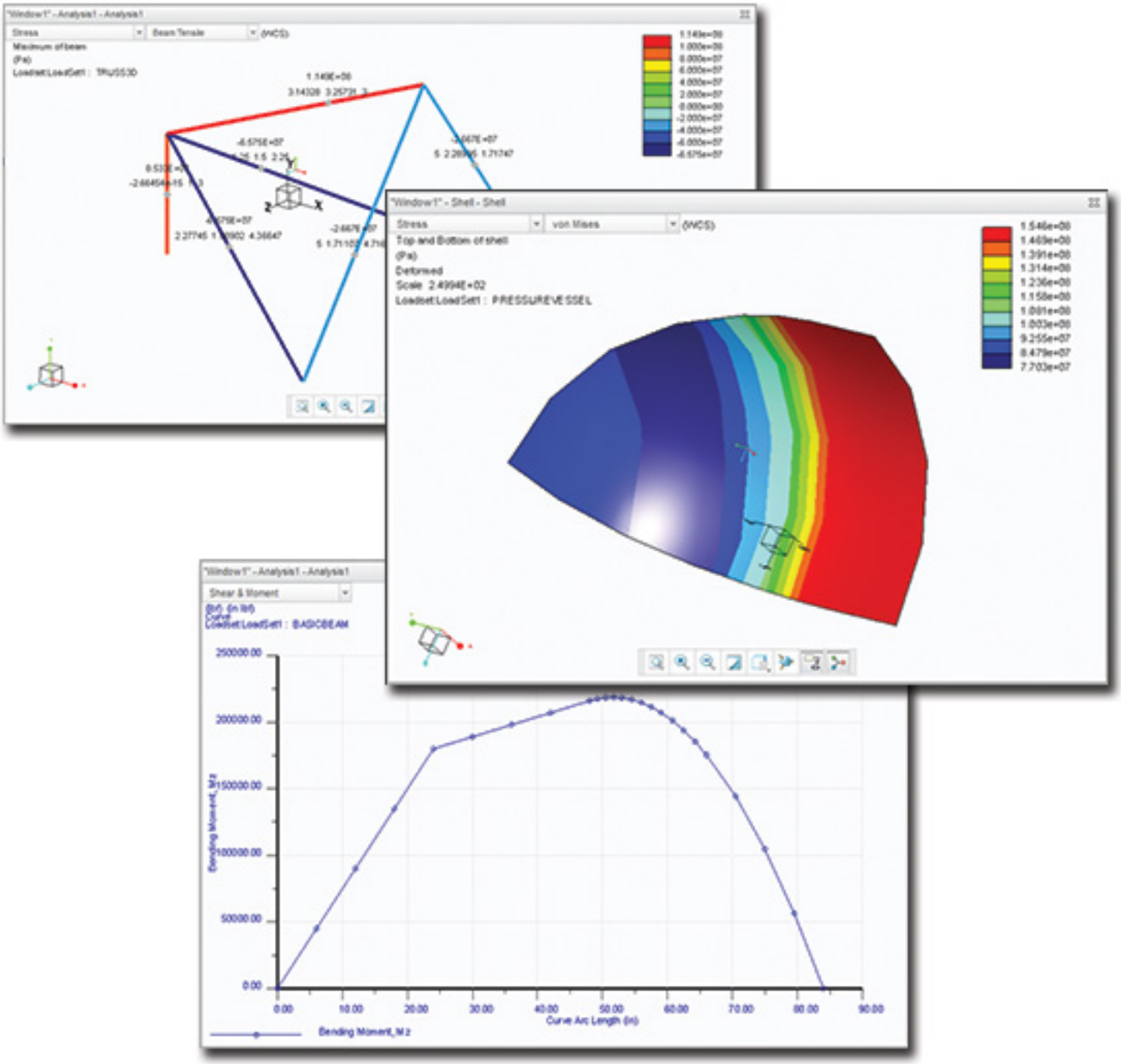
Introduction to Finite Element Analysis

Using Creo™ Simulate 3.0



Randy H. Shih

SDC PUBLICATIONS

Better Textbooks. Lower Prices.
www.SDCpublications.com

Visit the following websites to learn more about this book:

Table of Contents

Preface
Acknowledgments

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 *Creo Parametric* Intro-9
Starting *Creo Parametric* Intro-9
Creo Parametric Screen Layout Intro-10
- Ribbon Toolbar Intro-11
- Quick Access toolbar Intro-11
- Message area Intro-11
- Graphics Display area Intro-11
- Navigator Intro-12
- Web Browser Intro-12
- Navigator Display Controls Intro-12

