Basic System Finite Element Analysis
Part 2
# Contents

## 1 About the Verification Problems...

### Introduction

1-1

## 2 Linear Static Analysis

### Introduction

2-1

- S1: Pin Jointed Truss .......................... 2-6
- S2: Long Thick-Walled Cylinder .............. 2-7
- S3A, S3B: Simply Supported Rectangular Plate 2-9
- S4: Thermal Stress Analysis of a Truss Structure 2-10
- S5: Thermal Stress Analysis of a 2D Structure 2-12
- S6A, S6B: Deflection of a Cantilever Beam 2-13
- S7: Beam Stresses and Deflections ............. 2-14
- S8: Tip Displacements of a Circular Beam 2-15
- S9A: Clamped Beam Subject to Imposed Displacement 2-16
- S9B: Clamped Beam Subject to Imposed Rotation 2-18
- S10A, S10B: Bending of a Solid Beam 2-19
- S11: Thermal Stress Analysis of a 3D Structure 2-21
- S12: Deflection of a Hinged Support .......... 2-22
- S13: Statically Indeterminate Reaction Force Analysis 2-23
- S14A, S14B: Space Truss with Vertical Load 2-24
- S15: Out-of-Plane Bending of a Curved Bar 2-25
- S16A, S16B: Curved Pipe Deflection ............ 2-26
- S17: Rectangular Plate Under Triangular Thermal Loading 2-28
- S18: Hemispherical Dome Under Unit Moment Around Free Edge 2-29
S19: Hollow Thick-walled Cylinder Subject to Temperature and Pressure ........................................ 2-30
S20A, S20B: Cylindrical Shell Roof ................................................................. 2-31
S21A, S21B: Antisymmetric Cross-Ply Laminated Plate (SHELL4L) .... 2-33
S22: Thermally Loaded Support Structure ............................................... 2-35
S23: Thermal Stress Analysis of a Frame .................................................. 2-36
S24: Thermal Stress Analysis of a Simple Frame .................................. 2-38
S25: Torsion of a Square Box Beam ......................................................... 2-39
S26: Beam With Elastic Supports and a Hinge ............................................. 2-41
S27: Frame Analysis with Combined Loads ............................................. 2-43
S28: Cantilever Unsymmetric Beam ............................................................ 2-45
S29A, S29B: Square Angle-Ply Composite Plate Under Sinusoidal Loading ........................................ 2-47
S30: Effect of Transverse Shear on Maximum Deflection ......................... 2-48
S31: Square Angle-Ply Composite Plate Under Sinusoidal Loading ........ 2-50
S33A, S33M: Substructure of an Airplane (Wing) .................................... 2-53
S34: Tie Rod with Lateral Loading ............................................................... 2-54
S35A, S35B: Spherical Cap Under Uniform Pressure (Solid) .................. 2-56
S36A, S36M: Substructure of a Simply Supported Plate ......................... 2-58
S37: Hyperboloidal Shell Under Uniform Ring Load Around Free Edge .... 2-60
S38: Rotating Solid Disk ............................................................................ 2-62
S39: Unbalanced Rotating Flywheel ........................................................... 2-63
S40: Truss Structure Subject to a Concentrated Load .............................. 2-64
S41: Reactions of a Frame Structure .......................................................... 2-65
S42A, S42B: Reactions and Deflections of a Cantilever Beam .............. 2-66
S43: Bending of a T Section Beam ............................................................... 2-67
S44A, S44B: Bending of a Circular Plate with a Center Hole .................. 2-68
S45: Eccentric Frame .................................................................................. 2-70
S46, S46A, S46B: Bending of a Cantilever Beam ....................................... 2-71
S47, S47A, S47B: Bending of a Cantilever Beam ....................................... 2-73
S48: Rotation of a Tank of Fluid (PLANE2D Fluid) .................................... 2-75
S49A, S49B: Acceleration of a Tank of Fluid (PLANE2D Fluid) .......... 2-77
Part 2  Verification Problems

S51: Gable Frame with Hinged Supports ........................................ 2-80
S52: Support Reactions for a Beam with Intermediate Forces
and Moments ................................................. 2-81
S53: Beam Analysis with Intermediate Loads ......................... 2-83
S54: Analysis of a Plane Frame with Beam Loads ................. 2-85
S55: Laterally Loaded Tapered Beam ................................. 2-86
S56: Circular Plate Under a Concentrated Load (SHELL9 Element) . . . 2-87
S57: Test of a Pinched Cylinder with Diaphragm (SHELL9 Element) . . 2-89
S58A, S58B, S58C: Deflection of a Twisted Beam with Tip Force .... 2-91
S59A, S59B, S59C: Sandwich Square Plate Under Uniform Loading
(SHELL9L) .................................................. 2-93
S60: Clamped Square Plate Under Uniform Loading ............... 2-95
S61: Single-Edge Cracked Bend Specimen, Evaluation of Stress Intensity
Factor Using Crack Element .................................. 2-97
S62: Plate with Central Crack ........................................ 2-98
S63: Cyclic Symmetry Analysis of a Hexagonal Frame .......... 2-99
S64A, S64B: Cyclic Symmetry ................................. 2-100
S65: Fluid-Structure Interaction, Rotation of a Tank of Fluid .... 2-101
S66: Fluid-Structure Interaction, Acceleration of a Tank of Fluid .... 2-103
S67: MacNeal-Harder Test ......................................... 2-105
S68: P-Method Solution of a Square Plate with Hole ............. 2-106
S69: P-Method Analysis of an Elliptic Membrane Under Pressure .. 2-107
S70: Thermal Analysis with Temperature Dependent Material .... 2-108
S71: Sandwich Beam with Concentrated Load ...................... 2-109
S74: Constant Stress Patch Test (TETRA4R) ..................... 2-110
S75: Analysis of a Cantilever Beam with Gaps, Subject to Different
Loading Conditions ........................................ 2-111
S76: Simply Supported Beam Subject to Pressure from
a Rigid Parabolic Shaped Piston ................................ 2-113
S77: Bending of a Solid Beam Using Direct Material Matrix Input ... 2-115
S78: P-Adaptive Analysis of a Square Plate with a Circular Hole .... 2-117
S79: Hemispherical Shell Under Unit Moment Around Free Edge .. 2-119
S80: Axisymmetric Hyperbolic Shell Under a Cosine Harmonic
Loading on the Free Edge .................................. 2-120
S81: Circular Plate Under Non-Axisymmetric Load ................ 2-121
## Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>S82: Twisting of a Long Solid Shaft</td>
<td>2-123</td>
</tr>
<tr>
<td>S83: Bending of a Long Solid Shaft</td>
<td>2-125</td>
</tr>
<tr>
<td>S84: Submodeling of a Plate</td>
<td>2-127</td>
</tr>
<tr>
<td>S85: Plate on Elastic Foundation</td>
<td>2-128</td>
</tr>
<tr>
<td>S86: Plate with Coupled Degrees of Freedom</td>
<td>2-129</td>
</tr>
<tr>
<td>S87: Gravity Loading of ELBOW Element</td>
<td>2-130</td>
</tr>
<tr>
<td>S88A, S88B: Single-Edge Cracked Bend Specimen, Evaluation of Stress</td>
<td></td>
</tr>
<tr>
<td>Intensity Factor Using the J-integral</td>
<td></td>
</tr>
<tr>
<td>S89A, S89B: Slant-Edge Cracked Plate, Evaluation of Stress Intensity</td>
<td></td>
</tr>
<tr>
<td>Factors Using the J-integral</td>
<td></td>
</tr>
<tr>
<td>S90A, S90B: Penny-Shaped Crack in Round Bar, Evaluation of Stress</td>
<td></td>
</tr>
<tr>
<td>Intensity Factor Using the J-integral</td>
<td></td>
</tr>
<tr>
<td>S91: Crack Under Thermal Stresses, Evaluation of Stress Intensity</td>
<td></td>
</tr>
<tr>
<td>Using the J-integral</td>
<td></td>
</tr>
<tr>
<td>S92A, S92B: Simply Supported Rectangular Plate, Using Direct Material</td>
<td></td>
</tr>
<tr>
<td>Matrix Input</td>
<td>2-139</td>
</tr>
<tr>
<td>S93: Accelerating Rocket</td>
<td>2-141</td>
</tr>
<tr>
<td>S94A, S94B, S94C: P-Method Solution of a Square Plate with a Small</td>
<td></td>
</tr>
<tr>
<td>Hole</td>
<td>2-143</td>
</tr>
<tr>
<td>S95A, S95B, S95C: P-Method Solution of a U-Shaped Circumferential</td>
<td></td>
</tr>
<tr>
<td>Groove in a Circular Shaft</td>
<td>2-146</td>
</tr>
</tbody>
</table>

