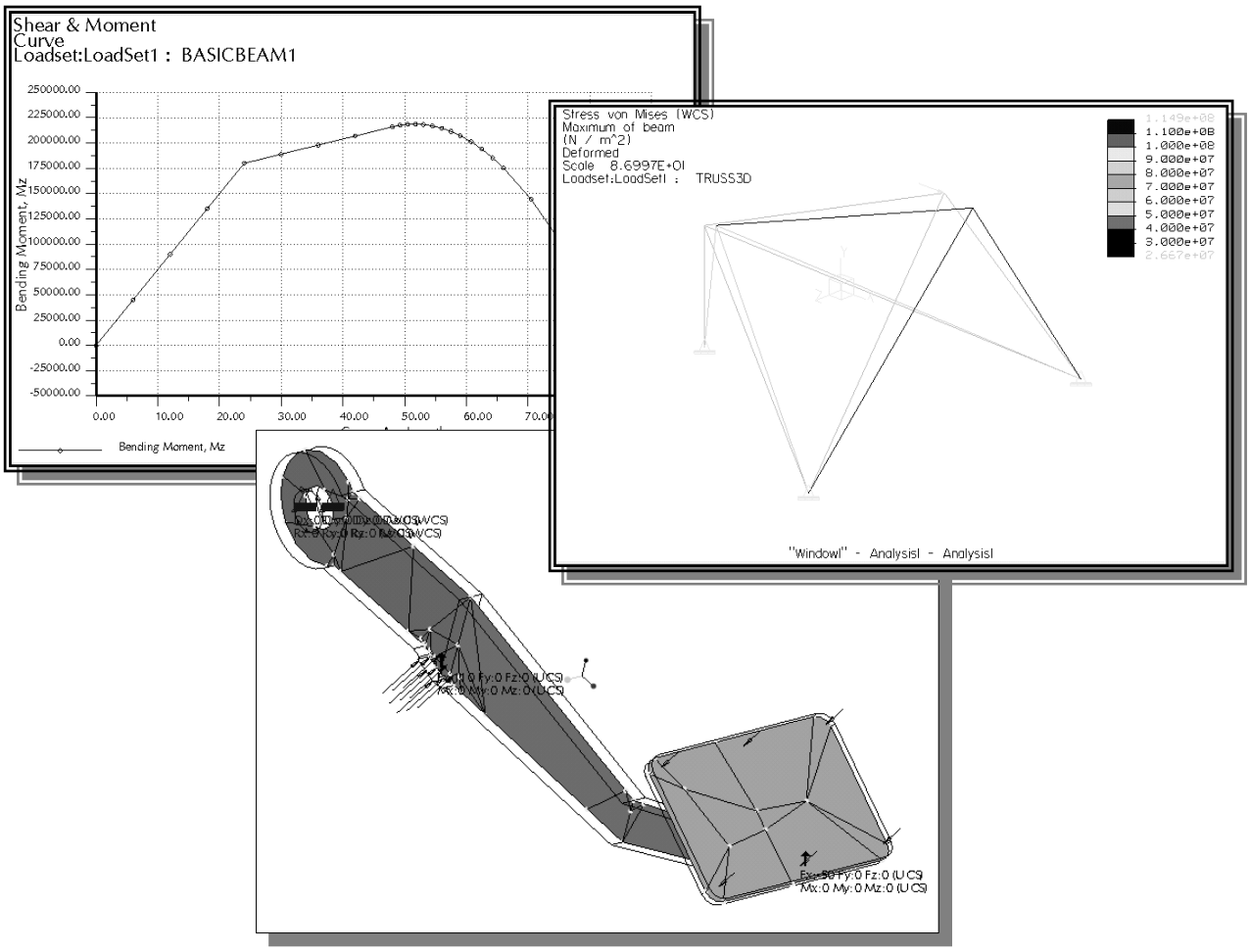
# Introduction to Finite Element Analysis

## Using Pro/MECHANICA Wildfire 4.0®




**Randy H. Shih**

***Oregon Institute of Technology***

SDC PUBLICATIONS

**Schroff Development Corporation**
**www.schroff.com**

Better Textbooks. Lower Prices.

# Table of Contents

**Preface** i
**Acknowledgments** ii

## Introduction

Introduction Intro-2
Development of Finite Element Analysis Intro-2
FEA Modeling Considerations Intro-3
Types of Finite Elements Intro-4
Finite Element Analysis Procedure Intro-6
Matrix Definitions Intro-6
Getting Started with *Pro/ENGINEER* Intro-9
Starting *Pro/ENGINEER* Intro-9
*Pro/ENGINEER* Screen Layout Intro-10
- Pull-down menus Intro-11
- Standard toolbar Intro-11
- Single Line Help Intro-11
- Dashboard Intro-11
- Graphics display Intro-11
- Datum toolbar Intro-12
- Navigator Intro-12
- Web Browser Intro-12

Basic Functions of Mouse Buttons Intro-13
Model Tree window and Feature Toolbars Intro-14
On-Line Help Intro-15
Leaving *Pro/ENGINEER* Intro-16
Creating a CAD files folder Intro-16

