Introduction to Finite Element Analysis
Using Creo™ Simulate 1.0

Randy H. Shih
Oregon Institute of Technology
Table of Contents

Preface
Acknowledgments

Introduction

Introduction
Development of Finite Element Analysis
FEA Modeling Considerations
Types of Finite Elements
Finite Element Analysis Procedure
Matrix Definitions
Getting Started with Creo Parametric
Starting Creo Parametric
Creo Parametric Screen Layout
  • Ribbon Toolbar
  • Quick Access toolbar
  • Message Area
  • Graphics display area
  • Navigator
  • Web Browser
  • Navigator Display Controls
Basic Functions of Mouse Buttons
Model Tree window and Feature Toolbars
On-Line Help
Leaving Creo Parametric
Creating a CAD files folder

Chapter 1

The Direct Stiffness Method

Introduction
One-dimensional Truss Element
Example 1.1
Example 1.2
Basic Solid Modeling using Creo Parametric
The Adjuster design
Starting Creo Parametric
Step 1: Units and Basic Datum Geometry Setups
Units Setup and Creo Parametric Menu Structure
Adding the First Part Features — Datum Planes
Step 2: Determine/Setup the Base Solid Feature
### Chapter 2

**Truss Elements in Two-Dimensional Spaces**

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction</td>
<td>2-2</td>
</tr>
<tr>
<td>Truss Elements in Two-Dimensional Spaces</td>
<td>2-2</td>
</tr>
<tr>
<td>Coordinate Transformation</td>
<td>2-5</td>
</tr>
<tr>
<td>EXAMPLE 2.1</td>
<td>2-9</td>
</tr>
<tr>
<td>Solution</td>
<td>2-10</td>
</tr>
<tr>
<td>Global Stiffness Matrix</td>
<td>2-10</td>
</tr>
<tr>
<td>EXAMPLE 2.2</td>
<td>2-13</td>
</tr>
<tr>
<td>Solution</td>
<td>2-13</td>
</tr>
<tr>
<td>Questions</td>
<td>2-19</td>
</tr>
<tr>
<td>Exercises</td>
<td>2-20</td>
</tr>
</tbody>
</table>

### Chapter 3

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

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Direct Stiffness Matrix method using Excel</td>
<td>3-2</td>
</tr>
<tr>
<td>EXAMPLE 3.1</td>
<td>3-2</td>
</tr>
<tr>
<td>Establish the Global K matrix for each member</td>
<td>3-3</td>
</tr>
<tr>
<td>Assembly of the Overall Global Stiffness Matrix</td>
<td>3-8</td>
</tr>
<tr>
<td>Solving the Global Displacements</td>
<td>3-10</td>
</tr>
<tr>
<td>Calculating Reaction Forces</td>
<td>3-16</td>
</tr>
</tbody>
</table>
## 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
- The Integrated mode of *Creo Simulate* 4-8
- Creating Datum Points as FEA Nodes 4-9
- Setting Up an Element Cross Section 4-13
- Setting Up an Element Releases 4-14
- Select and Examine the Element Material Property 4-15
- Creating Elements 4-16
- Beam Action Coordinate System (BACS) 4-17
- Applying Boundary Conditions - Constraints and Loads 4-20
- Applying External Loads 4-22
- Running the Solver 4-24
- Viewing the results 4-26
- Dynamic Query 4-27
- 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
- Creating a New Template 5-8
- Saving the New Template 5-10
- The Integrated mode of *Creo Simulate* 5-11
- Creating 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
- Creating Elements 5-16
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 the Isometric View 6-8
Saving as a New Template 6-11
The Integrated mode of Creo Simulate 6-12
Creating a Datum Curve for the Distributed Load 6-15
Setting Up an Element Cross Section 6-17
Select and Examine the Element Material Property 6-18
Creating Elements 6-19
Applying Boundary Conditions - Constraints and Loads 6-20
Applying External Loads 6-22
Running the Solver 6-25
Viewing the results 6-27
What Went Wrong? 6-29
Running the Solver 6-32
Viewing the results 6-33
➤ Bending Stress 6-35
➤ Shear Diagram 6-36
➤ Moment Diagram 6-38
Refine the FE Model 6-39
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
Chapter 8