### 3 Modal (Frequency) Analysis 3-1

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction</td>
<td>3-1</td>
</tr>
<tr>
<td>F1: Natural Frequencies of a Two-Mass Spring System</td>
<td>3-3</td>
</tr>
<tr>
<td>F2: Frequencies of a Cantilever Beam</td>
<td>3-4</td>
</tr>
<tr>
<td>F3: Frequency of a Simply Supported Beam</td>
<td>3-5</td>
</tr>
<tr>
<td>F4: Natural Frequencies of a Cantilever Beam</td>
<td>3-6</td>
</tr>
<tr>
<td>F5: Frequency of a Cantilever Beam with Lumped Mass</td>
<td>3-7</td>
</tr>
<tr>
<td>F6: Dynamic Analysis of a 3D Structure</td>
<td>3-8</td>
</tr>
<tr>
<td>F7A, F7B: Dynamic Analysis of a Simply Supported Plate</td>
<td>3-9</td>
</tr>
<tr>
<td>F8: Clamped Circular Plate</td>
<td>3-10</td>
</tr>
<tr>
<td>F9: Frequencies of a Cylindrical Shell</td>
<td>3-11</td>
</tr>
<tr>
<td>F10: Symmetric Modes and Natural Frequencies of a Ring</td>
<td>3-12</td>
</tr>
</tbody>
</table>
Part 2 Verification Problems

F11A, F11B: Eigenvalues of a Triangular Wing ....................... 3-13
F12: Vibration of an Unsupported Beam ............................. 3-14
F13: Frequencies of a Solid Cantilever Beam ..................... 3-15
F14: Natural Frequency of Fluid ................................... 3-16
F16A, F16B: Vibration of a Clamped Wedge ....................... 3-17
F17: Lateral Vibration of an Axially Loaded Bar .................. 3-19
F18: Simply Supported Rectangular Plate ......................... 3-20
F19: Lowest Frequencies of Clamped Cylindrical Shell
for Harmonic No. = 6 ............................................... 3-21
F20A, F20B, F20C, F20D, F20E, F20F, F20H: Dynamic Analysis of
Cantilever Beam ..................................................... 3-22
F21: Frequency Analysis of a Right Circular Canal of Fluid with Variable
Depth ................................................................. 3-23
F22: Frequency Analysis of a Rectangular Tank of Fluid with
Variable Depth ....................................................... 3-25
F23: Natural Frequency of Fluid in a Manometer .................. 3-27
F24: Modal Analysis of a Piezoelectric Cantilever ................ 3-29
F25: Frequency Analysis of a Stretched Circular Membrane ...... 3-30
F26: Frequency Analysis of a Spherical Shell ...................... 3-31
F27A, F27B: Natural Frequencies of a Simply-Supported Square Plate .. 3-32
F28: Cylindrical Roof Shell ......................................... 3-33
F29A, B, C: Frequency Analysis of a Spinning Blade ............ 3-34

4 Buckling Analysis ................................................. 4-1

Introduction ......................................................... 4-1
B1: Instability of Columns ......................................... 4-2
B2: Instability of Columns ......................................... 4-3
B3: Instability of Columns ......................................... 4-4
B4: Simply Supported Rectangular Plate ............................ 4-5
B5A, B5B: Instability of a Ring .................................... 4-6
B6: Buckling Analysis of a Small Frame ......................... 4-7
B7A, B7B: Instability of Frames ................................... 4-9
B8: Instability of a Cylinder ....................................... 4-10
B9: Simply Supported Stiffened Plate ............................... 4-11
Contents

B10: Stability of a Rectangular Frame .......................... 4-12
B11: Buckling of a Stepped Column .............................. 4-13
B12: Buckling Analysis of a Simply Supported Composite Plate .... 4-14
B13: Buckling of a Tapered Column .............................. 4-15
B14: Buckling of Clamped Cylindrical Shell Under External Pressure
Using the Nonaxisymmetric Buckling Mode Option ............... 4-16
B15A, B15B: Buckling of Simply-Supported Cylindrical Shell Under Axial
Load ................................................................. 4-18

Index
Introduction

COSMOS/M software modules are continually in the process of extensive development, testing, and quality assurance checks. New features and capabilities incorporated into the system are rigorously tested using verification examples and in-house quality assurance problems. All verification problems are provided to the user along with the software, and they are made available in the COSMOS/M directory. There are more than 150 verification problems for analysis modules in the Basic System.

The purpose of this section is dual fold: to present many example problems that test a combination of capabilities offered in the COSMOS/M Basic System, and to provide a large number of verification problems that validate the basic modeling and analysis features. The first part of this manual presented several fully described and illustrated examples which cover few aspects of modeling and analysis limitations. This part provides examples on many other analysis features of the Basic System.

The input files for all verification problems are provided in separate folders (depending on the analysis type) in the “...\Vprobs” directory where “...” denotes the COSMOS/M directory.
Chapter 1  About the Verification Problems

<table>
<thead>
<tr>
<th>Folder</th>
<th>Analysis Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geostar</td>
<td>GEOSTAR modeling examples</td>
</tr>
<tr>
<td>Buckling</td>
<td>Linearized buckling analysis</td>
</tr>
<tr>
<td>AdvDynamics</td>
<td>Linear dynamic response analysis</td>
</tr>
<tr>
<td>Emagnetic</td>
<td>Electromagnetic analysis</td>
</tr>
<tr>
<td>Frequency</td>
<td>Frequency (modal) analysis</td>
</tr>
<tr>
<td>Fatigue</td>
<td>Fatigue analysis</td>
</tr>
<tr>
<td>Nonlinear</td>
<td>Nonlinear dynamic analysis</td>
</tr>
<tr>
<td>Static</td>
<td>Linear static stress analysis</td>
</tr>
<tr>
<td>Thermal</td>
<td>Thermal (heat transfer) analysis (linear)</td>
</tr>
<tr>
<td>FFE</td>
<td>FFE modules</td>
</tr>
<tr>
<td>HFS</td>
<td>High frequency electromagnetic simulation</td>
</tr>
</tbody>
</table>

To use the verification problems, enter GEOSTAR and at the GEO> prompt, execute the command **Load... (FILE)** from the File menu. The following pages show a listing of the verification problems based on analysis and element type’s.

### Classification by Analysis Type

<table>
<thead>
<tr>
<th>Analysis Type</th>
<th>Folder</th>
<th>Problem Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>Buckling Analysis</td>
<td>...\Vprobs\Buckling</td>
<td>B1, B2, B3, B4, B5A, B5B, B6, B7A, B7B, B8, B9, B10, B11, B12, B13, B14, B15A, B15B</td>
</tr>
</tbody>
</table>
## Classification by Element Type

<table>
<thead>
<tr>
<th>Element Name</th>
<th>Analysis Type</th>
<th>Problem Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>BEAM2D</td>
<td>Buckling</td>
<td>B10, B11, B13 S9A, S9B, S24, S41, S46, S47, S51, S52, S53, S54, S75, S76</td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td></td>
</tr>
<tr>
<td>BEAM3D</td>
<td>Buckling</td>
<td>B1, B2, B3, B6, B9 F3, F4, F5, F12, F17</td>
</tr>
<tr>
<td></td>
<td>Modal Analysis</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td>S7, S22, S23, S26, S27, S28, S33A, S33M, S34, S39, S43, S45, S55</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOUND</td>
<td>Linear Static</td>
<td>NONE</td>
</tr>
<tr>
<td>ELBOW</td>
<td>Linear Static</td>
<td>S15, S16A, S16B</td>
</tr>
<tr>
<td>GAP</td>
<td>Linear Static</td>
<td>S75, S76</td>
</tr>
<tr>
<td>GENSTIF</td>
<td>All</td>
<td>NONE</td>
</tr>
<tr>
<td>MASS</td>
<td>Modal Analysis</td>
<td>F1, F5, F6 S39</td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td></td>
</tr>
<tr>
<td>PIPE</td>
<td>Modal Analysis</td>
<td>F6</td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td>S16A, S16B</td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td></td>
</tr>
<tr>
<td>RBAR</td>
<td>Linear Static</td>
<td>F28</td>
</tr>
<tr>
<td>SHELL3</td>
<td>Buckling</td>
<td>B5</td>
</tr>
<tr>
<td></td>
<td>Modal Analysis</td>
<td>F11</td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td>S3A, S3B, S8, S30, S33A</td>
</tr>
<tr>
<td>SHELL3L</td>
<td>Linear Static</td>
<td>NONE</td>
</tr>
<tr>
<td>SHELL3T</td>
<td>Modal Analysis</td>
<td>F8, F16A</td>
</tr>
<tr>
<td>SHELL4</td>
<td>Buckling</td>
<td>B4, B7, B9 F7, F9, F10, F18, F27A, F27B</td>
</tr>
<tr>
<td></td>
<td>Modal Analysis</td>
<td>S20, S25, S33A, S33M, S36A, S36M, S42, S44A, S44B, S85</td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td>S21A, S31, S43, S59B, S71</td>
</tr>
</tbody>
</table>
### Classification by Element Type (Concluded)