**Chapter 1**
## The Direct Stiffness Method

Introduction 1-2
One-dimensional Truss Element 1-3
Basic Solid Modeling using Pro/ENGINEER 1-10
The Adjuster design 1-10
Starting Pro/ENGINEER 1-11
Step 1: Units and Basic Datum Geometry Setups 1-12
Units Setup and Pro/ENGINEER Menu Structure 1-12
Tree Structure 1-13
Adding the First Part Features — Datum Planes 1-14
Step 2: Determine/Setup the Base Solid Feature 1-15
Defining the Sketching Plane 1-16

Defining the Orientation of the Sketching Plane 1-17
Step 3: Creating 2D rough sketches 1-19
Shape Before Size – Creating Rough Sketches 1-19
The Pro/ENGINEER SKETCHER and INTENT MANAGER 1-20
Graphics Cursors 1-21
Geometric Constraint Symbols 1-22
Dynamic Viewing Functions 1-23
Step 4: Apply/modify constraints and dimensions 1-24
Modifying the dimensions of the sketch 1-27
Repositioning Dimensions 1-29
Step 5: Completing the Base Solid Feature 1-30
The Third Dynamic Viewing Function 1-31
Display Modes: Wireframe, Shaded, Hidden Edge, No Hidden 1-32
Step 6: Adding additional features 1-33
Creating a CUT Feature 1-38
Creating the 2D Section of the CUT Feature 1-39
Save the Part and Exit 1-42
Questions 1-43
Exercises 1-44

**Chapter 2**
**Truss Elements in Two-Dimensional Spaces**

Introduction 2-2
Truss Elements in Two-Dimensional Spaces 2-2
Coordinate Transformation 2-5
EXAMPLE 2.1 2-9
Solution 2-10
Global Stiffness Matrix 2-10
EXAMPLE 2.2 2-13
Solution 2-13
Questions 2-19
Exercises 2-20

**Chapter 3**
**2D Trusses in MS Excel and the Truss Solver**

Direct Stiffness Matrix method using Excel 3-2
EXAMPLE 3.1 3-2
Establish the Global K matrix for each member 3-3
Assembly of the Overall Global Stiffness Matrix 3-8
Solving the Global Displacements 3-10
Calculating Reaction Forces 3-16
Determining the Stresses in Elements 3-18
The Truss Solver and the Truss View programs 3-23

The Truss View program 3-30
Questions 3-32
Exercises 3-33

## Chapter 4
## Pro/MECHANICA Two-Dimensional Truss Analysis

Finite Element Analysis Procedure 4-2
Preliminary Analysis 4-3
Starting *Pro/ENGINEER* 4-4
Units and Basic Datum Geometry Setups 4-5
The Integrated mode of Pro/MECHANICA 4-8
Creating Datum Points as FEA Nodes 4-10
Select and Examine the Element Material Property 4-13
Setting Up an Element Cross Section 4-15
Setting Up an Element Releases 4-16
Creating Elements 4-17
Beam Action Coordinate System (BACS) 4-18
Applying Boundary Conditions - Constraints and Loads 4-21
Applying External Loads 4-23
Running the Solver 4-25
Viewing the results 4-27
Dynamic Query 4-28
Questions 4-30
Exercises 4-31

## Chapter 5
## Three-Dimensional Truss Analysis

Three-Dimensional Coordinate Transformation Matrix 5-2
Stiffness Matrix 5-3
Degrees of Freedom 5-3
Problem Statement 5-5
Preliminary Analysis 5-5
Starting *Pro/ENGINEER* 5-7
Creating a New Template 5-8
Saving the New Template 5-10
The Integrated mode of Pro/MECHANICA 5-11
Creating 3D Datum Points 5-12
Select and Examine the Element Material Property 5-14
Setting Up an Element Cross Section 5-15
Setting Up Beam Element Releases 5-16
Creating Elements 5-17
Applying Boundary Conditions - Constraints and Loads 5-20
Applying External Loads 5-21

Running the Solver 5-22
Viewing the results 5-25
Questions 5-28
Exercises 5-29

## Chapter 6
## Basic Beam Analysis

Introduction 6-2
Modeling Considerations 6-2
Problem Statement 6-3
Preliminary Analysis 6-3
Starting *Pro/ENGINEER* 6-6
New Template Setup 6-7
Setup the *Isometric* View 6-8
Saving as a New Template 6-11
The Integrated mode of Pro/MECHANICA 6-12
Creating Datum Points 6-13
Creating a Datum Curve for the Distributed Load 6-16
Select and Examine the Element Material Property 6-18
Setting Up an Element Cross Section 6-20
Creating Elements 6-21
Applying Boundary Conditions - Constraints and Loads 6-22
Applying External Loads 6-24
Running the Solver 6-27
Viewing the results 6-29
What Went Wrong? 6-31
Running the Solver 6-34
Viewing the results 6-35
- Bending Stress 6-37
- Shear Diagram 6-38
- Moment Diagram 6-40

