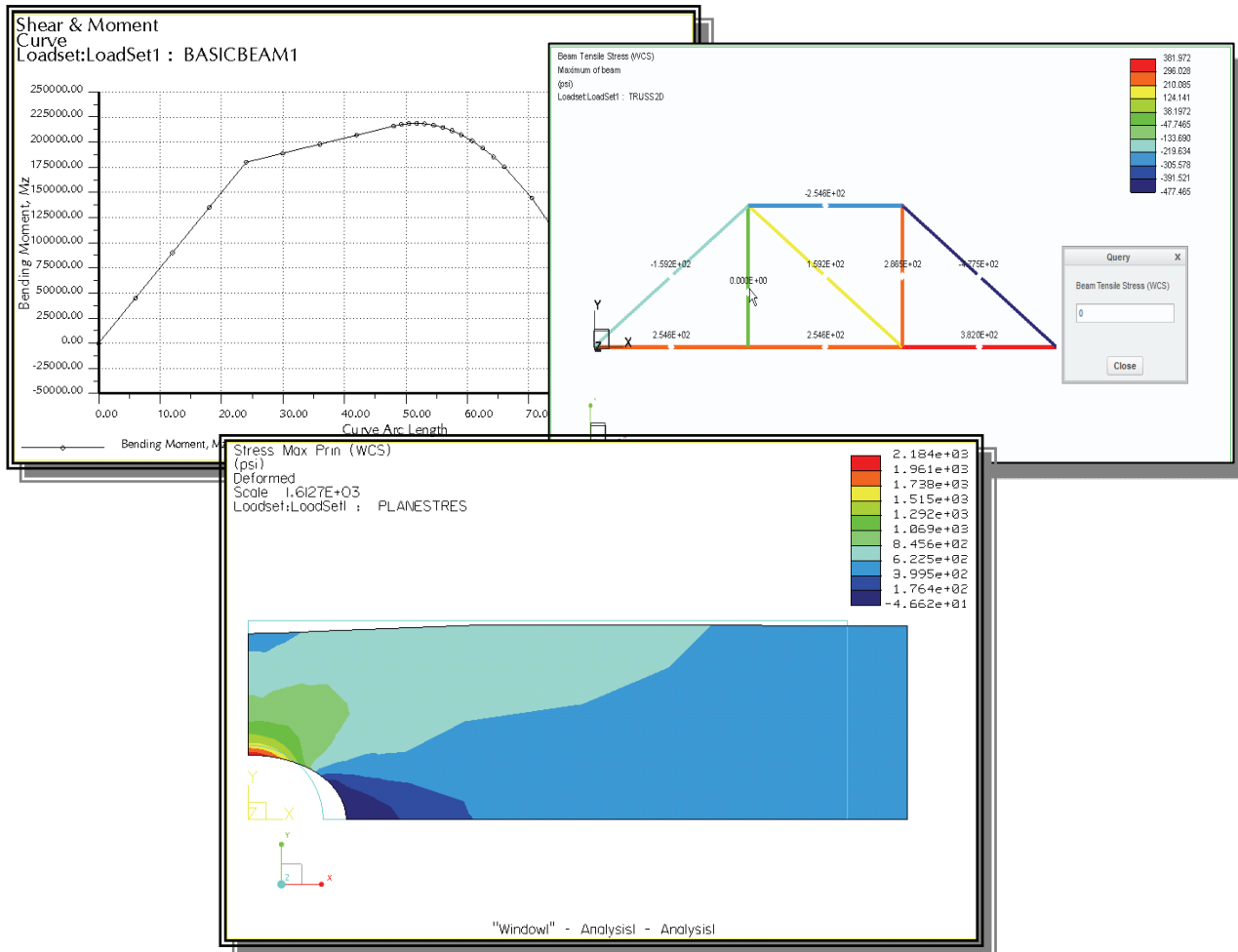


Introduction to Finite Element Analysis Using Creo™ Simulate 1.0



Randy H. Shih
Oregon Institute of Technology

SDC
PUBLICATIONS

www.SDCpublications.com

Schroff Development Corporation

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 <i>Creo Parametric</i>	Intro-9
Starting <i>Creo Parametric</i>	Intro-9
<i>Creo Parametric</i> 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 <i>Creo Parametric</i>	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 <i>Creo Parametric</i>	1-10
The Adjuster design	1-10
Starting <i>Creo Parametric</i>	1-10
Step 1: Units and Basic Datum Geometry Setups	1-12
Units Setup and <i>Creo Parametric</i> Menu Structure	1-12
Adding the First Part Features — Datum Planes	1-14
Step 2: Determine/Setup the Base Solid Feature	1-15