<table>
<thead>
<tr>
<th>Element Name</th>
<th>Analysis Type</th>
<th>Problem Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>SHELL4T</td>
<td>Modal Analysis</td>
<td>F16B</td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td>S50C</td>
</tr>
<tr>
<td>SHELL9</td>
<td>Linear Static</td>
<td>S46B, S56, S57, S58, S59A, S60</td>
</tr>
<tr>
<td>SHELL9L</td>
<td>Linear Static</td>
<td>S21B, S29A, S31A, S59A</td>
</tr>
<tr>
<td>SHELLAX</td>
<td>Buckling</td>
<td>B8, B14, B15</td>
</tr>
<tr>
<td></td>
<td>Modal Analysis</td>
<td>F19, F25, F26</td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td>S18, S37, S79, S80, S81</td>
</tr>
<tr>
<td>SOLID</td>
<td>Modal Analysis</td>
<td>F13, F20E, F20F, F22</td>
</tr>
<tr>
<td>SHELL6</td>
<td>Buckling</td>
<td>B5B, B7B</td>
</tr>
<tr>
<td></td>
<td>Modal Analysis</td>
<td>F7B, F11B, F20H</td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td>S6B, S20B, S42B, S50I</td>
</tr>
<tr>
<td>SOLIDL</td>
<td>Linear Static</td>
<td>S29, S35B, S59C</td>
</tr>
<tr>
<td>SOLIDPZ</td>
<td>Modal Analysis</td>
<td>F24</td>
</tr>
<tr>
<td>SPRING</td>
<td>Modal Analysis</td>
<td>NONE</td>
</tr>
<tr>
<td>TETRA10</td>
<td>Modal Analysis</td>
<td>F20D</td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td>S50E</td>
</tr>
<tr>
<td>TETRA4</td>
<td>Linear Static</td>
<td>NONE</td>
</tr>
<tr>
<td>TETRA4R</td>
<td>Linear Static</td>
<td>S50H, S58B, S74</td>
</tr>
<tr>
<td></td>
<td>Modal Analysis</td>
<td>F20G</td>
</tr>
<tr>
<td>TRIANG</td>
<td>Modal Analysis</td>
<td>F20C</td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td>S50D, S64A, S64B, S68, S69, S78, S84</td>
</tr>
<tr>
<td>TRUSS2D</td>
<td>Buckling</td>
<td>B6</td>
</tr>
<tr>
<td></td>
<td>Modal Analysis</td>
<td>F14</td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td>S4, S32A, S32B, S32C, S32D, S32M, S40, S63, S76</td>
</tr>
<tr>
<td>TRUSS3D</td>
<td>Modal Analysis</td>
<td>F1</td>
</tr>
<tr>
<td></td>
<td>Linear Static</td>
<td>S1, S12, S13, S14A, S14B, S22, S26, S33A, S33M</td>
</tr>
</tbody>
</table>
2

Linear Static Analysis

Introduction

This chapter contains verification problems to demonstrate the accuracy of the Linear Static Analysis module STAR.

<table>
<thead>
<tr>
<th>List of Linear Static Verification Problems</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>S1: Pin Jointed Truss</td>
<td>2-6</td>
</tr>
<tr>
<td>S2: Long Thick-Walled Cylinder</td>
<td>2-7</td>
</tr>
<tr>
<td>S3A, S3B: Simply Supported Rectangular Plate</td>
<td>2-9</td>
</tr>
<tr>
<td>S4: Thermal Stress Analysis of a Truss Structure</td>
<td>2-10</td>
</tr>
<tr>
<td>S5: Thermal Stress Analysis of a 2D Structure</td>
<td>2-12</td>
</tr>
<tr>
<td>S6A, S6B: Deflection of a Cantilever Beam</td>
<td>2-13</td>
</tr>
<tr>
<td>S7: Beam Stresses and Deflections</td>
<td>2-14</td>
</tr>
<tr>
<td>S8: Tip Displacements of a Circular Beam</td>
<td>2-15</td>
</tr>
<tr>
<td>S9A: Clamped Beam Subject to Imposed Displacement</td>
<td>2-16</td>
</tr>
<tr>
<td>S9B: Clamped Beam Subject to Imposed Rotation</td>
<td>2-18</td>
</tr>
<tr>
<td>S10A, S10B: Bending of a Solid Beam</td>
<td>2-19</td>
</tr>
<tr>
<td>S11: Thermal Stress Analysis of a 3D Structure</td>
<td>2-21</td>
</tr>
<tr>
<td>S12: Deflection of a Hinged Support</td>
<td>2-22</td>
</tr>
<tr>
<td>S13: Statically Indeterminate Reaction Force Analysis</td>
<td>2-23</td>
</tr>
</tbody>
</table>
## List of Linear Static Verification Problems (Continued)

<table>
<thead>
<tr>
<th>Problem Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>S14A, S14B: Space Truss with Vertical Load</td>
<td>2-24</td>
</tr>
<tr>
<td>S15: Out-of-Plane Bending of a Curved Bar</td>
<td>2-25</td>
</tr>
<tr>
<td>S16A, S16B: Curved Pipe Deflection</td>
<td>2-26</td>
</tr>
<tr>
<td>S17: Rectangular Plate Under Triangular Thermal Loading</td>
<td>2-28</td>
</tr>
<tr>
<td>S18: Hemispherical Dome Under Unit Moment Around Free Edge</td>
<td>2-29</td>
</tr>
<tr>
<td>S19: Hollow Thick-walled Cylinder Subject to Temperature and Pressure</td>
<td>2-30</td>
</tr>
<tr>
<td>S20A, S20B: Cylindrical Shell Roof</td>
<td>2-31</td>
</tr>
<tr>
<td>S21A, S21B: Antisymmetric Cross-Ply Laminated Plate (SHELL4L)</td>
<td>2-33</td>
</tr>
<tr>
<td>S22: Thermally Loaded Support Structure</td>
<td>2-35</td>
</tr>
<tr>
<td>S23: Thermal Stress Analysis of a Frame</td>
<td>2-36</td>
</tr>
<tr>
<td>S24: Thermal Stress Analysis of a Simple Frame</td>
<td>2-38</td>
</tr>
<tr>
<td>S25: Torsion of a Square Box Beam</td>
<td>2-39</td>
</tr>
<tr>
<td>S26: Beam With Elastic Supports and a Hinge</td>
<td>2-41</td>
</tr>
<tr>
<td>S27: Frame Analysis with Combined Loads</td>
<td>2-43</td>
</tr>
<tr>
<td>S28: Cantilever Unsymmetric Beam</td>
<td>2-45</td>
</tr>
<tr>
<td>S29A, S29B: Square Angle-Ply Composite Plate Under Sinusoidal Loading</td>
<td>2-47</td>
</tr>
<tr>
<td>S30: Effect of Transverse Shear on Maximum Deflection</td>
<td>2-48</td>
</tr>
<tr>
<td>S31: Square Angle-Ply Composite Plate Under Sinusoidal Loading</td>
<td>2-50</td>
</tr>
<tr>
<td>S33A, S33M: Substructure of an Airplane (Wing)</td>
<td>2-53</td>
</tr>
<tr>
<td>S34: Tie Rod with Lateral Loading</td>
<td>2-54</td>
</tr>
<tr>
<td>S35A, S35B: Spherical Cap Under Uniform Pressure (Solid)</td>
<td>2-56</td>
</tr>
<tr>
<td>S36A, S36B: Substructure of a Simply Supported Plate</td>
<td>2-58</td>
</tr>
<tr>
<td>S37: Hyperboloidal Shell Under Uniform Ring Load Around Free Edge</td>
<td>2-60</td>
</tr>
<tr>
<td>S38: Rotating Solid Disk</td>
<td>2-62</td>
</tr>
<tr>
<td>S39: Unbalanced Rotating Flywheel</td>
<td>2-63</td>
</tr>
<tr>
<td>S40: Truss Structure Subject to a Concentrated Load</td>
<td>2-64</td>
</tr>
<tr>
<td>S41: Reactions of a Frame Structure</td>
<td>2-65</td>
</tr>
</tbody>
</table>
**List of Linear Static Verification Problems (Continued)**

