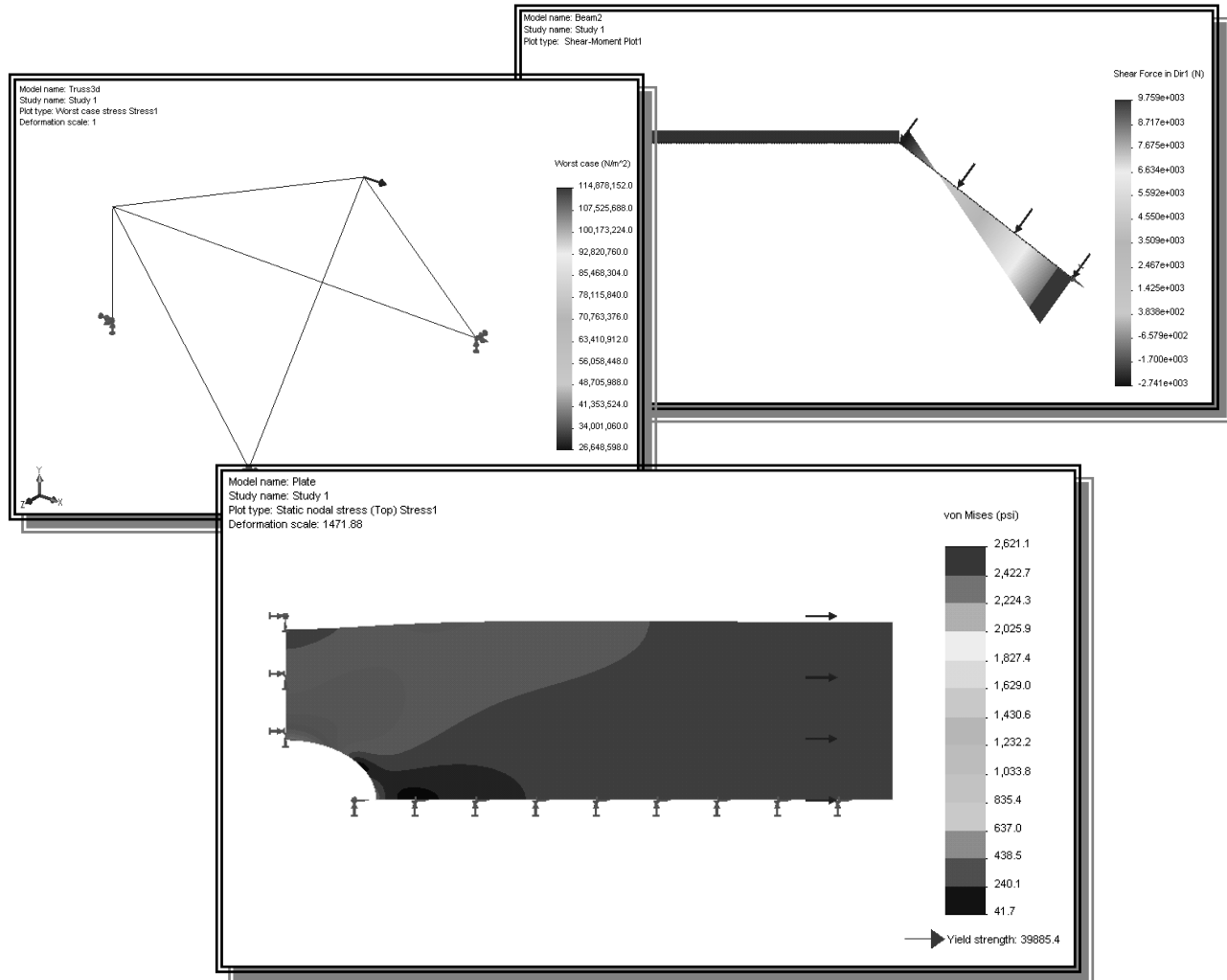


Introduction to Finite Element Analysis Using SolidWorks Simulation 2010[®]