Refine the FE Model 6-41
Questions 6-44
Exercises 6-45

## Chapter 7
## Beam Analysis Tools

Introduction 7-2
Problem Statement 7-2
Preliminary Analysis 7-3
Stress Components 7-4
Starting *Pro/ENGINEER* 7-6
The Integrated mode of Pro/MECHANICA 7-7

Creating Datum Points 7-8
Creating a Datum Curve 7-11
Select and Examine the Element Material Property 7-13
Setting Up an Element Cross Section 7-14
Creating Elements 7-15
Applying the First Displacement Constraint 7-16
Create a New Coordinate System for the second Support 7-17
Applying the First Displacement Constraint 7-19
Applying External Loads 7-20
Running the Solver 7-22
Viewing the results 7-24
Shear and Moment Diagrams 7-26
Refine the FE Model 7-28
Questions 7-31
Exercises 7-32

**Chapter 8**
**Statically Indeterminate Structures**

Introduction 8-2
Problem Statement 8-2
Preliminary Analysis 8-3
Starting *Pro/ENGINEER* 8-6
Creating a Wireframe Model in Pro/ENGINEER 8-7
A CAD Model is NOT an FEA Model 8-9
The Integrated mode of Pro/MECHANICA 8-10
Applying the Displacement Constraints 8-11
Applying External Loads 8-13
Select and Examine the Element Material Property 8-14
Setting Up an Element Cross Section 8-15
Creating Elements 8-16
Running the Solver 8-17
Viewing the results 8-19
Shear Diagram 8-20
Moment Diagram 8-23
Bending Stress 8-24
Questions 8-25
Exercises 8-26

**Chapter 9**
**Two Dimensional Solid Elements**

Introduction 9-2
Problem Statement 9-3
Preliminary Analysis 9-3

- Maximum Normal Stress 9-3
- Maximum Displacement 9-4

Geometric Considerations of Finite Elements 9-5
Starting *Pro/ENGINEER* 9-6
Creating a CAD Model in Pro/ENGINEER 9-7
Select and Examine the Element Material Property 9-11
The Integrated mode of Pro/MECHANICA 9-12
Applying Boundary Conditions - Constraints 9-14
Applying External Loads 9-16
FEA Surface Idealization 9-17
H-Element versus P-Element 9-18
Create the 2D Mesh 9-19
Running the Solver 9-20
Viewing the results 9-22
➢ Maximum Principal Stress 9-22
➢ X Displacement 9-23
Refinement of the P-mesh 9-24
Running the Solver 9-27
Viewing the results 9-29
➢ Maximum Principal Stress 9-29
➢ X Displacement 9-30
Questions 9-31
Exercises 9-32

**Chapter 10**
## Three-Dimensional Solid Elements

Introduction 10-2
Problem Statement 10-3
Preliminary Analysis 10-4
Starting *Pro/ENGINEER* 10-7
Creating a CAD Model in Pro/ENGINEER 10-8
➢ Define the Sweep Trajectory 10-8
➢ Define the Sweep Section 10-12
Select and Examine the Element Material Property 10-15
The Integrated mode of Pro/MECHANICA 10-16
Applying Boundary Conditions - Constraints 10-17
Applying External Loads 10-18
Create the 2D Mesh 10-19
Running the Solver 10-20
Viewing the results 10-23
➢ Von Mises Stress 10-23
➢ Viewing with the Cutting/Capping option 10-24
Notes on FEA Linear Static Analyses 10-26
Questions 10-27
Exercises 10-28

## Chapter 11
## Axisymmetric and Thin Shell Elements

Introduction 11-2
Problem Statement 11-4
Preliminary Analysis 11-4
Starting *Pro/ENGINEER* 11-6
Creating a CAD Model in Pro/ENGINEER 11-7
Select and Examine the Element Material Property 11-9
The Integrated mode of Pro/MECHANICA 11-10
Applying Boundary Conditions - Constraints 11-12
Applying the pressure 11-14
Create the 2D Mesh 11-15
Running the Solver 11-16
Viewing the Von Mises Stress 11-19
Perform a 3D Shell Analysis 11-20
Applying Boundary Conditions - Constraints 11-23
Applying the pressure 11-25
Create the 3D Shell Mesh 11-26
Running the Solver 11-27
Viewing the Von Mises Stress 11-29
Perform a 3D Solid Analysis 11-30
Create the 3D Solid Mesh 11-32
Running the Solver 11-34
Viewing the Von Mises Stress 11-36
Viewing Multiple Analyses Results 11-37
Questions 11-38
Exercises 11-39

## Chapter 12
## Dynamic Modal Analysis

Introduction 12-2
Problem Statement 12-3
Preliminary Analysis 12-3
The Cantilever Beam Modal Analysis program 12-6
Starting *Pro/ENGINEER* 12-9
Creating a CAD Model in Pro/ENGINEER 12-10
Select and Examine the Element Material Property 12-12
The Integrated mode of Pro/MECHANICA 12-13
Applying Boundary Conditions - Constraints 12-14
Create the 3D Mesh 12-15
Running the Solver 12-16
Viewing the Von Mises Stress 12-19
Adding an additional mass to the system 12-22
Conclusions 12-26

Questions 12-27
Exercises 12-28

## Index