<table>
<thead>
<tr>
<th>Problem</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>S42A, S42B</td>
<td>Reactions and Deflections of a Cantilever Beam</td>
<td>2-66</td>
</tr>
<tr>
<td>S43</td>
<td>Bending of a T Section Beam</td>
<td>2-67</td>
</tr>
<tr>
<td>S44A, S44B</td>
<td>Bending of a Circular Plate with a Center Hole</td>
<td>2-68</td>
</tr>
<tr>
<td>S45</td>
<td>Eccentric Frame</td>
<td>2-70</td>
</tr>
<tr>
<td>S46, S46A, S46B</td>
<td>Bending of a Cantilever Beam</td>
<td>2-71</td>
</tr>
<tr>
<td>S47, S47A, S47B</td>
<td>Bending of a Cantilever Beam</td>
<td>2-73</td>
</tr>
<tr>
<td>S48</td>
<td>Rotation of a Tank of Fluid (PLANE2D Fluid)</td>
<td>2-75</td>
</tr>
<tr>
<td>S49A, S49B</td>
<td>Acceleration of a Tank of Fluid (PLANE2D Fluid)</td>
<td>2-77</td>
</tr>
<tr>
<td>S50A, S50B, S50C, S50D, S50F, S50G, S50H, S50I</td>
<td>Deflection of a Curved Beam</td>
<td>2-79</td>
</tr>
<tr>
<td>S51</td>
<td>Gable Frame with Hinged Supports</td>
<td>2-80</td>
</tr>
<tr>
<td>S52</td>
<td>Support Reactions for a Beam with Intermediate Forces and Moments</td>
<td>2-81</td>
</tr>
<tr>
<td>S53</td>
<td>Beam Analysis with Intermediate Loads</td>
<td>2-83</td>
</tr>
<tr>
<td>S54</td>
<td>Analysis of a Plane Frame with Beam Loads</td>
<td>2-85</td>
</tr>
<tr>
<td>S55</td>
<td>Laterally Loaded Tapered Beam</td>
<td>2-86</td>
</tr>
<tr>
<td>S56</td>
<td>Circular Plate Under a Concentrated Load (SHELL9 Element)</td>
<td>2-87</td>
</tr>
<tr>
<td>S57</td>
<td>Test of a Pinched Cylinder with Diaphragm (SHELL9 Element)</td>
<td>2-89</td>
</tr>
<tr>
<td>S58A, S58B, S58C</td>
<td>Deflection of a Twisted Beam with Tip Force (SHELL9 and TETRA4R Elements)</td>
<td>2-91</td>
</tr>
<tr>
<td>S59A, S59B, S59C</td>
<td>Sandwich Square Plate Under Uniform Loading (SHELL9L)</td>
<td>2-93</td>
</tr>
<tr>
<td>S60</td>
<td>Clamped Square Plate Under Uniform Loading</td>
<td>2-95</td>
</tr>
<tr>
<td>S61</td>
<td>Single-Edge Cracked Bend Specimen, Evaluation of Stress Intensity Factor Using Crack Element</td>
<td>2-97</td>
</tr>
<tr>
<td>S62</td>
<td>Plate with Central Crack</td>
<td>2-98</td>
</tr>
<tr>
<td>S63</td>
<td>Cyclic Symmetry Analysis of a Hexagonal Frame</td>
<td>2-99</td>
</tr>
<tr>
<td>S64A, S64B</td>
<td>Cyclic Symmetry</td>
<td>2-100</td>
</tr>
<tr>
<td>S65</td>
<td>Fluid-Structure Interaction, Rotation of a Tank of Fluid</td>
<td>2-101</td>
</tr>
<tr>
<td>S66</td>
<td>Fluid-Structure Interaction, Acceleration of a Tank of Fluid</td>
<td>2-103</td>
</tr>
<tr>
<td>S67</td>
<td>MacNeal-Harder Test</td>
<td>2-105</td>
</tr>
<tr>
<td>S68</td>
<td>P-Method Solution of a Square Plate with Hole</td>
<td>2-106</td>
</tr>
</tbody>
</table>
List of Linear Static Verification Problems (Concluded)

<table>
<thead>
<tr>
<th>Problem ID</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>S69</td>
<td>P-Method Analysis of an Elliptic Membrane Under Pressure</td>
<td>2-107</td>
</tr>
<tr>
<td>S70</td>
<td>Thermal Analysis with Temperature Dependent Material</td>
<td>2-108</td>
</tr>
<tr>
<td>S71</td>
<td>Sandwich Beam with Concentrated Load</td>
<td>2-109</td>
</tr>
<tr>
<td>S74</td>
<td>Constant Stress Patch Test (TETRA4R)</td>
<td>2-110</td>
</tr>
<tr>
<td>S75</td>
<td>Analysis of a Cantilever Beam with Gaps, Subject to Different Loading Conditions</td>
<td>2-111</td>
</tr>
<tr>
<td>S76</td>
<td>Simply Supported Beam Subject to Pressure from a Rigid Parabolic Shaped Piston</td>
<td>2-113</td>
</tr>
<tr>
<td>S77</td>
<td>Bending of a Solid Beam Using Direct Material Matrix Input</td>
<td>2-115</td>
</tr>
<tr>
<td>S78</td>
<td>P-Adaptive Analysis of a Square Plate with a Circular Hole</td>
<td>2-117</td>
</tr>
<tr>
<td>S79</td>
<td>Hemispherical Shell Under Unit Moment Around Free Edge</td>
<td>2-119</td>
</tr>
<tr>
<td>S80</td>
<td>Axisymmetric Hyperbolic Shell Under a Cosine Harmonic Loading on the Free Edge</td>
<td>2-120</td>
</tr>
<tr>
<td>S81</td>
<td>Circular Plate Under Non-Axisymmetric Load</td>
<td>2-121</td>
</tr>
<tr>
<td>S82</td>
<td>Twisting of a Long Solid Shaft</td>
<td>2-123</td>
</tr>
<tr>
<td>S83</td>
<td>Bending of a Long Solid Shaft</td>
<td>2-125</td>
</tr>
<tr>
<td>S84</td>
<td>Submodeling of a Plate</td>
<td>2-127</td>
</tr>
<tr>
<td>S85</td>
<td>Plate on Elastic Foundation</td>
<td>2-128</td>
</tr>
<tr>
<td>S86</td>
<td>Plate with Coupled Degrees of Freedom</td>
<td>2-129</td>
</tr>
<tr>
<td>S87</td>
<td>Gravity Loading of ELBOW Element</td>
<td>2-130</td>
</tr>
<tr>
<td>S88A, S88B</td>
<td>Single-Edge Cracked Bend Specimen, Evaluation of Stress Intensity Factor Using the J-integral</td>
<td>2-131</td>
</tr>
<tr>
<td>S89A, S89B</td>
<td>Slant-Edge Cracked Plate, Evaluation of Stress Intensity Factors Using the J-integral</td>
<td>2-133</td>
</tr>
<tr>
<td>S90A, S90B</td>
<td>Penny-Shaped Crack in Round Bar, Evaluation of Stress Intensity Factor Using the J-integral</td>
<td>2-135</td>
</tr>
<tr>
<td>S91</td>
<td>Crack Under Thermal Stresses, Evaluation of Stress Intensity Using the J-integral</td>
<td>2-137</td>
</tr>
<tr>
<td>S92A, S92B</td>
<td>Simply Supported Rectangular Plate, Using Direct Material Matrix Input</td>
<td>2-139</td>
</tr>
<tr>
<td>S93</td>
<td>Accelerating Rocket</td>
<td>2-141</td>
</tr>
<tr>
<td>S94A, S94B, S94C</td>
<td>P-Method Solution of a Square Plate with Small Hole</td>
<td>2-143</td>
</tr>
<tr>
<td>S95A, S95B, S95C</td>
<td>P-Method Solution of a U-Shaped Circumferential Groove in Circular Shaft</td>
<td>2-146</td>
</tr>
</tbody>
</table>
Part 2 Verification Problems

S96A, S96B, S96C: P-Method Solution of a Square Plate with an Elliptical Hole 2-149
S1: Pin Jointed Truss

TYPE:
Static analysis, truss element (TRUSS3D).