Randy H. Shih
Oregon Institute of Technology

SDC

PUBLICATIONS

Schroff Development Corporation

www.schroff.com

Table of Contents

Preface	i
Acknowledgments	ii
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>SolidWorks</i>	Intro-9
Starting <i>SolidWorks</i>	Intro-9
Units Setup	Intro-11
<i>SolidWorks</i> Screen Layout	Intro-13
• Menu Bar Toolbar	Intro-13
• Menu Bar Pull-down menus	Intro-14
• Heads-up View Toolbar	Intro-14
• Features Toolbar	Intro-14
• Sketch Toolbar	Intro-14
• Command Manager	Intro-15
• Feature Manager Design Tree	Intro-16
• Graphics Area	Intro-17
• Reference Triad	Intro-17
• Origin	Intro-17
• Confirmation Corner	Intro-17
• Graphics Cursor or Crosshairs	Intro-17
• Message and Status Bar	Intro-17
Basic Functions of Mouse Buttons	Intro-18
[Esc] – Canceling Commands	Intro-18
On-Line Help	Intro-19
Leaving <i>SolidWorks</i>	Intro-19
Creating a CAD files folder	Intro-20
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 SolidWorks	1-10
The Adjuster design	1-10
Starting SolidWorks	1-10
Step 1: Creating a Rough Sketch	1-12
Graphics Cursors	1-12
Geometric Relation Symbols	1-14
Step 2: Apply/Modify Relations and Dimensions	1-15
Changing the Dimension Standard	1-16
Viewing Functions – <i>Zoom</i> and <i>Pan</i>	1-17
Modifying the Dimensions of the Sketch	1-18
Step 3: Completing the Base Solid Feature	1-19
Isometric View	1-20
Rotation of the 3D Model – <i>Rotate View</i>	1-20
Rotation and Panning – <i>Arrow Keys</i>	1-22
Dynamic Viewing – Quick Keys	1-23
Viewing Tools – Heads-up View Toolbar	1-25
View Orientation	1-26
Display Style	1-27
Orthographic vs. Perspective	1-27
Customizing the Heads-up View Toolbar	1-27
Sketch Plane	1-28
Step 4-1: Adding an Extruded Boss Feature	1-30
Step 4-2: Adding an Extruded Cut Feature	1-33
Save the Part and Exit	1-35
Questions	1-36
Exercises	1-37

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

Truss Elements in SolidWorks Simulation

One-dimensional Line Elements	4-2
Starting <i>SolidWorks</i>	4-4
Units Setup	4-5
Creating the CAD Model – Solid Modeling approach	4-6
A CAD Model is NOT an FEA Model	4-15
The SolidWorks Simulation Module	4-16
Creating an FEA Model	4-17
Assign the Element Material Property	4-19
Applying Boundary Conditions - Constraints	4-20
Applying External Loads	4-23
Create the FEA Mesh and Run the Solver	4-25
Viewing the Stress results	4-26
Viewing the Displacement results	4-28
Questions	4-29
Exercises	4-30

Chapter 5

SolidWorks Simulation Two-Dimensional Truss Analysis

Finite Element Analysis Procedure	5-2
Preliminary Analysis	5-3
Starting <i>SolidWorks</i>	5-4
Units Setup	5-5
Creating the CAD Model – Structural Member approach	5-6
Creating Structural Members in SolidWorks	5-8
Weldment Profiles	5-9
Activate the SolidWorks Simulation Module	5-12
Setting Up Truss Elements	5-14

Assign the Element Material Property	5-15
Applying Boundary Conditions - Constraints and Loads	5-16
Applying External Loads	5-21
Create the FEA Mesh and Run the Solver	5-23
Viewing the Stress results	5-24
Viewing the Internal Loads of All members	5-26
Questions	5-27
Exercises	5-28

Chapter 6

Three-Dimensional Truss Analysis

Three-Dimensional Coordinate Transformation Matrix	6-2
Stiffness Matrix	6-3
Degrees of Freedom	6-3
Problem Statement	6-5
Preliminary Analysis	6-5
Starting <i>SolidWorks</i>	6-7
Units Setup	6-8
Creating the CAD Model – Structural Member approach	6-9
<i>Creating Weldments Profiles in SolidWorks</i>	6-12
Creating Structural Members using the new profile	6-16
Editing the Dimensions of the new profile	6-18
Activate the SolidWorks Simulation Module	6-19
Setting Up Truss Elements	6-21
Assign the Element Material Property	6-22
Applying Boundary Conditions - Constraints	6-23
Applying External Loads	6-24
Create the FEA Mesh and Run the Solver	6-26
Using the Probe Option to View individual Stress	6-27
Viewing the Internal Loads of All members	6-28
Questions	6-29
Exercises	6-30

Chapter 7

Basic Beam Analysis

Introduction	7-2
Modeling Considerations	7-2
Problem Statement	7-3
Preliminary Analysis	7-3
Starting <i>SolidWorks</i>	7-6
Units Setup	7-7
Creating the CAD Model – Structural Member approach	7-8

Creating a rectangular Weldment Profile	7-10
Creating Structural Members using the new profile	7-14
Adjusting the Orientation of the profile	7-15
Adding a Datum Point for the Concentrated Load	7-16
Activate the SolidWorks Simulation Module	7-18
Assign the Element Material Property	7-20
Applying Boundary Conditions - Constraints	7-21
Applying Concentrated Point Loads	7-24
Applying the Distributed Load	7-26
Create the FEA Mesh and Run the Solver	7-28
What Went Wrong?	7-29
Directions 1 and 2 in Shear and Moment Diagrams	7-32
Questions	7-34
Exercises	7-35

Chapter 8

Beam Analysis Tools

Introduction	8-2
Problem Statement	8-2
Preliminary Analysis	8-3
Stress Components	8-4
Starting <i>SolidWorks</i>	8-6
Units Setup	8-7
Creating the CAD Model – Structural Member approach	8-8
Creating a rectangular Weldment Profile	8-10
Creating Structural Members using the new profile	8-14
Adjusting the Orientation of the profile	8-15
Adding a Datum Point	8-16
Activate the SolidWorks Simulation Module	8-18
Assign the Element Material Property	8-20
Applying Boundary Conditions - Constraints	8-21
Applying the Distributed Load	8-24
Create the FEA Mesh and Run the Solver	8-26
Shear and Moment Diagrams	8-27
Using the Probe Option to Examine Stress at Point 1	8-29
Questions	8-30
Exercises	8-31

