

Introduction to Finite Element Analysis Using SolidWorks Simulation[®] 2012

Randy H. Shih

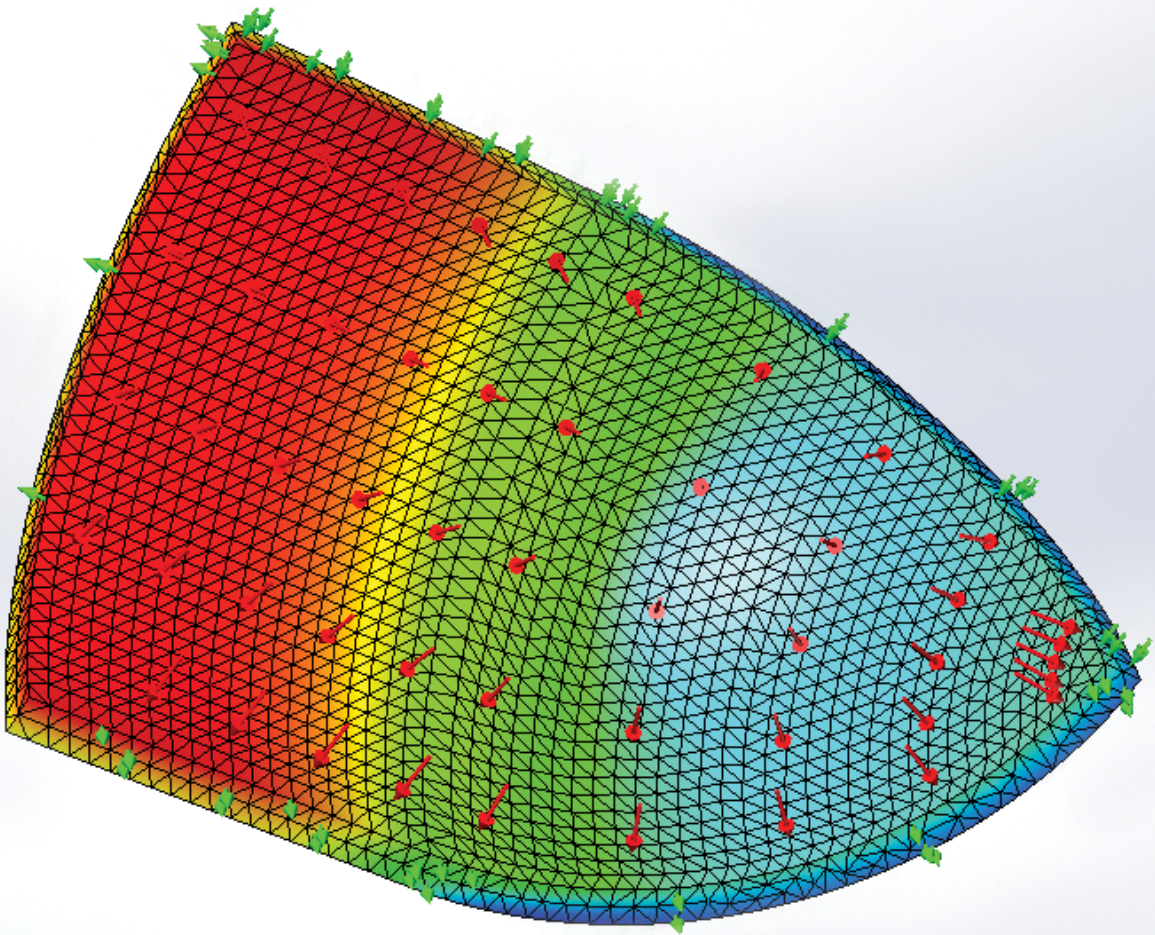


Table of Contents

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 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
Creating the CAD Model – Structural Member Approach	8-7
Creating a Rectangular Weldment Profile	8-9
Creating Structural Members using the New Profile	8-13
Adjusting the Orientation of the Profile	8-14
Adding a Datum Point	8-15
Activate the SolidWorks Simulation Module	8-17
Assign the Element Material Property	8-19
Applying Boundary Conditions - Constraints	8-20
Applying the Distributed Load	8-23
Create the FEA Mesh and Run the Solver	8-25
Shear and Moment Diagrams	8-26
Using the Probe Option to Examine Stress at Point1	8-28
Questions	8-29
Exercises	8-30