REFERENCE:

PROBLEM:
A 50 lb load is supported by three bars which are attached to a ceiling as shown. Determine the stress in each bar.

GIVEN:
Area of each bar = 1 in²
E = 30 x 10⁶ psi

<table>
<thead>
<tr>
<th></th>
<th>σ₁-4, psi</th>
<th>σ₂-4, psi</th>
<th>σ₃-₄, psi</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>10.40</td>
<td>31.20</td>
<td>22.90</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>10.39</td>
<td>31.18</td>
<td>22.91</td>
</tr>
</tbody>
</table>

Figure S1-1

Problem Sketch and Finite Element Model
S2: Long Thick-Walled Cylinder

TYPE:
Static analysis, 2D axisymmetric elements (PLANE2D).

REFERENCE:

PROBLEM:
Calculate the radial stresses for an infinitely long, thick walled cylinder subjected to an internal pressure $p$.

GIVEN:
\begin{align*}
a &= 100 \text{ in} \\
b &= 115 \text{ in} \\
p &= 1000 \text{ psi} \\
E &= 30 \times 10^6 \text{ psi} \\
\nu &= 0.3
\end{align*}

MODELING HINTS:
The model is meshed with three elements through the thickness and three elements along the length.

COMPARISON OF RESULTS:
\begin{center}
\begin{tabular}{|c|c|c|}
\hline
$r$ (Radial Distance) (in) & Radial Stress $\sigma_r$ (psi) & \\
& Theory & COSMOS/M & \\
\hline
102.5 (Element 1) & -802.40 & -802.51 & \\
107.5 (Element 2) & -447.75 & -447.84 & \\
112.5 (Element 3) & -139.34 & -139.42 & \\
\hline
\end{tabular}
\end{center}
Figure S2-1

Problem Sketch

Finite Element Model
**S3A, S3B: Simply Supported Rectangular Plate**

**TYPE:**
Static analysis, 3-node thin plate element (SHELL3).

**REFERENCE:**

**PROBLEM:**
Calculate the deflection at the center of a simply supported isotropic plate subjected to (A) concentrated load $F$, (B) uniform pressure $(P)$.

**GIVEN:**
- $E = 30,000,000$ psi
- $\nu = 0.3$
- $h = 1$ in
- $a = b = 40$ in
- $F = 400$ lb
- $p = 1$ psi

**MODELING HINTS:**
Due to double symmetry in geometry and loads, only a quarter of the plate is modeled.

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Case</th>
<th>X (in)</th>
<th>Y (in)</th>
<th>Deflection at Node 25 (UZ)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td><strong>Theory</strong></td>
</tr>
<tr>
<td>A</td>
<td>20</td>
<td>20</td>
<td>0.0270230 in</td>
</tr>
<tr>
<td>B</td>
<td>20</td>
<td>20</td>
<td>0.0037827 in</td>
</tr>
</tbody>
</table>
S4: Thermal Stress Analysis of a Truss Structure

**TYPE:**
Linear thermal stress analysis, truss elements (TRUSS2D).

**REFERENCE:**

**PROBLEM:**
Determine the member forces in truss structure shown in the figure subject to a 50°F rise in temperature at the top chords (elements 13 and 14).

**GIVEN:**
\( E = 30 \times 10^6 \text{ psi} \)

\[ \text{Coefficient of thermal expansion} = \alpha = 0.65 \times 10^{-5}/\text{°F} \]

\( L(\text{ft})/A(\text{in}^2) = 1 \) for all members

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Member Forces (kips)</th>
<th>Members</th>
<th>Theory</th>
<th>COSMOS/M</th>
<th>Members</th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
<td>8</td>
<td>35.1</td>
<td>35.1</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0</td>
<td>0</td>
<td>9</td>
<td>0</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-21.1</td>
<td>-21.1</td>
<td>10</td>
<td>0</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0</td>
<td>0</td>
<td>11</td>
<td>35.1</td>
<td>35.1</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>0</td>
<td>0</td>
<td>12</td>
<td>0</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>-28.1</td>
<td>-28.1</td>
<td>13</td>
<td>0</td>
<td>0</td>
<td></td>
</tr>
</tbody>
</table>

COSMOS/M results are calculated by listing element stress results and multiplying by the corresponding area.
Part 2  Verification Problems

Figure S4-1

Problem Sketch and Finite Element Model

4 \times 24 \text{ ft} = 96 \text{ ft}
S5: Thermal Stress Analysis of a 2D Structure

**TYPE:**
Linear thermal stress analysis, 2D elements (plane strain, PLANE2D).

**PROBLEM:**
Determine the displacements and stresses of the plane strain problem shown in figure due to a uniform temperature rise.

**GIVEN:**
- \( E = 30 \times 10^6 \text{ psi} \)
- \( \alpha = 0.65 \times 10^{-5}/\degree \text{ F} \)
- \( \nu = 0.25 \)
- \( T = 100\degree \text{ F} \)
- \( L = 1 \text{ in} \)

**COMPARISON OF RESULTS:**
Displacements at Nodes (2, 4, and 6)

<table>
<thead>
<tr>
<th></th>
<th>Y-Displacement (in)</th>
<th>SX-Stress (psi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>0.001083</td>
<td>-26000.0</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>0.001083</td>
<td>-26000.1</td>
</tr>
</tbody>
</table>

**Figure S5-1**

Problem Sketch and Finite Element Model
Part 2  Verification Problems

S6A, S6B: Deflection of a Cantilever Beam

TYPE:
Static analysis, plane stress element PLANE2D and SHELL6.

PROBLEM:
A cantilever beam is subjected to a concentrated load at the free end. Determine the
deflections at the free end and the average shear stress.

GIVEN:

\begin{align*}
E &= 30 \times 10^6 \text{ psi} \\
L &= 10 \text{ in} \\
h &= 1 \text{ in} \\
A &= 0.1 \text{ in}^2 \\
\nu &= 0 \\
P &= 1 \text{ lb}
\end{align*}

COMPARISON OF RESULTS:

\begin{center}
\begin{tabular}{|c|c|c|}
\hline
Theory & Max. Deflection in the Y-direction & Shear Stress (psi) \\
\hline
Theory & -0.001333 & -10.0 \\
PLANE2D & -0.001337 & -10.0* \\
SHELL6 (Curved) & -0.0013398 & -9.820667* \\
SHELL6 (Assembled) & -0.00072411 & -8.530667* \\
\hline
\end{tabular}
\end{center}

* averaged results of nodes at the free edge

Figure S6-1
S7: Beam Stresses and Deflections

TYPE:
Static analysis, beam elements (BEAM3D).

REFERENCE:

PROBLEM:
A standard 30” Wide Flange beam is supported as shown below and loaded on the overhangs by a uniformly distributed load of 10,000 lb per ft. Determine the maximum stress in the middle portion of the beam and the deflection at the center of the beam.

GIVEN:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Area</td>
<td>50.65 in²</td>
</tr>
<tr>
<td>E</td>
<td>30 x 10⁶ psi</td>
</tr>
<tr>
<td>p</td>
<td>10,000 lb/ft</td>
</tr>
</tbody>
</table>

COMPARISON OF RESULTS:
At the middle of the span (node 3):

<table>
<thead>
<tr>
<th></th>
<th>σ max (Psi)</th>
<th>δ (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>11400.0</td>
<td>0.182</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>11400.0</td>
<td>0.182</td>
</tr>
</tbody>
</table>

MODELING HINTS:
Use consistent length units. A half-model has been used because of symmetry. Resultant force and moment have been applied at node 2 instead of distributed load.