Statically Indeterminate Structures
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
Creating a CAD Model in Creo Parametric 9-7
Select and Examine the Element Material Property 9-11
The Integrated mode of Creo Simulate 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 Creo Parametric 10-7
Creating a CAD Model in Creo Parametric 10-8
- Define the Sweep Trajectory 10-8
- Define the Sweep Section 10-11
Select and Examine the Element Material Property 10-14
The Integrated mode of Creo Simulate 10-15
Applying Boundary Conditions - Constraints 10-16
Applying External Loads 10-17
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create the 3D Mesh</td>
<td>10-18</td>
</tr>
<tr>
<td>Running the Solver</td>
<td>10-19</td>
</tr>
<tr>
<td>Viewing the results</td>
<td>10-22</td>
</tr>
<tr>
<td>- Von Mises Stress</td>
<td>10-22</td>
</tr>
<tr>
<td>- Viewing with the Cutting/Capping option</td>
<td>10-23</td>
</tr>
<tr>
<td>Notes on FEA Linear Static Analyses</td>
<td>10-26</td>
</tr>
<tr>
<td>Questions</td>
<td>10-27</td>
</tr>
<tr>
<td>Exercises</td>
<td>10-28</td>
</tr>
</tbody>
</table>

**Chapter 11**

**Axisymmetric and Thin Shell Elements**

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction</td>
<td>11-2</td>
</tr>
<tr>
<td>Problem Statement</td>
<td>11-4</td>
</tr>
<tr>
<td>Preliminary Analysis</td>
<td>11-4</td>
</tr>
<tr>
<td>Starting Creo Parametric</td>
<td>11-6</td>
</tr>
<tr>
<td>Creating a CAD Model in Creo Parametric</td>
<td>11-7</td>
</tr>
<tr>
<td>Select and Examine the Element Material Property</td>
<td>11-9</td>
</tr>
<tr>
<td>The Integrated mode of Creo Simulate</td>
<td>11-10</td>
</tr>
<tr>
<td>Applying Boundary Conditions - Constraints</td>
<td>11-12</td>
</tr>
<tr>
<td>Applying the pressure</td>
<td>11-14</td>
</tr>
<tr>
<td>Create the 2D Mesh</td>
<td>11-15</td>
</tr>
<tr>
<td>Running the Solver</td>
<td>11-16</td>
</tr>
<tr>
<td>Viewing the Von Mises Stress</td>
<td>11-19</td>
</tr>
<tr>
<td>Perform a 3D Shell Analysis</td>
<td>11-20</td>
</tr>
<tr>
<td>Applying Boundary Conditions - Constraints</td>
<td>11-22</td>
</tr>
<tr>
<td>Applying the pressure</td>
<td>11-25</td>
</tr>
<tr>
<td>Create the 3D Shell Mesh</td>
<td>11-26</td>
</tr>
<tr>
<td>Running the Solver</td>
<td>11-27</td>
</tr>
<tr>
<td>Viewing the Von Mises Stress</td>
<td>11-29</td>
</tr>
<tr>
<td>Perform a 3D Solid Analysis</td>
<td>11-30</td>
</tr>
<tr>
<td>Create the 3D Solid Mesh</td>
<td>11-32</td>
</tr>
<tr>
<td>Running the Solver</td>
<td>11-34</td>
</tr>
<tr>
<td>Viewing the Von Mises Stress</td>
<td>11-36</td>
</tr>
<tr>
<td>Viewing Multiple Analyses Results</td>
<td>11-37</td>
</tr>
<tr>
<td>Questions</td>
<td>11-38</td>
</tr>
<tr>
<td>Exercises</td>
<td>11-39</td>
</tr>
</tbody>
</table>

**Chapter 12**

**Dynamic Modal Analysis**

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction</td>
<td>12-2</td>
</tr>
<tr>
<td>Problem Statement</td>
<td>12-3</td>
</tr>
<tr>
<td>Preliminary Analysis</td>
<td>12-3</td>
</tr>
</tbody>
</table>
The Cantilever Beam Modal Analysis program 12-6
Starting Creo Parametric 12-9
Creating 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
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