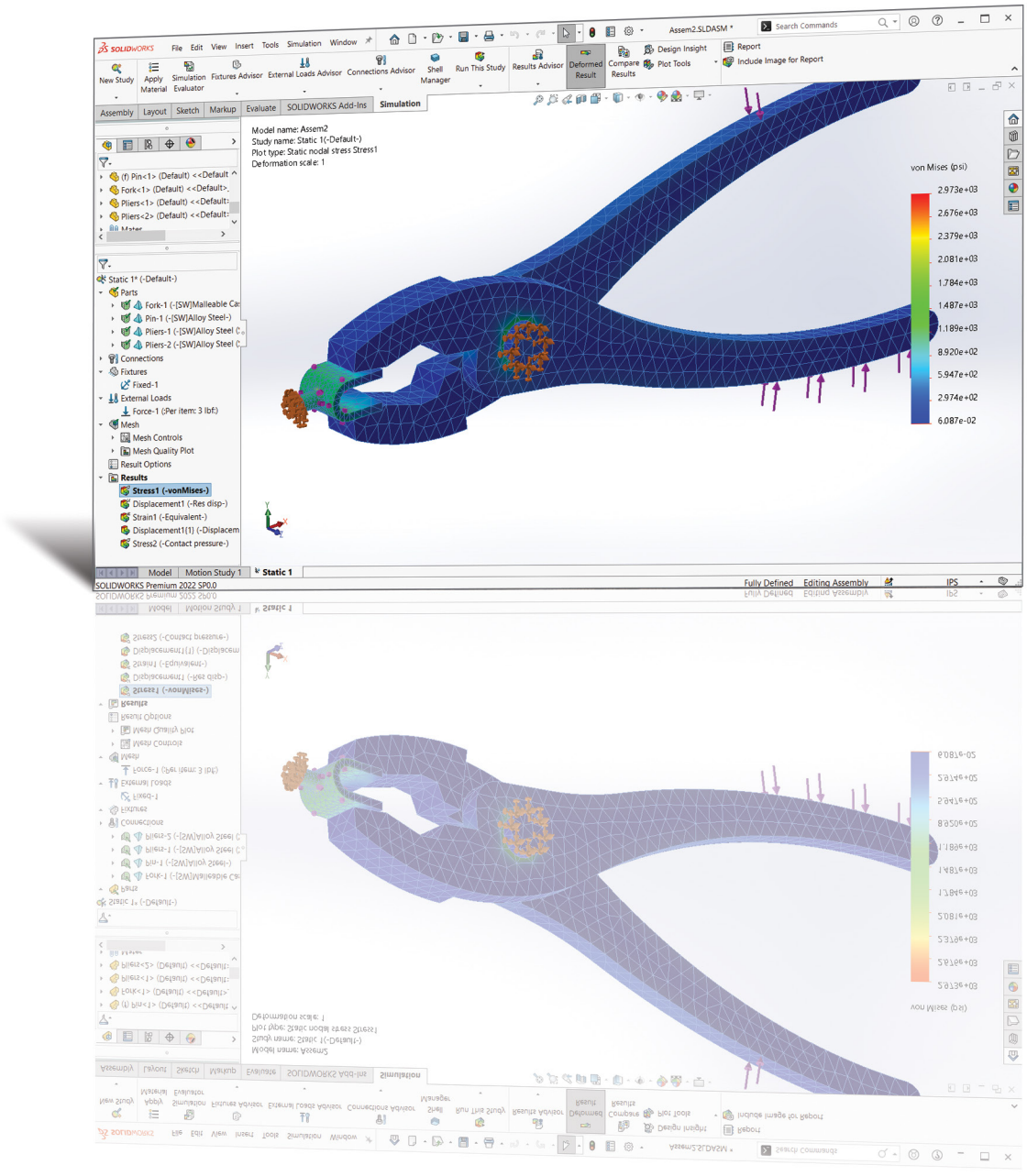
Introduction to

Finite Element Analysis Using SOLIDWORKS® Simulation 2022



Randy H. Shih

SDC PUBLICATIONS
Better Textbooks. Lower Prices.
www.SDCpublications.com

Visit the following websites to learn more about this book:

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 SOLIDWORKS Intro-9
Starting SOLIDWORKS Intro-9
SOLIDWORKS Screen Layout Intro-12
- Menu Bar Intro-12
- Menu Bar Pull-down Menus Intro-13
- Heads-up View Toolbar Intro-13
- Features Toolbar Intro-13
- Sketch Toolbar Intro-13
- Feature Manager Design Tree Intro-14
- Graphics Area Intro-15
- Reference Triad Intro-15
- Origin Intro-15
- Confirmation Corner Intro-15
- Graphics Cursor or Crosshairs Intro-15
- Message and Status Bar Intro-15

Using the SOLIDWORKS Command Manager Intro-16
Mouse Buttons Intro-17
[Esc] – Canceling Commands Intro-17
SOLIDWORKS Help System Intro-18
Leaving SOLIDWORKS Intro-18
Create a CAD Files Folder Intro-19

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: Create a Rough Sketch 1-12
Graphics Cursors 1-12
Geometric Relation Symbols 1-14
Step 2: Apply/Modify Relations and Dimensions 1-15
Viewing Functions – Zoom and Pan 1-17
Delete an Existing Geometry of the Sketch 1-18
Modify the Dimensions of the Sketch 1-19
Step 3: Complete the Base Solid Feature 1-20
Isometric View 1-21
Rotation of the 3D Model – Rotate View 1-21
Rotation and Panning – Arrow Keys 1-23
Dynamic Viewing – Quick Keys 1-24
3D Rotation 1-26
Viewing Tools – Heads-up View Toolbar 1-26
View Orientation 1-27
Display Style 1-28
Orthographic vs. Perspective 1-28
Customize the Heads-up View Toolbar 1-28
Step 4-1: Adding an Extruded Boss Feature 1-29
Step 4-2: Adding an Extruded Cut Feature 1-33
Step 4-3: Adding another Cut Feature 1-35
Save the Model 1-37
Questions 1-38
Exercises 1-39

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 SOLIDWORKS 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
Apply Boundary Conditions - Constraints 4-20
Apply 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 SOLIDWORKS 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 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
Viewing the Reaction Forces at the supports 5-27
Questions 5-28
Exercises 5-29

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
Start SOLIDWORKS 6-7
Units Setup 6-8
Create the CAD Model – Structural Member Approach 6-9
Create a New Weldment Profile in SOLIDWORKS 6-12
Create Structural Members using the New Profile 6-17
Editing the Dimensions of the New Profile 6-19
Activate the SOLIDWORKS Simulation Module 6-20
Setting Up the Truss Elements 6-22
Assign the Element Material Property 6-23
Applying Boundary Conditions - Constraints 6-24
Applying the External Load 6-25
Create the FEA Mesh and Run the Solver 6-27
Using the Probe Option to View Individual Stress 6-28
Viewing the Internal Loads of All Members 6-29
Questions 6-30
Exercises 6-31

Chapter 7
Basic Beam Analysis

Introduction 7-2
Modeling Considerations 7-2
Problem Statement 7-3
Preliminary Analysis 7-3
Start SOLIDWORKS 7-6
Units Setup 7-7
Create the CAD Model – Structural Member Approach 7-8
Create a Rectangular Weldment Profile 7-10
Create Structural Members Using the New Profile 7-14
Adjust the Orientation of the Profile 7-15

Add a Datum Point for the Concentrated Load 7-16
Activate the SOLIDWORKS Simulation Module 7-18
Assign the Element Material Property 7-20
Apply Boundary Conditions - Constraints 7-21
Apply the Concentrated Point Load 7-24
Apply 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
Start SOLIDWORKS 8-6
Create the CAD Model – Structural Member Approach 8-7
Create a Rectangular Weldment Profile 8-9
Create Structural Members Using the New Profile 8-13
Adjust the Orientation of the Profile 8-14
Add a Datum Point for the 1.5m Location 8-15
Activate the SOLIDWORKS Simulation Module 8-17
Assign the Element Material Property 8-19
Apply Boundary Conditions - Constraints 8-20
Apply 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
Start SOLIDWORKS 9-6
Create the CAD Model 9-7
Create a Circular Weldment Profile 9-9