Sketching plane – It is an XY CRT, but an XYZ World	1-15
Defining the Sketching Plane	1-16
Defining the Orientation of the Sketching Plane	1-18
Step 3: Creating 2D rough sketches	1-20
Shape Before Size – Creating Rough Sketches	1-20
The <i>Creo Parametric</i> SKETCHER and INTENT MANAGER	1-21
Graphics Cursors	1-22
Geometric Constraint Symbols	1-23
Dynamic Viewing Functions	1-24
Step 4: Apply/modify constraints and dimensions	1-25
Modifying the dimensions of the sketch	1-28
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	1-36
Creating a CUT Feature	1-39
Creating the 2D Section of the CUT Feature	1-40
Save the Part and Exit	1-43
Questions	1-44
Exercises	1-45

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

Creo Simulate Two-Dimensional Truss Analysis

Finite Element Analysis Procedure	4-2
Preliminary Analysis	4-3
Starting <i>Creo Parametric</i>	4-4
Units and Basic Datum Geometry Setups	4-5
The Integrated mode of <i>Creo Simulate</i>	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 <i>Creo Parametric</i>	5-7
Creating a New Template	5-8
Saving the New Template	5-10
The Integrated mode of <i>Creo Simulate</i>	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

Applying Boundary Conditions - Constraints and Loads	5-19
Applying External Loads	5-20
Running the Solver	5-21
Viewing the results	5-24
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 <i>Creo Parametric</i>	6-6
New Template Setup	6-7
Setup the <i>Isometric</i> View	6-8
Saving as a New Template	6-11
The Integrated mode of <i>Creo Simulate</i>	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

Starting <i>Creo Parametric</i>	7-6
The Integrated mode of <i>Creo Simulate</i>	7-7
Creating a Datum Curve	7-10
Select and Examine the Element Material Property	7-12
Setting Up an Element Cross Section	7-13
Creating Elements	7-14
Applying the First Displacement Constraint	7-15
Create a New Coordinate System for the second Support	7-16
Applying the First Displacement Constraint	7-18
Applying External Loads	7-19
Running the Solver	7-21
Viewing the results	7-23
Shear and Moment Diagrams	7-25
Refine the FE Model	7-27
Questions	7-30
Exercises	7-31

Chapter 8

Statically Indeterminate Structures

Introduction	8-2
Problem Statement	8-3
Preliminary Analysis	8-3
Starting <i>Creo Parametric</i>	8-6
Creating a Wireframe Model in <i>Creo Parametric</i>	8-7
A CAD Model is NOT an FEA Model	8-9
The Integrated mode of <i>Creo Simulate</i>	8-10
Select and Examine the Element Material Property	8-10
Setting Up an Element Cross Section	8-11
Creating Elements	8-12
Applying the Displacement Constraints	8-13
Applying External Loads	8-15
Running the Solver	8-16
Viewing the results	8-18
Shear Diagram	8-20
Moment Diagram	8-22
Bending Stress	8-23
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 <i>Creo Parametric</i>	9-6
Creating a CAD Model in <i>Creo Parametric</i>	9-7
Select and Examine the Element Material Property	9-11
The Integrated mode of <i>Creo Simulate</i>	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 <i>Creo Parametric</i>	10-7
Creating a CAD Model in <i>Creo Parametric</i>	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 <i>Creo Simulate</i>	10-15
Applying Boundary Conditions - Constraints	10-16
Applying External Loads	10-17

Create the 3D Mesh	10-18
Running the Solver	10-19
Viewing the results	10-22
➤ Von Mises Stress	10-22
➤ Viewing with the Cutting/Capping option	10-23
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 <i>Creo Parametric</i>	11-6
Creating a CAD Model in <i>Creo Parametric</i>	11-7
Select and Examine the Element Material Property	11-9
The Integrated mode of <i>Creo Simulate</i>	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	11-22
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 <i>Creo Parametric</i>	12-9
Creating a CAD Model in <i>Creo Parametric</i>	12-10
Select and Examine the Element Material Property	12-12
The Integrated mode of <i>Creo Simulate</i>	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