ANSYS Workbench 14.0 for Engineers and Designers

(A Tutorial Approach)

CADCIM Technologies

525 St. Andrews Drive
Schererville, IN 46375, USA
(www.cadcim.com)

Contributing Author

Sham Tickoo

Professor
Department of Mechanical Engineering Technology
Purdue University Calumet
Hammond, Indiana, USA

DEDICATION

To teachers, who make it possible to disseminate knowledge
to enlighten the young and curious minds
of our future generations

To students, who are dedicated to learning new technologies
and making the world a better place to live in

THANKS

To the faculty and students of the MET department of
Purdue University Calumet for their cooperation

To Ashok Dasari and Gaurav Verma for technically editing this manuscript

To Anju Jethwani and Rajendra Badola for copy editing this manuscript

Online Training Program Offered by CADCIM Technologies

CADCIM Technologies provides effective and affordable virtual online training on various software packages related to Computer Aided Design, Manufacturing, and Engineering (CAD/CAM/CAE), computer programming languages, animation, architecture, and GIS. The training is delivered 'live' via Internet at any time, any place, and at any pace to individuals and the students of colleges, universities, and CAD/CAM training centers. The main features of this program are:

Training for Students and Companies in a Classroom Setting

Highly experienced instructors and qualified Engineers at CADCIM Technologies conduct the classes under the guidance of Prof. Sham Tickoo of Purdue University Calumet, USA. This team has authored several textbooks that are rated "one of the best" in their categories and are used in various colleges, universities, and training centers in North America, Europe, and in other parts of the world.

Training for Individuals

CADCIM Technologies with its cost effective and time saving initiative strives to deliver the training in the comfort of your home or work place, thereby relieving you from the hassles of traveling to training centers.

Training Offered on Software Packages

We provide basic and advanced training on the following software packages:

***CAD/CAM/CAE**: ANSYS Workbench, CATIA, Pro/ENGINEER Wildfire, Creo Paremetric, SolidWorks, Autodesk Inventor, Solid Edge, NX, AutoCAD, AutoCAD LT, Customizing AutoCAD, EdgeCAM, and ANSYS*

***Computer Programming**: C++, VB.NET, Oracle, AJAX, and Java*

***Animation and Styling**: Autodesk 3ds Max, 3ds Max Design, Maya, and Autodesk Alias*

***Architecture and GIS**: Autodesk Revit Architecture, AutoCAD Civil 3D, AutoCAD Revit Structure, and AutoCAD Map 3D*

For more information, please visit the following link:

http://www.cadcim.com

Note

If you are a faculty member, you can register by clicking on the following link to access the teaching resources: ***http://www.cadcim.com/Registration.aspx***. The student resources are available at ***http://www.cadcim.com***. We also provide Live **Virtual Online Training** on various software packages. For more information, write us at ***sales@cadcim.com***.

Table of Contents

Dedication iii
Preface ix

Chapter 1: Introduction to FEA

Introduction to FEA 1-2
General Working of FEA 1-3
Elements and Element Shapes 1-4
General Procedure to Conduct Finite Element Analysis 1-6
FEA Software 1-7
Advantages and Limitations of FEA Software 1-7
Key Assumptions in FEA 1-8
Assumptions Related to Geometry 1-8
Assumptions Related to Material Properties 1-8
Assumptions Related to Boundary Conditions 1-8
Assumptions Related to Fasteners 1-9
Types of Engineering Analyses 1-9
Structural Analysis 1-9
Thermal Analysis 1-11
Fluid Flow Analysis 1-11
Electromagnetic Field Analysis 1-11
Coupled Field Analysis 1-12
Important Terms and Definitions 1-12
Self-Evaluation Test 1-16
Review Questions 1-16

Chapter 2: Introduction to ANSYS Workbench

Introduction to ANSYS Workbench 2-2
System Requirements 2-2
Starting ANSYS Workbench 14.0 2-3
Toolbox Window 2-5
Project Schematic Window 2-9
Menu bar 2-10
Standard Toolbar 2-10
Shortcut Menu 2-11
Working on a New Project 2-11
Adding a System to a Project 2-11
Renaming a System 2-13
Deleting a System from a Project 2-14
Duplicating a System in a Project 2-14
Saving the Current Project 2-14
Opening a Project 2-15
Archiving the Project Data 2-15
Extracting the Archive File 2-17