Create Structural Members using the New Profile 9-13
Add a Datum Point for the Concentrated Load 9-14
Activate the SOLIDWORKS Simulation Module 9-16
Assign the Element Material Property 9-18
Apply Boundary Conditions - Constraints 9-19
Apply the Concentrated Point Load 9-22
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
Start SOLIDWORKS 10-6
Create the CAD Model 10-7
Activate the SOLIDWORKS Simulation Module 10-10
Assign the Element Material Property 10-13
Apply Boundary Conditions - Constraints 10-14
Apply the External Load 10-17
H-Element versus P-Element 10-18
Create the first FEA Mesh –Coarse Mesh 10-19
Run the Solver 10-21
Refinement of the FEA Mesh– Global Element Size 0.10 10-23
Refinement of the FEA Mesh– Global Element Size 0.05 10-25
Refinement of the FEA Mesh– Global Element Size 0.03 10-27
Refinement of the FEA Mesh– Global Element Size 0.02 10-28
Comparison of Results 10-29
Questions 10-30
Exercises 10-31

Chapter 11
Three-Dimensional Solid Elements

Introduction 11-2
Problem Statement 11-3
Preliminary Analysis 11-4

Start SOLIDWORKS 11-7
Create 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
Apply Boundary Conditions – Constraints 11-18
Apply the External Load to the system 11-19
Create the first FEA Mesh – Coarse Mesh 11-20
Run the Solver 11-22
Refinement of the FEA 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
Questions 11-32
Exercises 11-33

Chapter 12
3D Thin Shell Analysis

Introduction 12-2
Problem Statement 12-4
Preliminary Analysis 12-4
Start SOLIDWORKS 12-6
Create a 3D Solid Model in SOLIDWORKS 12-7
Activate the SOLIDWORKS Simulation Module 12-9
Assign the Element Material Property 12-11
Apply Boundary Conditions – Constraints 12-12
Apply the Pressure to the System 12-14
Create the first FEA Mesh – Coarse Mesh 12-15
Run the Solver and View the Results 12-16
Refinement of the FEA Mesh – Global Element Size 12.5 12-17
Start a New 3D Surface Model 12-18
Start a New FEA Study 12-21
Completing the Definition of the Surface Model 12-22
Assign the Element Material Property 12-23
Apply Boundary Conditions – Constraints 12-24
Apply the Pressure to the System 12-28
Create the first FEA Mesh – Coarse Mesh 12-29
Run the Solver and View the Results 12-30
Refinement of the FEA Mesh– Global Element Size 10.0 12-31
Notes on FEA Linear Static Analyses 12-32
Questions 12-33

Exercises 12-34

Chapter 13
FEA Static Contact Analysis

Introduction 13-2
Problem Statement 13-3
Parts 13-4
(1) Pliers-Jaw 13-4
(2) Pin 13-6
(3) Fork 13-6
Start SOLIDWORKS 13-8
Document Properties 13-8
Insert the First Component 13-9
Insert the Second Component 13-10
Assembly Mates 13-11
Insert the Third Component 13-14
Insert the Upper Jaw Component 13-16
Identifying Coincident Surfaces in the Model 13-19
Activate the SOLIDWORKS Simulation Module 13-20
Assign the Element Material Property 13-22
Apply Boundary Conditions – Constraints 13-23
Apply the External Load on the Handles 13-24
Global Contact Settings 13-25
Set up Specific Local Surfaces Interaction 13-26
Set up another Surface Interaction set on the PIN part 13-28
SOLIDWORKS Curvature-Based Mesh 13-30
Create the FEA Mesh 13-31
Run the Solver and View the Results 13-32
Use the Animate Option 13-34
Refinement of the FEA Mesh – Apply Mesh Control 13-35
Use the Section Clipping Option 13-37
Use the Iso Clipping Option 13-38
Set up a Contact Pressure Plot 13-39
Questions 13-42
Exercises 13-43

Chapter 14
Dynamic Modal Analysis

Introduction 14-2
Problem Statement 14-3
Preliminary Analysis 14-3
The Cantilever Beam Modal Analysis Program 14-6

Start SOLIDWORKS 14-9
Create the CAD Model 14-10
Activate the SOLIDWORKS Simulation Module 14-12
Assign the Element Material Property 14-14
Apply Boundary Conditions – Constraints 14-15
Create the first FEA Mesh 14-16
Viewing the Results 14-18
Refinement of the FEA Mesh – Global Element Size 0.15 14-21
Add an Additional Mass to the System 14-23
One-Dimensional Beam Frequency Analysis 14-27
Conclusions 14-28
Questions 14-29
Exercises 14-30

Appendix

Index