

Analysis of Machine Elements

using COSMOSWorks 2008



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

SDC
PUBLICATIONS

Schroff Development Corporation
www.schroff.com

Better Textbooks. Lower Prices.

Table of Contents

Table of Contents	i
Preface	vii
Intended Audience for this Text	vii
Using this COSMOSWorks User Guide	vii
Instructors Preface	ix
Introduction	
Finite Element Analysis	I-1
Nodes, Elements, Degrees of Freedom, and Equations	I-2
COSMOSWorks Elements	I-3
Solid Elements	I-3
Solid Element Degrees of Freedom	I-4
Shell Elements	I-4
Shell Elements Degrees of Freedom	I-5
Meshing a Model	I-5
Introduction to the COSMOSWorks User Interface	I-7
Orientation and Set-up of SolidWorks Work Environment	I-7
Orientation to the COSMOSWorks Work Environment	I-11
Property Managers and Dialogue Boxes	I-12
Chapter 1	
Basic Stress Analysis Using COSMOSWorks	1-1
Learning Objectives	1-1
Problem Statement	1-1
Creating a Static Stress Analysis Study	1-3
Assigning Material to the Model	1-4
Applying Restraints	1-6
Applying Loads	1-8
Meshing the Model	1-11
Running the Solution	1-13
Examination of Results	1-14
Default COSMOSWorks Graphical Results	1-14
Results Predicted by Classical Stress Equations	1-15
COSMOSWorks Results for Stress in Y-Direction	1-16
Using the Probe Tool	1-18
Summary	1-25
Exercises	1-26

Chapter 2

Curved Beam Analysis

2-1

Learning Objectives	2-1
Problem Statement	2-1
Creating a Static Analysis (Study)	2-2
Assign Material Properties to the Model	2-4
Applying Restraints	2-6
Applying Load(s)	2-7
Inserting Split Lines	2-8
Applying Force to an Area Bounded by Split Lines	2-11
Meshing the Model	2-12
Solution	2-14
Examination of Results	2-15
Analysis of von Mises Stresses Within the Model	2-14
Verification of Results	2-18
Results Predicted by Classical Stress Equations	2-18
Comparison with Finite Element Results	2-20
Assessing Safety Factor for the Curved Beam	2-23
Reaction Forces	2-28
Logging Out of the Current Analysis	2-29
Exercises	2-30

Chapter 3

Stress Concentration Analysis

3-1

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 Restraints and Loads	3-5
Meshing the Model	3-6
Solution	3-8
Examination of Results	3-8
Stress Plots	3-8
Creating a Copy of a Plot	3-11
Displacement Plot	3-13
Creating New Studies	3-15
Study Using a High Quality Elements and COARSE Mesh Size	3-15
Study Using a High Quality Elements and DEFAULT Mesh Size	3-20
Study Using a High Quality Elements and FINE Mesh Size	3-21
Study Using High Quality Elements and Mesh CONTROL	3-21
Results Analysis	3-26
Create Multiple Viewports	3-26
What Can Be Learned from this Example?	3-28

Other Uses of the Copy Feature	3-28
Comparison of Classical and FEA Results	3-32
Exercises	3-33

Chapter 4
Thin and Thick Wall Pressure Vessels

Learning Objectives	4-1
Thin-Wall Pressure Vessel	4-1
Problem Statement	4-1
Defining Options at Start of a Study	4-4
Creating a Static Analysis Using Shell Elements	4-10
Assigning Material Properties	4-10
Assigning Loads and Restraints	4-11
Symmetry Restraints Applied	4-11
Pressure Load Applied	4-14
Meshing the Model	4-14
Solution	4-15
Results Analysis	4-16

Thick Wall Pressure Vessel	4-21
Problem Statement	4-21
Defining the Study	4-21
Assign Material Properties	4-22
Define Restraints and Loads	4-22
Mesh the Model	4-25
Solution	4-27
Results Analysis	4-27
Displacement Analysis	4-27
Von Mises Stress Analysis	4-29
Tangential Stress Analysis	4-29
Adjusting Stress Magnitude Display Parameters	4-31
Using Section Clipping to Observe Stress Results	4-34
Exercises	4-38

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 the Model	5-5
Apply Loads and Restraints	5-6

Analysis of Machine Elements using COSMOSWorks

Un-suppress Part of the Model to Use Symmetry	5-6
Define Symmetry Restraints	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 the 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	5-28
Exercises	5-31

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
Create a Finite Element Analysis (Study)	6-10
Assign Material Properties	6-10
Cut Model on Symmetry Plane	6-10
Assign Restraints and Loads	6-14
Symmetry and Immovable Restraints	6-14
Contact/Gaps Restraints	6-14
Apply a Directional Load to the Pin	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	7-2
Assign Material Properties to the Model	7-2
Apply Loads and Restraints	7-3
Traditional Loads and Restraints	7-3
Define Bolted Joint Restraints	7-4
Define Local Contact Conditions	7-11
Mesh the Model and Run Solution	7-13
Results Analysis (Downward Load)	7-14
Von Mises Stress	7-14
Bolt Forces	7-15
Define a New Study with the Applied Load Reversed	7-16
Results Analysis (Upward Load)	7-17
Von Mises Stress	7-17
Bolt Forces (Upward Load)	7-17
Bolt Clamping Pressure	7-22
Summary	7-26
Exercises	7-27