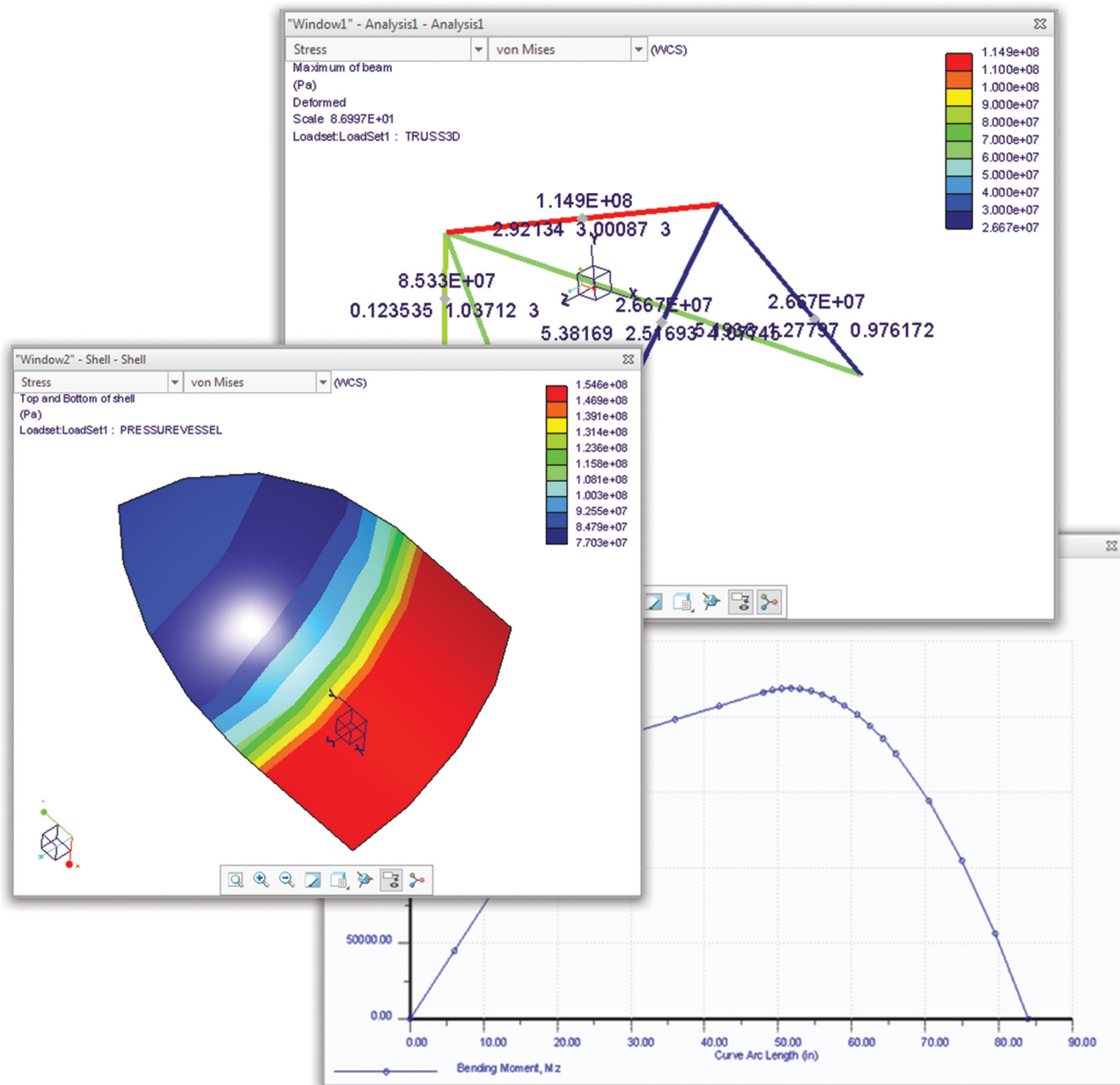


Introduction to Finite Element Analysis Using Creo™ Simulate 8.0



Randy H. Shih

Visit the following websites to learn more about this book:



[amazon.com](https://www.amazon.com)

[Google books](https://books.google.com)

[BARNES & NOBLE](https://www.barnesandnoble.com)

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
Online 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/Off the Plane Tag Display	1-15
Step 2: Determine/Set up 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 Sketching 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
Set up an Element Cross Section	4-13
Set up Beam Element Releases	4-14
Select and Examine the Element Material Property	4-15
Create Elements	4-18
Beam Action Coordinate System (BACS)	4-19
Apply Boundary Conditions - Constraints and Loads	4-22
Apply External Loads	4-24
Run the Solver	4-26
View the Results	4-28
Dynamic Query	4-29
Review Questions	4-31
Exercises	4-32

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
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
Set up 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
Set up 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
Reactions at supports	6-33
Bending Stress	6-35
Shear Diagram	6-36
Moment Diagram	6-38
Refine the FE Model	6-39
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
Set up 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 Second 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
Set up 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
Reactions at supports	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
Use the Model Setup of Creo Simulate	9-12
Apply the Boundary Conditions - Constraints	9-13
Apply the External Loads	9-15
FEA Surface Idealization	9-16
H-Element versus P-Element	9-17
Create the 2D Mesh	9-18
Run the FEA Solver	9-19
View the FEA Results	9-21
Maximum Principal Stress	9-21
X Displacement	9-22
Refinement of the P-mesh	9-23
Run the FEA Solver	9-26
View the FEA Results	9-28
Maximum Principal Stress	9-28
X Displacement	9-29
Review Questions	9-30
Exercises	9-31

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
Review Questions	10-26
Exercises	10-27

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 Element 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
Notes on FEA Linear Static Analyses	11-38
Review Questions	11-39
Exercises	11-40

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