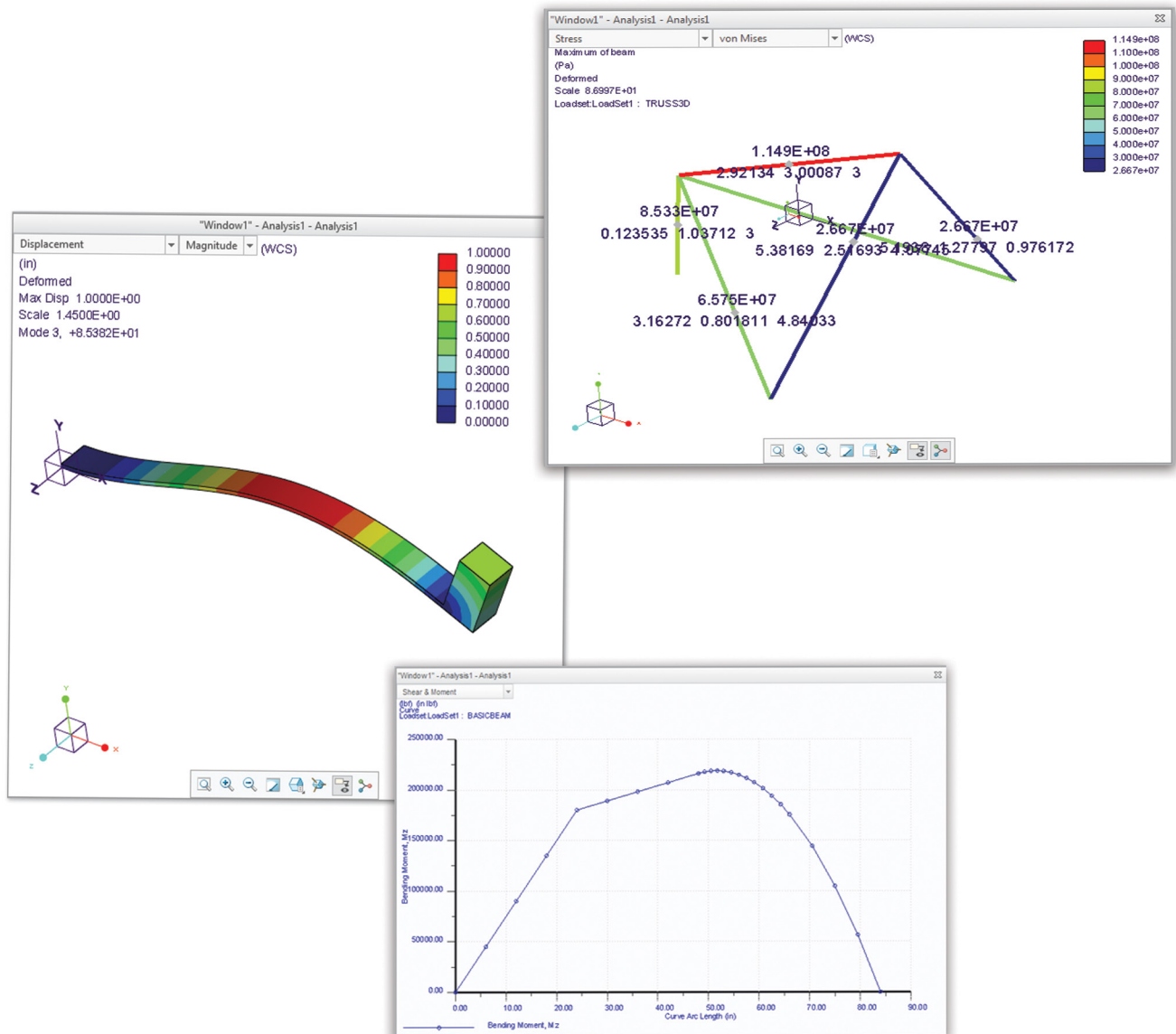


# Introduction to Finite Element Analysis

## Using Creo® Simulate 13



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  |
| 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 and Browser                  | 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 |
| Creo AI assistant                      | Intro-16 |
| Leaving Creo Parametric                | Intro-18 |
| Creating a CAD files folder            | Intro-18 |

## 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                                 | 1-30 |
| Step 5: Completing the Base Solid Feature                | 1-31 |
| The Third Dynamic Viewing Function                       | 1-32 |
| Display Modes: Wireframe, Shaded, Hidden Edge, No Hidden | 1-33 |
| Step 6: Adding Additional Features                       | 1-34 |
| Creating a Cut Feature                                   | 1-39 |
| Creating the 2D Section of the Cut Feature               | 1-40 |
| Create another Cut Feature                               | 1-43 |
| Delete/Select Sketching References                       | 1-44 |
| Create a 2D Section                                      | 1-46 |
| Save the Part                                            | 1-48 |
| Review Questions                                         | 1-49 |
| Exercises                                                | 1-50 |

## **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 |
| Create an Analysis and Run the FEA 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 3D Truss Elements                           | 5-16 |

|                                                   |      |
|---------------------------------------------------|------|
| Apply Boundary Conditions - Constraints and Loads | 5-19 |
| Apply External Loads                              | 5-20 |
| Create an Analysis and 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 Beam 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 |
| Create an Analysis and Run the FEA 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 |
| Create an Analysis and Run the FEA Solver             | 7-21 |
| View the FEA Results                                  | 7-23 |
| Shear and Moment Diagrams                             | 7-25 |
| Create an Analysis and 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 Part 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 Part 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 Part 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**