Figure S7-1
**Part 2  Verification Problems**

---

**S8: Tip Displacements of a Circular Beam**

**TYPE:**
Static analysis, thin or thick shell element (SHELL3).

**REFERENCE**

**PROBLEM:**
Determine the deflections in X, Y direction of a circular beam fixed at one end and free at the other end, when subjected to a force along X direction at force end.

**GIVEN:**
\[ \begin{align*} 
E &= 30 \text{E}6 \text{ psi} \\
\nu &= 0 \\
b &= 4 \text{ in} \\
h &= 1 \text{ in} \\
R &= 10 \text{ in} \\
F &= 200 \text{ lb} 
\end{align*} \]

**COMPARISON OF RESULTS:**
The loaded end.

<table>
<thead>
<tr>
<th></th>
<th>Displacement (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>X</strong></td>
<td><strong>Y</strong></td>
</tr>
<tr>
<td>Theory</td>
<td>0.712E-2 0.99E-2</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>0.718E-2 0.99E-2</td>
</tr>
</tbody>
</table>

---

*Problem Sketch and Finite Element Model*
Chapter 2  Linear Static Analysis

---

**S9A: Clamped Beam Subject to Imposed Displacement**

**TYPE:**
Static analysis, beam elements (BEAM2D).

**REFERENCE**

**PROBLEM:**
Determine the end forces of a clamped beam due to a 1 inch settlement at the right end.

**GIVEN:**
\[ \begin{align*}
E &= 30 \times 10^6 \text{ psi} \\
L &= 80 \text{ in} \\
A &= 4 \text{ in}^2 \\
I &= 1.33 \text{ in}^4 \\
h &= 2 \text{ in}
\end{align*} \]

**ANALYTICAL SOLUTION:**
Reaction: \( R = \frac{-12EI}{L^3} \)
Moment: \( M = \frac{6EI}{L^2} \)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>Imposed Displacement (in)</td>
<td>-1.0</td>
<td>-1.0</td>
</tr>
<tr>
<td>End Shear (lb)</td>
<td>-937.5</td>
<td>-937.5</td>
</tr>
<tr>
<td>End Moment (lb-in)</td>
<td>-37,500.0</td>
<td>-37,500.0</td>
</tr>
</tbody>
</table>
Figure S9A-1

Problem Sketch

Finite Element Model
**Chapter 2  Linear Static Analysis**

---

**S9B: Clamped Beam Subject to Imposed Rotation**

**TYPE:**
Static analysis, beam elements (BEAM2D).

**REFERENCE:**

**PROBLEM:**
Determine the end forces of a clamped-clamped beam due to a 1 radian imposed rotation at the right end.

**GIVEN:**

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$E$</td>
<td>$30 \times 10^6$ psi</td>
</tr>
<tr>
<td>$l$</td>
<td>80 in</td>
</tr>
<tr>
<td>$A$</td>
<td>4 in$^2$</td>
</tr>
<tr>
<td>$I$</td>
<td>1.3333 in$^4$</td>
</tr>
<tr>
<td>$h$</td>
<td>2 in</td>
</tr>
</tbody>
</table>

**ANALYTICAL SOLUTION:**

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Reaction:</td>
<td>$R = -6EI / L^2$</td>
</tr>
<tr>
<td>Moment:</td>
<td>$M = 4EI / L$</td>
</tr>
</tbody>
</table>

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>Imposed Rotation (1 rad)</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>End Shear</td>
<td>-37,500</td>
<td>-37,500</td>
</tr>
<tr>
<td>End Moment</td>
<td>-2,000,000</td>
<td>-2,000,000</td>
</tr>
</tbody>
</table>

Figure S9B-1

---

**Problem Sketch**

---
S10A, S10B: Bending of a Solid Beam

TYPE:
Static analysis, SOLID element.

REFERENCE:

PROBLEM:
A beam of length L and height h is built-in at one end and loaded at free end: (A) with a shear force F, and (B) a moment M. Determine the deflection at the free end.

GIVEN:
\[ \begin{align*}
L &= 10 \text{ in} \\
h &= 2 \text{ in} \\
E &= 30 \times 10^6 \text{ psi} \\
\nu &= 0 \\
F &= 300 \text{ lb} \\
M &= 2000 \text{ in-lb}
\end{align*} \]

MODELING HINTS:
Two load cases have been used (S10A, S10B).

1. Four forces equal to F/4 have been applied at nodes 21, 22, 23, and 24 in xz direction (S10A), and,
2. Two couples equal M/2 have been applied at nodes 21, 22, 23 and 24 (S10B).

COMPARISON OF RESULTS:
Displacement in Z-direction (in) (node 21-24):

<table>
<thead>
<tr>
<th></th>
<th>S10A</th>
<th>S10B</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>0.00500</td>
<td>0.00495</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>0.005007</td>
<td>-0.00495</td>
</tr>
</tbody>
</table>
Chapter 2   Linear Static Analysis

Figure S10A-1

Problem Sketch

Finite Element Model
S11: Thermal Stress Analysis of a 3D Structure

**TYPE:**
Linear thermal stress analysis, 3D SOLID element.

**PROBLEM:**
Determine the displacements of the three-dimensional structure shown below due to a uniform temperature rise.

**GIVEN:**
- \( E = 3 \times 10^7 \text{ psi} \)
- \( \alpha = 0.65 \times 10^{-5}/\degree\text{F} \)
- \( \nu = 0.25 \)
- \( T = 100\degree \text{F} \)
- \( L = 1\text{ in} \)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>5, 6, 7, 8</th>
<th>9, 10, 11, 12</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>0.000650</td>
<td>0.001300</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>0.000650</td>
<td>0.001300</td>
</tr>
</tbody>
</table>

---

Figure S11-1

Problem Sketch and Finite Element Model
**S12: Deflection of a Hinged Support**

**TYPE:**
Static analysis, truss element (TRUSS3D).

**REFERENCE:**

**PROBLEM:**
A structure consisting of two equal steel bars, 15 feet long and with hinged ends, is submitted to the action of a vertical load $P$. Determine the forces in the members $AB$ and $BC$ along with the vertical deflection at $B$.

**GIVEN:**
- $P = 5000$ lbs
- $\theta = 30^\circ$
- $AB = BC = 15$ ft
- $E = 30 \times 10^6$ psi
- Cross-sectional area = $0.5$ in$^2$

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vertical Deflection at $B$ in inches</td>
<td>0.12</td>
<td>0.12</td>
</tr>
<tr>
<td>Forces in Members $AB$ and $BC$ in lbs</td>
<td>5000</td>
<td>5000</td>
</tr>
</tbody>
</table>

**Figure S12-1**

Problem Sketch and Finite Element Model
S13: Statically Indeterminate Reaction Force Analysis

**TYPE:**
Static analysis, truss elements (TRUSS3D).

**REFERENCE:**

**PROBLEM:**
A prismatic bar with built-in ends is loaded axially at two intermediate cross-sections by forces $\mathbf{F}_1$ and $\mathbf{F}_2$. Determine the reaction forces $R_1$ and $R_2$.

**GIVEN:**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a$</td>
<td>$b = 0.3 \text{ L}$</td>
</tr>
<tr>
<td>$L$</td>
<td>$10 \text{ in}$</td>
</tr>
<tr>
<td>$F_1$</td>
<td>$2F_2 = 1000 \text{ lb}$</td>
</tr>
<tr>
<td>$E$</td>
<td>$30 \times 10^6 \text{ psi}$</td>
</tr>
</tbody>
</table>

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>$R_1 \text{ lbs}$</th>
<th>$R_2 \text{ lbs}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>900</td>
<td>600</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>900</td>
<td>600</td>
</tr>
</tbody>
</table>

Figure S13-1
Chapter 2  Linear Static Analysis

S14A, S14B: Space Truss with Vertical Load

TYPE:
Static analysis, truss elements (TRUSS3D).

REFERENCE:

PROBLEM:
The simple space truss shown in the figure below consists of two panels ABCD and ABEF, attached to a vertical wall at points C, D, E, F, the panel ABCD being in a horizontal plane. All bars have the same cross-sectional area, A, and the same modulus of elasticity, E.

Calculate:
1. The axial force produced in the redundant bar AD by the vertical load \( P = 1 \text{ kip} \) at joint A (S14A).
2. The thermal force induced in the bar AD if there is a uniform rise in temperature of 50° F (S14B).

