

Analysis of Machine Elements Using

SOLIDWORKS® Simulation 2021



Shahin S. Nudehi, Ph.D., P.E.
John R. Steffen, Ph.D., P.E.



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

Table of Contents	i
Preface	xi
Intended Audience for this Text	xi
Using this SOLIDWORKS Simulation User Guide	xi
Instructor's Preface	xiii
Introduction	
Finite Element Analysis	I-1
Nodes, Elements, Degrees of Freedom, and Equations	I-2
SOLIDWORKS Simulation Elements	I-3
Solid Elements	I-3
Solid Element Degrees of Freedom	I-4
Shell Elements	I-5
Shell Elements Degrees of Freedom	I-6
Beam and Truss Elements	I-6
Meshing a Model	I-7
Stress Calculations for Nodes and Elements	I-8
Nodal Stress Values	I-9
Element Stress Values	I-10
Assumptions Applied to Linear Static Finite Element Analysis	I-11
Static Loading	I-11
Linear Material Behavior	I-11
Small Deformations	I-12
Closing Comments	I-12
Introduction to the SOLIDWORKS Simulation User Interface	I-13
Orientation and Set-up of SOLIDWORKS Work Environment	I-13
Customizing the SOLIDWORKS Screen	I-16
Orientation to the SOLIDWORKS Simulation Work Environment	I-20
Property Managers and Dialogue Boxes	I-23
Cautions and Other Facts You Should Know	I-25
Chapter 1	
Stress Analysis Using SOLIDWORKS Simulation	
Learning Objectives	1-1
Problem Statement	1-1
Creating a Static Stress Analysis (Study)	1-3
Assigning Material to the Model	1-6
Applying Fixtures	1-7
Applying External Loads	1-9

Analysis of Machine Elements Using SOLIDWORKS Simulation

Meshing the Model	1-12
Running the Solution	1-14
Examination of Results	1-15
Understanding Default Graphical Results	1-15
Results Predicted by Classical Stress Equations	1-17
Simulation Results for Stress in Y-Direction	1-19
Using the Probe Tool	1-21
Customizing Graphs	1-28
Summary	1-31
To SAVE or Not to SAVE Files, That is the Question?	1-31
Option 1 - Close a File and SAVE its Contents Using the Original File Name	1-32
Option 2 - Close a File and SAVE its Contents Using a Different File Name	1-32
Option 3 - Close a File without Saving Results	1-33
Option 4 - Re-open a SOLIDWORKS Simulation File Saved Using Option 1 or Option 2	1-33
Exercises	1-35

Chapter 2 Curved Beam Analysis

Learning Objectives	2-1
Problem Statement	2-1
Creating a Static Analysis (Study)	2-2
Setting up the Simulation Toolbar	2-3
Assign Material Properties to the Model	2-5
Applying Fixtures	2-6
Applying External Load(s)	2-8
Inserting Split Lines	2-9
Applying Force to an Area Bounded by Split Lines	2-12
Meshing the Model	2-13
Solution	2-15
Examination of Results	2-16
Analysis of von Mises Stresses Within the Model	2-16
Verification of Results	2-19
Results Predicted by Classical Stress Equations	2-19
Comparison with Finite Element Results	2-21
Assessing Safety Factor	2-25
Alternate Stress Display Option	2-31
Determining Reaction Forces	2-32
Is it Significant That the Model is NOT in Equilibrium?	2-33
Soft Springs Can Be Applied Where Other Restraints are not Appropriate	2-34
Important Caution Regarding Strength and Safety Factor	2-35
Logging Out of the Current Analysis	2-35
Exercises	2-36

Chapter 3

Stress Concentration Analysis

Learning Objectives	3-1
Problem Statement	3-1
Create a Static Analysis (Study)	3-2
Defeaturing the Model	3-3
Assign Material Properties to the Model	3-4
Apply Fixtures and External Loads	3-4
Mesh the Model	3-6
Solution	3-8
Examination of Results	3-8
Stress Plots	3-8
Creating a “Copy” of a Plot	3-12
Displacement Plot	3-14
Creating New Studies	3-17
Basic Parts of the Graphical User Interface	3-17
Study Using High Quality Elements and COARSE Mesh Size	3-18
Study Using High Quality Elements and DEFAULT Mesh Size	3-23
Study Using High Quality Elements and FINE Mesh Size	3-24
Study Using High Quality Elements and MESH CONTROL	3-25
Summary	3-29
Results Analysis	3-30
Create Multiple Viewports	3-30
What Can Be Learned from this Example?	3-33
Other Uses of the Copy Feature	3-33
Analysis Insight - Solution Convergence	3-33
Comparison of Classical and FEA Results	3-36
Logging Out of the Current Analysis	3-37
Exercises	3-38