Chapter 9

Statically Indeterminate Structures

Introduction	9-2
Problem Statement	9-3
Preliminary Analysis	9-3

Starting <i>SolidWorks</i>	9-6
Creating the CAD Model	9-7
Creating a Circular Weldment Profile	9-9
Creating Structural Members using the New Profile	9-13
Adding a Datum Point for the Concentrated Load	9-14
Activate the SolidWorks Simulation Module	9-16
Assign the Element Material Property	9-18
Applying Boundary Conditions - Constraints	9-19
Applying the Concentrated Point Load	9-23
Create the FEA Mesh and Run the Solver	9-24
Viewing the Internal Loads of All members	9-25
Shear and Moment Diagrams	9-26
Questions	9-28
Exercises	9-29

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
Creating a CAD Model in SolidWorks	10-7
Activate the SolidWorks Simulation Module	10-10
Defining A Surface Model	10-12
Assign the Element Material Property	10-13
Applying Boundary Conditions - Constraints	10-14
Applying the External Load	10-17
H-Element versus P-Element	10-20
Create the first 2D Mesh –Coarse Mesh	10-21
Run the Solver	10-23
Refinement of the Mesh– Global Element Size 0.10	10-25
Refinement of the Mesh– Global Element Size 0.05	10-27
Refinement of the Mesh– Global Element Size 0.03	10-29
Refinement of the Mesh– Global Element Size 0.02	10-30
Comparison of Results	10-31
Questions	10-32
Exercises	10-33

Chapter 11

Three-Dimensional Solid Elements

Introduction	11-2
Problem Statement	11-3
Preliminary Analysis	11-4
Starting <i>SolidWorks</i>	11-7
Creating a CAD Model in SolidWorks	11-8
➤ Define the Sweep Path	11-8
➤ Define the Sweep Section	11-10
➤ Create the Swept Feature	11-12
➤ Create a Cut Feature	11-13
Activate the SolidWorks Simulation Module	11-15
Assign the Element Material Property	11-17
Applying Boundary Conditions - Constraints	11-18
Applying the External Load	11-19
Create the first FEA Mesh –Coarse Mesh	11-20
Run the Solver	11-22
Refinement of the Mesh– Global Element Size 0.10	11-24
Refinement of the FEA Mesh – Mesh Control Option	11-26
Refinement of the FEA Mesh – Automatic Transition	11-29
Comparison of Results	11-31
Notes on FEA Linear Static Analyses	11-32
Questions	11-33
Exercises	11-34

Chapter 12

3D Thin Shell Analysis

Introduction	12-2
Problem Statement	12-4
Preliminary Analysis	12-4
Starting <i>SolidWorks</i>	12-6
Creating a CAD Model in SolidWorks	12-7
Activate the SolidWorks Simulation Module	12-9
Assign the Element Material Property	12-11
Applying Boundary Conditions - Constraints	12-12
Applying the External Pressure	12-14
Create the first FEA Mesh –Coarse Mesh	12-15
Run the Solver and View the Results	12-16
Refinement of the Mesh– Global Element Size 0.0125	12-17
Starting a New 3D Surface Model	12-18
Starting a New FEA Study	12-21
Completing the Definition of the Surface Model	12-22
Assign the Element Material Property	12-23

Applying Boundary Conditions - Constraints	12-24
Applying the External Pressure	12-28
Create the first FEA Mesh –Coarse Mesh	12-29
Run the Solver and View the Results	12-30
Refinement of the Mesh– Global Element Size 0.010	12-31
Questions	12-32
Exercises	12-33

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
Creating a CAD Model in SolidWorks	13-10
Activate the SolidWorks Simulation Module	13-12
Assign the Element Material Property	13-15
Create the first FEA Mesh	13-16
View the Results	13-18
Refinement of the Mesh– Global Element Size 0.15	13-21
Adding an Additional Mass to the System	13-23
One-Dimensional Beam Frequency Analysis	13-27
Conclusions	13-28
Questions	13-29
Exercises	13-30

Index