Basic Functions of Mouse Buttons Intro-13
Model Tree Window and Feature Toolbars Intro-14
On-Line Help Intro-15
Leaving *Creo Parametric* Intro-16
Creating a CAD files folder Intro-16

Chapter 1
The Direct Stiffness Method

Introduction 1-2
One-dimensional Truss Element 1-3
Example 1.1 1-5
Example 1.2 1-7
Basic Solid Modeling using *Creo Parametric* 1-10
The Adjuster Design 1-10
Starting *Creo Parametric* 1-10
Step 1: Units and Basic Datum Geometry Setups 1-12
Units Setup 1-12
Adding the First Part Features — Datum Planes 1-14
Switching On the Plane Tag Display 1-15

Step 2: Determine/Setup the Base Solid Feature 1-16
Sketching Plane – It is an XY CRT, but an XYZ World 1-17
Defining the Sketching Plane 1-18
Defining the Orientation of the Sketching Plane 1-19
Step 3: Creating 2D Rough Sketches 1-21
Shape Before Size – Creating Rough Sketches 1-21
The *Creo Parametric* SKETCHER and INTENT MANAGER 1-22
Graphics Cursors 1-23
Geometric Constraint Symbols 1-24
Dynamic Viewing Functions 1-25
Step 4: Apply/Modify Constraints and Dimensions 1-26
Modifying the Dimensions in a Sketch 1-29
Repositioning Dimensions 1-31
Step 5: Completing the Base Solid Feature 1-32
The Third Dynamic Viewing Function 1-33
Display Modes: Wireframe, Shaded, Hidden Edge, No Hidden 1-34
Step 6: Adding Additional Features 1-35
Creating a CUT Feature 1-40
Creating the 2D Section of the CUT Feature 1-41
Create another CUT Feature 1-44
Delete/Select Sketch References 1-45
Create the 2D Section 1-47
Save the Part 1-49
Review Questions 1-50
Exercises 1-51

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
Review 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
Solve for the Global Displacements 3-10
Calculate the Reaction Forces 3-16
Determine the Stresses in Elements 3-18
The Truss Solver and the Truss View Programs 3-23
The Truss View program 3-30
Review Questions 3-32
Exercises 3-33

Chapter 4
Creo Simulate Two-Dimensional Truss Analysis

Finite Element Analysis Procedure 4-2
Preliminary Analysis 4-3
Starting *Creo Parametric* 4-4
Units and Basic Datum Geometry Setups 4-5
Adding the First Part Features – Datum Planes 4-7
The Integrated Mode of *Creo Simulate* 4-8
Create Datum Points as FEA Nodes 4-9
Setup an Element Cross Section 4-13
Setup Beam Element Releases 4-14
Select and Examine the Element Material Property 4-15
Create Elements 4-16
Beam Action Coordinate System (BACS) 4-17
Apply Boundary Conditions - Constraints and Loads 4-20
Apply External Loads 4-22
Run the Solver 4-24
View the Results 4-26
Dynamic Query 4-27
Review Questions 4-29
Exercises 4-30

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 *Creo Parametric* 5-7
Create a New Template 5-8
Save the Current Setup as a New Template 5-10
The Integrated mode of *Creo Simulate* 5-11

Create 3D Datum Points 5-11
Setting Up an Element Cross Section 5-13
Setting Up Beam Element Releases 5-14
Select and Examine the Element Material Property 5-15
Create 2D Truss Elements 5-16
Apply Boundary Conditions - Constraints and Loads 5-19
Apply External Loads 5-20
Run the FEA Solver 5-21
View the FEA Results 5-24
Review Questions 5-27
Exercises 5-28

Chapter 6
Basic Beam Analysis

Introduction 6-2
Modeling Considerations 6-2
Problem Statement 6-3
Preliminary Analysis 6-3
Starting *Creo Parametric* 6-6
New Template Setup 6-7
Setup an *Isometric* View 6-8
Save Current Setup as a New Template 6-11
The Integrated Mode of *Creo Simulate* 6-12
Create a Datum Curve for the Distributed Load 6-15
Setup an Element Cross Section 6-17
Select and Examine the Element Material Property 6-18
Create 3D Elements 6-19
Apply Boundary Conditions - Constraints 6-20
Apply External Loads 6-22
Run the FEA Solver 6-25
View the FEA Results 6-27
What Went Wrong? 6-29
Run the Solver 6-32
View the FEA Results 6-33
- Bending Stress 6-35
- Shear Diagram 6-36
- Moment Diagram 6-38