Chapter 4

Thin and Thick Wall Pressure Vessels

Learning Objectives	4-1
Thin-Wall Pressure Vessel (Using Shell Elements)	4-1
Problem Statement	4-1
Understanding System Default Settings	4-4
Creating a Static Analysis Using Shell Elements	4-10
Converting a Solid Model to a Shell Model	4-11
Open a New Simulation Study	4-15
Assign Material Properties	4-15
Define Shell Thickness	4-15
Assign Fixtures and External Loads	4-16
Restraints Applied Using Reference Geometry	4-16

Analysis of Machine Elements Using SOLIDWORKS Simulation

Pressure Load Applied	4-20
Mesh the Model	4-21
Solution	4-23
Results Analysis	4-23
Results Comparison (Tangential Stress = σ_1 on Cylindrical Surface)	4-25
Other Meaningful Results Comparisons	4-27
Guidance for Determining Symmetry Fixtures	4-27
Application of Symmetry Fixtures	4-29
Closing Observations	4-30
Log Out of the Current Analysis	4-31

Thick Wall Pressure Vessel

Problem Statement	4-32
Defining the Study	4-33
Assign Material Properties	4-34
Define Fixtures and External Loads	4-34
Create a Duplicate Study	4-37
Mesh the Model – Default Size Standard Mesh	4-37
Mesh the Model – Fine Size Standard Mesh	4-38
Solution	4-40
Results Analysis	4-40
Displacement Analysis	4-40
von Mises Stress Analysis	4-42
Tangential Stress Analysis	4-42
Adjusting Stress Magnitude Display Parameters	4-45
Using Section Clipping to Observe Stress Results	4-48

Thick Wall Pressure Vessel (Axisymmetrical Approach)

Problem Statement	4-52
Thick-Wall Pressure Vessel	4-52
Create a Static Analysis (Study)	4-53
Assign Material Properties	4-54
Define Fixtures and External Loads	4-54
Mesh the Model – Standard Size Standard Mesh	4-55
Solution	4-56
Results Analysis	4-56
Exercises	4-59

Chapter 5

Interference Fit Analysis

Learning Objectives	5-1
Problem Statement	5-1
Interference Check	5-2
Create a Static Analysis (Study)	5-3
Assign Material Properties to the Model	5-4

SOLIDWORKS Default Connection Definition	5-4
Defeature and Simplify the Model	5-5
Apply Fixtures	5-6
Reduce Model Size Using Symmetry	5-6
Define Symmetry Restraints (Fixtures)	5-7
Apply Fixtures to Eliminate Rigid Body Motion	5-9
Use Contact Interactions to Define a Shrink Fit	5-11
Mesh the Model and Run the Solution	5-14
Examination of Results	5-15
Default Stress Plot	5-15
Stress Plots in a Cylindrical Coordinate System	5-18
Tangential (Circumferential or Hoop) Stress	5-18
Radial Stress	5-21
Verification of Results	5-23
Stress Predicted by Classical Interference Fit Equations	5-23
Stress Predicted by Finite Element Analysis	5-24
Radial Stress Comparison	5-24
Tangential Stress Comparison	5-26
Quantifying Radial Displacements	5-27
Generating a Report	5-30
Exercises	5-32

Chapter 6 Contact Analysis

Learning Objectives	6-1
Problem Statement	6-1
Preparing the Model for Analysis	6-2
Add Reference Planes	6-3
Insert Split Lines	6-4
Create the Assembly Model	6-5
Cut Model on Symmetry Plane	6-10
Create a Finite Element Analysis (Study)	6-13
Assign Material Properties	6-13
Assign Fixtures and External Loads	6-13
Symmetry and Immovable Restraints	6-13
Connections Define Contact Conditions	6-14
Apply a Directional Load	6-16
Mesh the Model and Run the Solution	6-19
Results Analysis	6-19
Von Mises Stress	6-19
Iso Clipping	6-20
Animating Stress Results	6-22
Displacement Results	6-23
Contact Pressure/Stress	6-24
Exercises	6-26

Chapter 7 Bolted Joint Analysis

Learning Objectives	7-1
Problem Statement	7-1
Create a Static Analysis (Study)	7-2
Assign Material Properties to the Model	7-2
Apply External Load and Fixtures	7-3
Traditional Loads and Fixtures	7-3
Define Bolted Joint Restraints	7-4
Define Local Contact Conditions	7-11
Mesh the Model and Run Solution	7-13
Results Analysis for the Downward External Load	7-16
von Mises Stress	7-16
Bolt Forces (for Downward Load)	7-18
Define a New Study with the Applied Force Acting Upward	7-19
Results Analysis for the Upward External Load	7-20
von Mises Stress	7-20
Bolt Forces (for Upward Load)	7-23
Bolt Clamping Pressure	7-26
Summary	7-30
Exercises	7-31