GIVEN:
\[ E = 30 \times 10^6 \text{ psi} \]
\[ \alpha = 6.5 \times 10^{-6}/\degree \text{ F} \]
\[ A = 1 \text{ in}^2 \]
\[ L = 4 \text{ ft} \]

COMPARISON OF RESULTS:
For Element 2:

<table>
<thead>
<tr>
<th></th>
<th>S14A</th>
<th>S14B</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>56.0 lb</td>
<td>-1259.0 lb</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>55.92 lb</td>
<td>-1292.4 lb</td>
</tr>
</tbody>
</table>
**S15: Out-of-Plane Bending of a Curved Bar**

**TYPE:**
Static analysis, curved elbow element (ELBOW).

**REFERENCE:**

**PROBLEM:**
A portion of a horizontal circular ring, built-in at A, is loaded by a vertical load \( P \) applied at the end B. The ring has a solid circular cross-section of diameter \( d \). Determine the deflection at end B, and the maximum bending stress.

**GIVEN:**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( P )</td>
<td>50 lb</td>
</tr>
<tr>
<td>( r )</td>
<td>100 in</td>
</tr>
<tr>
<td>( d )</td>
<td>2 in</td>
</tr>
<tr>
<td>( E )</td>
<td>( 30 \times 10^6 ) psi</td>
</tr>
<tr>
<td>( \alpha )</td>
<td>90°</td>
</tr>
<tr>
<td>( \nu )</td>
<td>0.3</td>
</tr>
</tbody>
</table>

**COMPARISON OF RESULTS**

<table>
<thead>
<tr>
<th>Method</th>
<th>( \delta_z ), inch</th>
<th>( \sigma_{Bend} ), psi</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>-2.648</td>
<td>6366.0</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>-2.650</td>
<td>6366.2</td>
</tr>
</tbody>
</table>

**MODELING HINTS:**

COSMOS/M does not yet have a curved beam element, although this element will be incorporated into the program shortly. Hence, the curved elbow element is used to model this problem. Therefore, it is necessary to use equivalent thickness \( t \) which is equal to the radius of the solid rod.

---

Figure S15-1
S16A, S16B: Curved Pipe Deflection

TYPE:
Static analysis, elbow element (ELBOW).

REFERENCE:

PROBLEM:
Calculate deflections x and y for a curved pipe shown in the figure subjected to:
1. Moment $M_z = 3 \times 10^6$ lb-in and internal pressure $p = 900$ psi (S16A).
2. Internal pressure $p = 900$ psi (S16B).

GIVEN:
- $E = 30 \times 10^6$ psi
- $\nu = 0.3$
- $R = 72$ in
- Thickness = 1.031 in
- Outer diameter of pipe = 20 in

COMPARISON OF RESULTS:
Blake gives the following results for a 90 curved member. These results do not include the effects of distortion of the cross-section and internal pressure.

\[
\delta_y = \frac{M_z R^2}{EI} = 0.187039 \text{ in}
\]
\[
\hat{\delta}_y = \frac{M_z R^2}{EI (P/2-1)} = 0.106761 \text{ in}
\]

The pipe flexibility factor is given by

\[
K_p = 1.65/h\{1 + 6P/Eh\} (R/t)^{4/3}, \text{ where } h = tR/t^2
\]

for $p = 900$ psi, $K_p = 1.8814761$

To obtain the nodal deflections for case 1, the deflections calculated by Blake's formulas must be multiplied by $k_p$ and added to the deflections produced by the internal pressure.
Part 2 Verification Problems

S16A

<table>
<thead>
<tr>
<th></th>
<th>δx, inch</th>
<th>δy, inch</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>0.37035</td>
<td>0.20515</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>0.37034</td>
<td>0.20515</td>
</tr>
</tbody>
</table>

S16B

<table>
<thead>
<tr>
<th></th>
<th>δx, inch</th>
<th>δy, inch</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>1.84356 x 10^{-2}</td>
<td>4.2873 x 10^{-3}</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>1.84355 x 10^{-2}</td>
<td>4.28043 x 10^{-3}</td>
</tr>
</tbody>
</table>

Figure S16A-1

Problem Sketch and Finite Element Model
S17: Rectangular Plate Under Triangular Thermal Loading

**TYPE:**
Linear thermal stress analysis, 2D elements (plane stress analysis, PLANE2D).

**REFERENCE:**

**PROBLEM:**
A finite rectangular plate is subjected to a temperature distribution in only one direction as shown in figure. Determine the normal stress at point A.

**GIVEN:**
\[ \begin{align*}
    a &= 15 \text{ in} \\
    b &= 10 \text{ in} \\
    T_o &= -100 \degree F \\
    t &= 1 \text{ in} \\
    E &= 30 \times 10^6 \text{ psi} \\
    \alpha &= 0.65 \times 10^{-5} \text{ in/in/} \degree F
\end{align*} \]

**MODELING HINTS:**
Due to the double symmetry in geometry and loading, only one quarter of the plate was analyzed.

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Reference</th>
<th>Method 1</th>
<th>Method 2</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.42</td>
<td>0.40</td>
<td>0.437</td>
</tr>
</tbody>
</table>
S18: Hemispherical Dome Under Unit Moment Around Free Edge

**TYPE:**
Static linear analysis, axisymmetric shell element (SHELLAX).

**REFERENCE:**

**PROBLEM:**
Determine the horizontal displacement of a hemispherical shell under uniform unit moment around the free edge.

**GIVEN:**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>R</td>
<td>100 in</td>
</tr>
<tr>
<td>r</td>
<td>50 in</td>
</tr>
<tr>
<td>E</td>
<td>$1 \times 10^7$ psi</td>
</tr>
<tr>
<td>v</td>
<td>0.33</td>
</tr>
<tr>
<td>t</td>
<td>1 in</td>
</tr>
<tr>
<td>M</td>
<td>1 in lb</td>
</tr>
</tbody>
</table>

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Reference</th>
<th>Horizontal Displacement (Node 29) (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference</td>
<td>1.580 E-5</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>1.589 E-5</td>
</tr>
</tbody>
</table>

**MODELING HINTS:**
Nodal spacing is shown in the Figure. For convenience, cylindrical coordinate system is chosen for node generation. It is important to note that nodal load is to be specified per unit radian which in this case is 50 in lb/rad.

**Figure S18-1**

Finite Element Model

Problem Sketch
Chapter 2  Linear Static Analysis

S19: Hollow Thick-walled Cylinder Subject to Temperature and Pressure

TYPE:
Static analysis, 2D axisymmetric element (PLANE2D).

REFERENCE:

PROBLEM:
The hollow cylinder in plane strain is subjected to two independent load conditions.
1. An internal pressure.
2. A steady state axisymmetric temperature distribution given by the equation:
   \[ T(r) = (\frac{Ta}{\ln(b/a)}) \cdot \ln(b/r) \]
   where \( T_a \) is the temperature of the inner surface and \( T(r) \) is the temperature at any radius.

GIVEN:
- \( E = 30 \times 10^6 \) psi
- \( a = 1 \) in
- \( b = 2 \) in
- \( \nu = 0.3 \)
- \( \alpha = 1 \times 10^{-6} \) in/(in °F)
- \( P_a = 100 \) psi
- \( T_a = 100 \) °F

COMPARISON OF RESULTS:
At \( r = 1.2875 \) in (elements 13, 15)

<table>
<thead>
<tr>
<th></th>
<th>( \sigma_r ), psi</th>
<th>( \sigma_\theta ), psi</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>-398.34</td>
<td>-592.47</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>-398.15</td>
<td>-596.46</td>
</tr>
</tbody>
</table>

Figure S19-1

Problem Sketch  Finite Element Model
**S20A, S20B: Cylindrical Shell Roof**

**TYPE:**
Static analysis, shell element (SHELL4, SHELL6).

**REFERENCE:**

**PROBLEM:**
Determine the vertical deflections across the midspan of a shell roof under its own weight. Dimensions and boundary conditions are shown in the figure below.

**GIVEN:**
- \( r = 25 \text{ ft} \)
- \( E = 3 \times 10^6 \text{ psi} \)
- \( \nu = 0 \)
- Shell Weight = 90 lbs/sq ft

**MODELING HINTS:**
Due to symmetry, a quarter of the shell is considered for modeling. The distributed force (self weight) is lumped at the nodes.

