

# 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
Units Setup	Intro-11
SOLIDWORKS 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
• Feature Manager Design Tree	Intro-15
• Graphics Area	Intro-16
• Reference Triad	Intro-16
• Origin	Intro-16
• Confirmation Corner	Intro-16
• Graphics Cursor or Crosshairs	Intro-16
• Message and Status Bar	Intro-16
Using the SOLIDWORKS Command Manager	Intro-17
Mouse Buttons	Intro-18
[Esc] – Canceling Commands	Intro-18
SOLIDWORKS Help System	Intro-19
Leaving SOLIDWORKS	Intro-19
Creating a CAD Files Folder	Intro-20
<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: Creating 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
Modifying the Dimensions of the Sketch	1-19
Step 3: Completing 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
Customizing 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
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 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
Starting SOLIDWORKS	6-7
Units Setup	6-8
Creating the CAD Model – Structural Member Approach	6-9
Create a New Weldment Profile in SOLIDWORKS	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 the Truss Elements	6-21
Assign the Element Material Property	6-22
Applying Boundary Conditions - Constraints	6-23
Applying the External Load	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 SOLIDWORKS	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 the Concentrated Point Load	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 SOLIDWORKS	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 for the 1.5m Location	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 SOLIDWORKS	9-6
Create 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	9-22
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
Creating the CAD Model	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	10-18
H-Element versus P-Element	10-20
Create the first FEA Mesh –Coarse Mesh	10-21
Run the Solver	10-23
Refinement of the FEA Mesh– Global Element Size 0.10	10-25
Refinement of the FEA Mesh– Global Element Size 0.05	10-27
Refinement of the FEA Mesh– Global Element Size 0.03	10-29
Refinement of the FEA 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
Start SOLIDWORKS	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 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
Creating a 3D Solid 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 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
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 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
Starting SOLIDWORKS	13-8
Document Properties	13-8
Inserting the First Component	13-9
Inserting the Second Component	13-10
Assembly Mates	13-11
Inserting the Third Component	13-14
Inserting 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
Applying Boundary Conditions – Constraints	13-23
Applying the External Load on the Handles	13-24
Global Contact Settings	13-25
Set up Specific Local Contact Sets	13-26
Set up Another Contact Set on the PIN part	13-28
Create the FEA Mesh	13-30
Run the Solver and View the Results	13-31
Use the Animate Option	13-33
Use the Section Clipping Option	13-34
Use the Iso Clipping Option	13-35
Set up a Contact Pressure Plot	13-36
Refinement of the FEA Mesh – Apply Mesh Control	13-39
Questions	13-41
Exercises	13-42



## **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
Creating the CAD Model	14-10
Activate the SOLIDWORKS Simulation Module	14-12
Assign the Element Material Property	14-14
Applying 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
Adding 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**