Chapter 8 Design Optimization

Learning Objectives	8-1
Problem Statement	8-1
Complete and Run the Initial Study	8-2
Creating an Optimization Design Study	8-6
Understanding Optimization Design Terminology	8-7
Selecting and Specifying Design Parameters (Variables)	8-9
Define Constraints and their Sensors	8-14
Define the Stress Constraint and Sensor	8-14
Define the Displacement Constraint and Sensor	8-15
Define a Goal and its Sensor	8-17
Overview of Results	8-18
General Observations	8-19
Local Trend Graphs	8-20
Local Trend Graph of the Design Goal versus a Design Variable	8-20
Local Trend Graph of a Constraint versus a Design Variable	8-21
Closing Observations	8-23
Exercises	8-25

Chapter 9
Elastic Buckling

Learning Objectives	9-1
Problem Statement	9-1
Opening SOLIDWORKS	9-2
Create a Buckling Analysis (Fixed-Free End Conditions)	9-2
Assign Material Properties to the Model	9-2
Assign Fixed-Free Fixture End Conditions to the Model	9-3
External Loads	9-3
Buckling Analysis Options	9-4
Meshing the Model	9-4
Solution	9-5
Examination of Results	9-6
Buckling Mode Shapes (Optional)	9-7
Validity of Elastic Buckling Simulation	9-10
Create a Buckling Analysis Using Fixed-Fixed End Conditions	9-11
Assign Material Properties to the Model	9-11
Prepare the Model for Assignment of Fixed-Fixed End Conditions	9-12
Assign Fixed-Fixed Fixture to Top End of Model	9-13
External Loads	9-14
Buckling Analysis Options	9-15
Meshing and Running the Model	9-16
Examination of Results	9-17
Validity of Elastic Buckling Simulation	9-17
Create a Buckling Analysis Using Pinned-Pinned End Conditions	9-19
Assign Material Properties to the Model	9-19
Prepare the Model for Assignment of Pinned-Pinned End Conditions	9-19
Pinned-Pinned Fixture	9-20
External Loads	9-22
Meshing and Running the Model	9-22
Examination of Results	9-23
Buckling Mode Shape (Optional)	9-24
Validity of Elastic Buckling Simulation	9-25
Create a Buckling Analysis assuming an Eccentrically Loaded Pin with Pinned-Pinned End Conditions	9-26
Pinned-Pinned Fixture	9-27
External Loads	9-28
Meshing and Running the Model	9-29
Examination of Results	9-29
Summary	9-31
Exercises	9-32

Chapter 10 Fatigue Testing Analysis

Learning Objectives	10-1
Problem-1: Fatigue Analysis of a Cylindrical Rod Subject to Reversed Bending	10-1
Creating a Static Analysis (Study)	10-2
Assign Material Properties to the Model	10-2
Applying Fixtures And External Loads	10-3
Meshing the Model	10-4
Static Solution	10-5
Fatigue Study	10-5
Add Event	10-5
Generating the SN-Curve	10-6
Navigating the SN-Curve Calculator	10-7
Setting Fatigue Study Parameters	10-10
Fatigue Study Solution	10-11
Verification of Results	10-12
Problem-2: Fatigue Analysis of a Cylinder Subject to a Combined Loading	10-15
Duplicating the Static Analysis (Study) and Fatigue Studies	10-15
Modifying External Load(s)	10-16
Modifying Fatigue Study Parameters	10-17
Edit Fatigue Event	10-17
Setting Fatigue Study Parameters	10-17
Modifying the SN-Curve	10-18
Fatigue Study Solution	10-20
Verification of Results	10-21
Exercises	10-23

Chapter 11 Thermal Stress Analysis

Learning Objectives	11-1
Problem Statement	11-1
Static Analysis	11-2
Material Properties Applied to the Model	11-2
Assign Connection in the Model	11-3
Assign Fixtures to the Model	11-4
External Loads (Method-1)	11-4
External Loads (Method-2)	11-6
Material Properties Applied to the Model	11-6
Applying Thermal Loads for Uniform Temperature	11-6
Solve the Thermal Study and Apply the Solution	11-8
Apply Mesh for Thermal Study	11-10
Reference Temperature at Zero Strain	11-10
Examination of Results	11-11
Default Stress Plot	11-11
Stress Plots in a Cylindrical Coordinate System	11-12

Table of Contents

Validation of Radial Stress Results	11-16
Validation of Axial Displacement Results	11-17
Exercises	11-18
APPENDIX A	
Organizing Assignments Using MS Word®	A-1
APPENDIX B	
Alternative Method to Change Screen Background color	B-1
INDEX	Index-1