Chapter 9

Statically Indeterminate Structures

Introduction	9-2
Problem Statement	9-3

Preliminary Analysis	9-3
Starting <i>SolidWorks</i>	9-6
Units Setup	9-7
Creating the CAD Model	9-8
Creating a circular Weldment Profile	9-10
Creating Structural Members using the new profile	9-14
Adding a Datum Point for the Concentrated Load	9-15
Activate the SolidWorks Simulation Module	9-17
Assign the Element Material Property	9-19
Applying Boundary Conditions - Constraints	9-20
Applying the Concentrated Point Load	9-24
Create the FEA Mesh and Run the Solver	9-25
Viewing the Internal Loads of All members	9-26
Shear and Moment Diagrams	9-27
Questions	9-29
Exercises	9-30

Chapter 10

Two Dimensional Surface Analysis

Introduction	10-2
Problem Statement	10-3
Preliminary Analysis	10-3
• Maximum Normal Stress	10-3
• Maximum Displacement	10-4
Geometric Considerations of Finite Elements	10-5
Starting <i>SolidWorks</i>	10-6
Units Setup	10-7
Creating a CAD Model in SolidWorks	10-8
Activate the SolidWorks Simulation Module	10-11
Defining A Surface Model	10-13
Assign the Element Material Property	10-14
Applying Boundary Conditions - Constraints	10-15
Applying the External Load	10-18
H-Element versus P-Element	10-21
Create the first 2D Mesh –Coarse Mesh	10-22
Run the Solver	10-24
Refinement of the Mesh– Global Element Size 0.10	10-26
Refinement of the Mesh– Global Element Size 0.05	10-28
Refinement of the Mesh– Global Element Size 0.03	10-30
Refinement of the Mesh– Global Element Size 0.02	10-31
Comparison of Results	10-32
Questions	10-33
Exercises	10-34

Chapter 11

Three-Dimensional Solid Elements

Introduction	11-2
Problem Statement	11-3
Preliminary Analysis	11-4
Starting <i>SolidWorks</i>	11-7
Units Setup	11-8
Creating a CAD Model in SolidWorks	11-9
➤ Define the Sweep Path	11-9
➤ Define the Sweep Section	11-11
➤ Create the Swept Feature	11-13
➤ Create a Cut Feature	11-14
Activate the SolidWorks Simulation Module	11-16
Assign the Element Material Property	11-18
Applying Boundary Conditions - Constraints	11-19
Applying the External Load	11-20
Create the first FEA Mesh –Coarse Mesh	11-21
Run the Solver	11-23
Refinement of the Mesh– Global Element Size 0.10	11-25
Refinement of the FEA Mesh – Mesh Control Option	11-27
Refinement of the FEA Mesh – Automatic Transition	11-30
Comparison of Results	11-32
Notes on FEA Linear Static Analyses	11-33
Questions	11-34
Exercises	11-35

Chapter 12

3D Thin Shell Analysis

Introduction	12-2
Problem Statement	12-4
Preliminary Analysis	12-4
Starting <i>SolidWorks</i>	12-6
Units Setup	12-7
Creating a CAD Model in SolidWorks	12-8
Activate the SolidWorks Simulation Module	12-10
Assign the Element Material Property	12-12
Applying Boundary Conditions - Constraints	12-13
Applying the External Pressure	12-15
Create the first FEA Mesh –Coarse Mesh	12-16
Run the Solver and View the results	12-17
Refinement of the Mesh– Global Element Size 0.0125	12-18
Starting a New 3D Surface Model	12-19
Starting a New FEA Study	12-22

Completing the Definition of the Surface Model	12-23
Assign the Element Material Property	12-24
Applying Boundary Conditions - Constraints	12-25
Applying the External Pressure	12-29
Create the first FEA Mesh –Coarse Mesh	12-30
Run the Solver and View the results	12-31
Refinement of the Mesh– Global Element Size 0.009	12-32
Questions	12-33
Exercises	12-34

Chapter 13

Dynamic Modal Analysis

Introduction	13-2
Problem Statement	13-3
Preliminary Analysis	13-3
The Cantilever Beam Modal Analysis program	13-6
Starting <i>SolidWorks</i>	13-9
Units Setup	13-10
Creating a CAD Model in SolidWorks	13-11
Activate the SolidWorks Simulation Module	13-13
Assign the Element Material Property	13-16
Create the first FEA Mesh	13-17
View the results	13-19
Refinement of the Mesh– Global Element Size 0.0625	13-22
Adding an additional mass to the system	13-24
One-Dimensional Beam Frequency Analysis	13-28
Conclusions	13-29
Questions	13-30
Exercises	13-31

Index