Workbench Help	2-31

Quick Help	2-31
Context Sensitive Help	2-31
Exiting ANSYS Workbench	2-32
Tutorial 1	2-32
Tutorial 2	2-34
Self-Evaluation Test	2-41
Review Questions	2-42
Exercise 1	2-43

Chapter 3: Part Modeling - I

Introduction to Part Modeling	3-2
Introduction to DesignModeler Window	3-2
Sketching Mode	3-3
Modeling Mode	3-3
Screen Components of DesignModeler Window	3-4
Tree Outline	3-4
Details View window	3-5
Model View/Print Preview	3-5
Ruler	3-5
Triad	3-6
Status Bar	3-6
Tutorial 1	3-6
Tutorial 2	3-29
Tutorial 3	3-40
Self-Evaluation Test	3-57
Review Questions	3-58
Exercise 1	3-58
Exercise 2	3-59
Exercise 3	3-59

Chapter 4: Part Modeling- II

Introduction to Concept Menu	4-2
Concept Menu	4-2
Tutorial 1	4-4
Tutorial 2	4-11
Tutorial 3	4-20
Tutorial 4	4-25
Self-Evaluation Test	4-34
Review Questions	4-35
Exercise 1	4-36

Chapter 5: Part Modeling- III

Introduction to 3D Feature	5-2
Extrude	5-2

Revolve	5-3
Sweep	5-4
Skin/Loft	5-5
Thin/Surface	5-6
Tutorial 1	5-7
Tutorial 2	5-24
Tutorial 3	5-37
Self-Evaluation Test	5-52
Review Questions	5-53
Exercise 1	5-54

Chapter 6: Defining Material Properties

Introduction to Engineering Data Workspace	6-2
Creating and Adding Materials	6-4
Creating a New Material in the Outline Window	6-4
Tutorial 1	6-5
Tutorial 2	6-10
Tutorial 3	6-16
Self-Evaluation Test	6-22
Review Questions	6-23
Exercise 1	6-23

Chapter 7: Generating Mesh- I

Introduction	7-2
Refining the Mesh	7-5
Decision Making to Find Optimum Results	7-5
Tutorial 1	7-6
Tutorial 2	7-11
Tutorial 3	7-27
Self-Evaluation Test	7-43
Review Questions	7-44
Exercise 1	7-45

Chapter 8: Generating Mesh - II

Tutorial 1	8-2
Tutorial 2	8-9
Tutorial 3	8-23
Tutorial 4	8-38
Self-Evaluation Test	8-50
Review Questions	8-50
Exercise 1	8-51

Chapter 9: Static Structural Analysis

Introduction to Static Structural Analysis	9-2
Linear Static Analysis	9-2
Non Linear Analysis	9-2
Fatigue Analysis	9-2
Mean Stress	9-2
Alternating Stress	9-3
Endurance Limit	9-3
Endurance Strength	9-3
Stress-Life Approach	9-3
Strain-Life Approach	9-3
Buckling Analysis	9-3
Points to Remember	9-4
General Procedure to Conduct Static Structural Analysis	9-4
Pre-Processing	9-5
Solution	9-6
Post-Processing	9-6
Tutorial 1	9-7
Tutorial 2	9-20
Tutorial 3	9-42
Tutorial 4	9-55
Self-Evaluation Test	9-67
Review Questions	9-68
Exercise 1	9-68
Exercise 2	9-69

Chapter 10: Vibration Analysis

Introduction to Vibration analysis	10-2
Model Analysis	10-2
Harmonic Analysis	10-3
Performing the Model Analysis	10-4
Adding Model Analysis to ANSYS Workbench	10-4
Starting the Mechanical Window	10-4
Specifying Analysis Setting	10-5
Plotting the Deformed Shape (Mode Shape)	10-6
Tutorial 1	10-7
Tutorial 2	10-14
Tutorial 3	10-29
Self-Evaluation Test	10-47
Review Questions	10-48
Exercise 1	10-49
Exercise 2	10-49

Chapter 11: Thermal Analysis

Introduction to Thermal Analysis	11-2
Important Terms Used in Thermal Analysis	11-2
Heat Transfer Mode	11-2
Thermal Gradient	11-3
Thermal Flux	11-3
Bulk Temperature	11-3
Film Coefficient	11-3
Emissivity	11-3
Stefan-Boltzmann Constant	11-3
Thermal Conductivity	11-3
Specific Heat	11-3
Types of Thermal Analysis	11-4
Steady-State Thermal Analysis	11-4
Transient Thermal Analysis	11-4
Thermal Stress Analysis	11-4
Tutorial 1	11-5
Tutorial 2	11-16
Tutorial 3	11-26
Self-Evaluation Test	11-40
Review Questions	11-41
Exercise 1	11-41

Index	I-1
--------------	------------