Refine the FE Model 6-39
- Moment Diagram 6-45
- Shear Diagram 6-46

Review Questions 6-47
Exercises 6-48

Chapter 7
Beam Analysis Tools

Introduction	7-2
Problem Statement	7-2
Preliminary Analysis	7-3
Stress Components	7-4
Starting *Creo Parametric*	7-6
The Integrated mode of *Creo Simulate*	7-7
Create Datum Curves	7-10
Select and Examine the Element Material Property	7-12
Setup the Element Cross Section	7-13
Create Beam Elements	7-14
Apply the First Displacement Constraint	7-15
Create a New Coordinate System for the Second Support	7-16
Apply the First Displacement Constraint	7-18
Apply the External Loads	7-19
Run the FEA Solver	7-21
View the FEA Results	7-23
Shear and Moment Diagrams	7-25
Refine the FE Model	7-27
Review Questions	7-30
Exercises	7-31

Chapter 8
Statically Indeterminate Structures

Introduction	8-2
Problem Statement	8-3
Preliminary Analysis	8-3
Starting *Creo Parametric*	8-6
Create a Wireframe Model in *Creo Parametric*	8-7
A CAD Model is NOT an FEA Model	8-9
The Integrated Mode of *Creo Simulate*	8-10
Select and Examine the Element Material Property	8-10
Setup the Element Cross Section	8-11
Create the Beam Elements	8-12
Apply the Displacement Constraints	8-13
Apply the External Loads	8-15
Run the FEA Solver	8-16
View the FEA Results	8-18
Shear Diagram	8-20
Moment Diagram	8-22
Bending Stress	8-23
Review Questions	8-24
Exercises	8-25

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 *Creo Parametric* 9-6
Create a CAD Model in *Creo Parametric* 9-7
Select and Examine the Part Material Property 9-11
The Integrated Mode of *Creo Simulate* 9-12
Apply the Boundary Conditions - Constraints 9-14
Apply the External Loads 9-16
FEA Surface Idealization 9-17
H-Element versus P-Element 9-18
Create the 2D Mesh 9-19
Run the FEA Solver 9-20
View the FEA Results 9-22
- Maximum Principal Stress 9-22
- X Displacement 9-23

Refinement of the P-mesh 9-24
Run the FEA Solver 9-27
View the FEA Results 9-29
- Maximum Principal Stress 9-29
- X Displacement 9-30

Review Questions 9-31
Exercises 9-32

Chapter 10
Three-Dimensional Solid Elements

Introduction 10-2
Problem Statement 10-3
Preliminary Analysis 10-4
Starting *Creo Parametric* 10-7
Create a CAD Model in *Creo Parametric* 10-8
- Define the Sweep Trajectory 10-8

The Sweep Command 10-11
- Define the Sweep Section 10-11

Select and Examine the Element Material Property 10-14
The Integrated Mode of *Creo Simulate* 10-15
Apply the Boundary Conditions - Constraints 10-16
Apply the External Loads 10-17

Create the 3D Mesh 10-18
Run the FEA Solver 10-19
View the FEA Results 10-22
- Von Mises Stress 10-22
- Viewing with the Cutting/Capping Option 10-23

Notes on FEA Linear Static Analyses 10-26
Review 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 *Creo Parametric* 11-6
Create a CAD Model in *Creo Parametric* 11-7
Select and Examine the Element Material Property 11-9
The Integrated Mode of *Creo Simulate* 11-10
Apply the Boundary Conditions - Constraints 11-12
Apply the Internal Pressure 11-14
Create the 2D Mesh 11-15
Run the FEA Solver 11-16
View the Von Mises Stress 11-19
Perform a 3D Shell Analysis 11-20
Apply the Boundary Conditions - Constraints 11-22
Apply the Internal Pressure 11-25
Create the 3D Shell Mesh 11-26
Run the FEA Solver 11-27
View the Von Mises Stress 11-29
Perform a 3D Solid Analysis 11-30
Create the 3D Solid Mesh 11-32
Run the FEA Solver 11-34
View the Von Mises Stress 11-36
View Multiple Analyses Results 11-37
Review 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 *Creo Parametric* 12-9
Create a CAD Model in *Creo Parametric* 12-10
Select and Examine the Element Material Property 12-12
The Integrated Mode of *Creo Simulate* 12-13
Apply the Boundary Conditions - Constraints 12-14
Create the 3D Mesh 12-15
Run the FEA Solver 12-16
View the FEA Results 12-19
Adding an Additional Mass to the System 12-22
Conclusions 12-26
Review Questions 12-27
Exercises 12-28

Index