

Analysis of Machine Elements

using SolidWorks Simulation 2010



John R. Steffen,
Ph.D., P.E.

SDC
PUBLICATIONS

www.SDCpublications.com

Schroff Development Corporation

Table of Contents

Table of Contents	i
Preface	vii
Intended Audience for this Text	vii
Using this SolidWorks Simulation User Guide	vii
Instructors Preface	ix
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-10
Static Loading	I-10
Linear Materials	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-15
Orientation to the SolidWorks Simulation Work Environment	I-18
Property Managers and Dialogue Boxes	I-20
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	1-5
Applying Fixtures	1-6
Applying External Loads	1-8
Meshing the Model	1-11
Running the Solution	1-13

Analysis of Machine Elements using SolidWorks Simulation

Examination of Results	1-14
Default SolidWorks Simulation Graphical Results	1-14
Results Predicted by Classical Stress Equations	1-16
SolidWorks Simulation Results for Stress in Y-Direction	1-18
Using the Probe Tool	1-20
Customizing Graphs	1-27
Summary	1-29
Exercises	1-31

Chapter 2

Curved Beam Analysis

Learning Objectives	2-1
Problem Statement	2-1
Creating a Static Analysis (Study)	2-2
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 Display Option	2-31
Reaction Forces	2-32
Logging Out of the Current Analysis	2-33
Exercises	2-34

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
Meshing the Model	3-6
Solution	3-7

Examination of Results	3-7
Stress Plots	3-7
Creating a Copy of a Plot	3-11
Displacement Plot	3-13
Creating New Studies	3-15
Basic Parts of the Graphical User Interface	3-15
Study Using High Quality Elements and COARSE Mesh Size	3-17
Method 1 – Item by Item Copying	3-17
Study Using High Quality Elements and DEFAULT Mesh Size	3-20
Method 2 – Copying Multiple Items Simultaneously	3-20
Study Using High Quality Elements and FINE Mesh Size	3-22
Method 3 – Duplicating an Entire Study	3-22
Study Using High Quality Elements and MESH CONTROL	3-25
Summary	3-30
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
Comparison of Classical and FEA Results	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
Assigning Fixtures and External Loads	4-16
Symmetry Restraints Applied	4-16
Pressure Load Applied	4-20
Mesh the Model	4-21
Solution	4-22
Results Analysis	4-22
Thick Wall Pressure Vessel	4-28
Problem Statement	4-28
Defining the Study	4-29
Assign Material Properties	4-30
Define Fixtures and External Loads	4-30
Mesh the Model	4-33
Solution	4-35

Analysis of Machine Elements using SolidWorks Simulation

Results Analysis	4-35
Displacement Analysis	4-35
von Mises Stress Analysis	4-37
Tangential Stress Analysis	4-38
Adjusting Stress Magnitude Display Parameters	4-40
Using Section Clipping to Observe Stress Results	4-43
Exercises	4-47

Chapter 5

Interference Fit Analysis

Learning Objectives	5-1
Problem Statement	5-1
Interference Check	5-2
Creating a Static Analysis (Study)	5-3
Assign Material Properties to the Model	5-4
Defeature and Simplify the Model	5-5
Apply Fixtures	5-6
Un-suppress Part of the Model to Use Symmetry	5-6
Define Symmetry Restraints (Fixtures)	5-7
Apply Restraints to Eliminate Rigid Body Motion	5-8
Define a Shrink Fit	5-10
Mesh the Model and Run the Solution	5-13
Examination of Results	5-14
Default Stress Plot	5-14
Stress Plots in a Cylindrical Coordinate System	5-17
Circumferential (Tangential or Hoop) Stress	5-17
Radial Stress	5-20
Verification of Results	5-22
Stress Predicted by Classical Interference Fit Equations	5-22
Stress Predicted by Finite Element Analysis	5-23
Radial Stress Comparison	5-23
Circumferential Stress Comparison	5-25
Quantifying Radial Displacements	5-26
Generating a Report	5-29
Exercises	5-32

Chapter 6

Contact Analysis in a Trunion Mount

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

Creating the Assembly Model	6-5
Cut Model on Symmetry Plane	6-9
Create a Finite Element Analysis (Study)	6-12
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
Meshing the Model and Running 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 Loads 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

APPENDIX A

Organizing Assignments using MS Word®	A-1
---------------------------------------	-----

INDEX

Index-1
