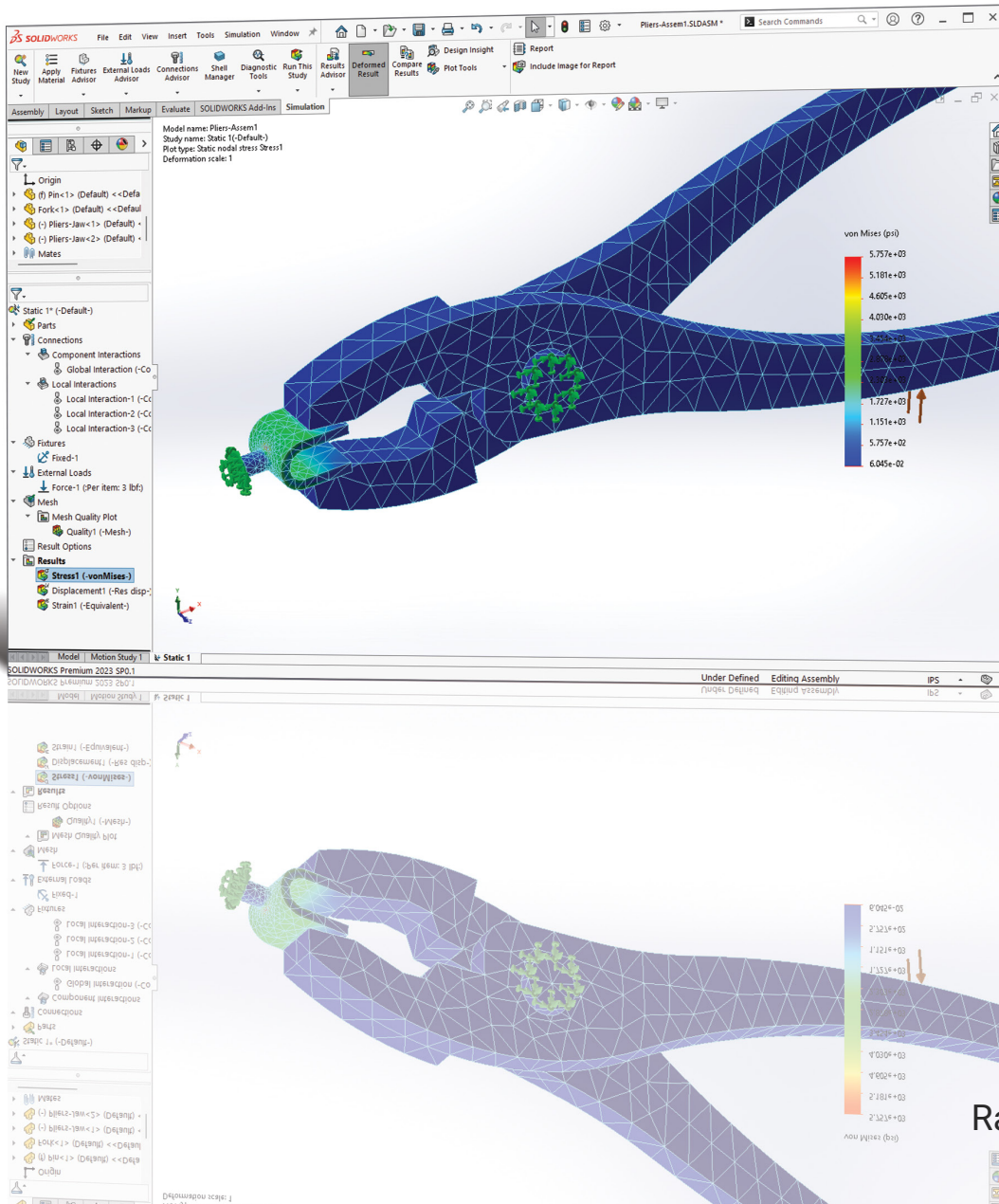


Introduction to

# Finite Element Analysis Using SOLIDWORKS® Simulation 2023



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

<b>Preface</b>	i
<b>Acknowledgments</b>	ii
<b>Introduction</b>	
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
<b>Chapter 1</b>	
<b>The Direct Stiffness Method</b>	
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

### 2D Axisymmetric and 3D Thin Shell Analyses

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	12-12
Apply Boundary Conditions – Constraints	12-13
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 5.0	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
Create a 3D Solid Model in SOLIDWORKS	12-32
Activate the SOLIDWORKS Simulation Module	12-34

Assign the Element Material Property	12-36
Apply Boundary Conditions – Constraints	12-37
Apply the Pressure to the System	12-39
Create the first FEA Mesh – Coarse Mesh	12-40
Run the Solver and View the Results	12-41
Refinement of the FEA Mesh – Global Element Size 12.5	12-42
Notes on FEA Linear Static Analyses	12-43
Questions	12-44
Exercises	12-45

## **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**