Units in ANSYS Workbench 2-17
ANSYS Workbench Database and File Formats 2-18
Changing the Unit Systems 2-19
Components of a System 2-21
Engineering Data Cell 2-21
Geometry Cell 2-22
Model Cell 2-22
Mesh Cell 2-23
Setup Cell 2-23
Solution Cell 2-23
Results Cell 2-23
States of a cell in an Analysis System 2-23
Refreshing and Updating a Project 2-24
Adding Second System to a Project 2-25
Adding Connectors 2-25
Specifying a Geometry for Analysis 2-26
Creating a Geometry 2-27
Using Help in ANSYS Workbench 2-28
ANSYS Workbench Help 2-28
Quick Help 2-29
Context Sensitive Help 2-29
Exiting ANSYS Workbench 2-30
Tutorial 1 2-30
Self-Evaluation Test 2-32
Review Questions 2-33
Exercise 1 2-33

Chapter 3: Part Modeling - I

Introduction to Part Modeling 3-2
Introduction to DesignModeler Window 3-2
Sketching Mode 3-4
Modeling Mode 3-4
Screen components of DesignModeler Window 3-5
Tree Outline 3-5
Details View window 3-6
Model View/Print Preview 3-6
Ruler 3-6
Triad 3-6
Status Bar 3-7
Tutorial 1 3-7
Tutorial 2 3-29
Tutorial 3 3-40
Self-Evaluation Test 3-57
Review Questions 3-58
Exercise 1 3-59
Exercise 2 3-59
Exercise 3 3-60

Chapter 4: Part Modeling- II

Tutorial 1 4-2
Tutorial 2 4-8
Tutorial 3 4-18
Tutorial 4 4-22
Self-Evaluation Test 4-30
Review Questions 4-31
Exercise1 4-32

Chapter 5: Part Modeling- III

Tutorial 1 5-2
Tutorial 2 5-18
Tutorial 3 5-30
Self-Evaluation Test 5-44
Review Questions 5-45
Exercise 1 5-46

Chapter 6: Defining Material Properties

Introduction to Engineering Data Workspace 6-2
Creating and Adding Materials 6-4
    Creating and Adding a New Material in the Engineering Data Workspace 6-4
    Creating and Adding a New Material in the Library 6-4
Tutorial 1 6-7
Tutorial 2 6-12
Tutorial 3 6-17
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 7-6
Tutorial 1 7-6
Tutorial 2 7-11
Tutorial 3 7-23
Self-Evaluation Test 7-38
Review Questions 7-38
Exercise 1 7-39

Chapter 8: Generating Mesh - II

Tutorial 1 8-2
Tutorial 2 8-9
Tutorial 3 8-20

Self-Evaluation Test 8-33
Review Questions 8-33
Exercise 1 8-34

Chapter 9: Static Structural Analysis

Introduction to Static Structural Analysis 9-2
Pre-Processing 9-3
Solution 9-3
Post-Processing 9-3
Tutorial 1 9-4
Tutorial 2 9-17
Tutorial 3 9-31
Self-Evaluation Test 9-38
Review Questions 9-39
Exercise 1 9-39
Exercise 2 9-40

Chapter 10: Modal Analysis

Introduction to Modal Analysis 10-2
Performing the Modal Analysis 10-3
Adding Modal Analysis System to ANSYS Workbench 10-3
Starting the Mechanical Window 10-4
Defining the Analysis Type and Options 10-4
Plotting the Deformed Shape (Mode Shape) 10-5
Tutorial 1 10-6
Tutorial 2 10-12
Tutorial 3 10-21
Self-Evaluation Test 10-31
Review Questions 10-31
Exercise 1 10-32
Exercise 2 10-32

Chapter 11: Thermal Analysis

Introduction to Thermal Analysis 11-2
Important Terms Used in Thermal Analysis 11-2
Types of Thermal Analysis 11-4
Steady-State Thermal Analysis 11-4
Transient Thermal Analysis 11-4
Tutorial 1 11-5
Tutorial 2 11-15
Tutorial 3 11-24
Self-Evaluation Test 11-33
Review Questions 11-33
Exercise 1 11-34

Index **I-1**