**COMPARISON OF RESULTS:**
Vertical Deflection at Midspan of free edge (Node 25):

<table>
<thead>
<tr>
<th>Theory</th>
<th>( \delta_x ) (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>-0.3024</td>
</tr>
<tr>
<td>COSMOS/M SHELL4</td>
<td>-0.3036</td>
</tr>
<tr>
<td>SHELL6 (Curved)</td>
<td>-0.24580</td>
</tr>
<tr>
<td>SHELL6 (Assembled)</td>
<td>-0.29353</td>
</tr>
</tbody>
</table>
Chapter 2  Linear Static Analysis

Figure S20-1

Problem Sketch and Finite Element Model

Figure S20-2

COSMOS/M Basic FEA System
Part 2 Verification Problems

S21A, S21B: Antisymmetric Cross-Ply Laminated Plate (SHELL4L)

**TYPE:**
Static analysis, composite shell element (SHELL4L, SHELL9L).

**REFERENCE:**

**PROBLEM:**
Calculate the maximum deflection of a simply supported antisymmetric cross-ply laminated plate under sinusoidal load. The plate is made up of 6-layers and the material in each layer is orthotropic.

**GIVEN:**
\[
\begin{align*}
a &= 100 \text{ in} \\
b &= 20 \text{ in} \\
h &= 1 \text{ in} \\
E_a &= 40E6 \text{ psi} \\
E_b &= 1E6 \text{ psi} \\
\nu_{ab} &= 0.25 \\
G_{ab} &= G_{ac} = G_{bc} = 5E5 \text{ psi} \\
\text{For each layer, pressure loading} &= \cos \pi x/a \cdot \cos \pi y/b
\end{align*}
\]

**MODELING HINT:**
Due to symmetry, a quarter of the plate is considered for modeling.
## COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>Maximum Deflection (in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>0.105E-2</td>
</tr>
<tr>
<td>COSMOS/M 4-nod shell</td>
<td>0.104E-2</td>
</tr>
<tr>
<td>COSMOS/M 9-node shell</td>
<td>0.111E-2</td>
</tr>
</tbody>
</table>

![Problem Sketch and Finite Element Model](image)

Figure S21-1
**S22: Thermally Loaded Support Structure**

**TYPE:**
Static, thermal stress analysis, truss and beam elements (TRUSS3D, BEAM3D).

**REFERENCE:**

**PROBLEM:**
Find the stresses in the copper and steel wire structure shown below. The structure is subjected to a load Q and a temperature rise of 10°F after assembly.

**GIVEN:**
Cross-sections area = 0.1 in²
Q = 4000 lb
\( \alpha_c = 92 \times 10 \text{ in/in - °F} \)
\( \alpha_s = 70 \times 10 \text{ in/in - °F} \)
\( E_c = 16 \times 10^6 \text{ psi} \)
\( E_s = 30 \times 10^6 \text{ psi} \)

**MODELING HINTS:**
Length and spacing between wires are arbitrarily selected. Truss element is used for elements number (1), (2), and (3), and the beam element for elements (4) and (5). Beam type and material are arbitrarily selected.

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>( \sigma_{\text{steel}} ) psi</th>
<th>( \sigma_{\text{copper}} ) psi</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>19695.0</td>
<td>10152.0</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>19704.2</td>
<td>10147.9</td>
</tr>
</tbody>
</table>

**Problem Sketch**

20°

**Figure S22-1**

Finite Element Model
S23: Thermal Stress Analysis of a Frame

**TYPE:**
Linear thermal stress analysis, beam elements (BEAM3D).

**REFERENCE:**

**PROBLEM:**
An irregular frame subjected to differential temperature. Find member end moments.

<table>
<thead>
<tr>
<th>Member</th>
<th>d (ft)</th>
<th>b (ft)</th>
<th>Ar-r (ft)</th>
<th>lt-t (ft)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.5</td>
<td>1.5</td>
<td>2.25</td>
<td>0.422</td>
</tr>
<tr>
<td>2</td>
<td>2.25</td>
<td>1.25</td>
<td>2.8125</td>
<td>1.187</td>
</tr>
<tr>
<td>3</td>
<td>2.0</td>
<td>1.5</td>
<td>3.0</td>
<td>1.0</td>
</tr>
<tr>
<td>4</td>
<td>2.5</td>
<td>1.25</td>
<td>3.125</td>
<td>1.628</td>
</tr>
<tr>
<td>5</td>
<td>2.0</td>
<td>1.5</td>
<td>3.0</td>
<td>1.0</td>
</tr>
</tbody>
</table>

**GIVEN:**

\[E = 192857 \text{ tons/ft}^2\]

\[\alpha = 0.00001 \text{ ft/ft}^2 \text{ C}\]

**COMPARISON OF RESULTS:**
Moments (lb-in):

<table>
<thead>
<tr>
<th>Member No.</th>
<th>COSMOS/M</th>
<th>Reference Solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-17.96</td>
<td>-17.96</td>
</tr>
<tr>
<td>2</td>
<td>+17.96</td>
<td>+17.96</td>
</tr>
<tr>
<td>3</td>
<td>+38.73</td>
<td>+38.64</td>
</tr>
<tr>
<td>4</td>
<td>+84.79</td>
<td>+84.92</td>
</tr>
<tr>
<td>5</td>
<td>-57.50</td>
<td>-57.40</td>
</tr>
</tbody>
</table>
Part 2  Verification Problems

Figure S23-1

Problem Sketch and Finite Element Model

Figure S23-2

Section A-A  Section B-B
Chapter 2  Linear Static Analysis

S24: Thermal Stress Analysis of a Simple Frame

TYPE:
Linear thermal stress analysis, beam elements (BEAM2D).

PROBLEM:
Determine displacements and end forces of the frame shown in the figure below due to temperature rise at the nodes and thermal gradients of members as specified below.

GIVEN:
\[ E = 30,000 \text{ kips/in}^2 \]
\[ \alpha = 0.65 \times 10 \text{ in/in/°F} \]

<table>
<thead>
<tr>
<th>Element No.</th>
<th>( \text{Difference in Temperature (°F)} )</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>( S )-dir</td>
</tr>
<tr>
<td>1</td>
<td>72</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
</tr>
</tbody>
</table>

COMPARISON OF RESULTS:
Displacements at node 2 (in):

<table>
<thead>
<tr>
<th></th>
<th>( \delta_x )</th>
<th>( \delta_y )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>-0.0583</td>
<td>0.1157</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>-0.0583</td>
<td>0.1168</td>
</tr>
</tbody>
</table>

Figure S24-1

Problem Sketch and Finite Element Model
S25: Torsion of a Square Box Beam

**TYPE:**
Static analysis, shell elements (SHELL4).

**REFERENCE:**

**PROBLEM:**
Find the shear stress and the angle of twist for the square box beam subjected to a torsional moment $T$.

**GIVEN:**
- $E = 7.5$ psi
- $\nu = 0.3$
- $t = 3$ in
- $a = 150$ in
- $L = 1500$ in
- $T = 300$ lb in

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Shear Stress $\tau$ psi</th>
<th>Rotation $\theta^*$, rad</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>0.00222</td>
<td>0.0154074</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>0.0021337 (average)</td>
<td>0.0154035*</td>
</tr>
</tbody>
</table>

* $\theta$ is calculated as:

$$\theta = \sin^{-1}(\text{resultant displacement of node 25/distance from node 25 to the center of the cross section})$$
**S26: Beam With Elastic Supports and a Hinge**

**TYPE:**
Static analysis, beam and truss elements (TRUSS3D, BEAM3D).

**REFERENCE:**

**PROBLEM:**
The final end actions of the members and the reactions of the supports resulting from the applied loading are to be determined for the structural system described in the figure below. At the beam-column connection, joint 3, the beam is continuous and the column is pin-connected to the beam.

**GIVEN:**
- Cross-sectional area of beams: $A_1 = A_2 = 0.125 \text{ ft}^2$
- Moment of inertia of beams: $I_1 = I_2 = 0.263 \text{ ft}^4$
- Cross-sectional area of column: $A_3 = 0.175 \text{ ft}^2$
- Moment of inertia of column: $I_3 = 0.193 \text{ ft}^4$
- Young’s modulus: $E = 1.44 \times 10^4 \text{ kip/ft}^2$
- Spring stiffness: $K = 1200 \text{ kips/ft}$

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Node 2</th>
<th>Node 4</th>
<th>Node 5</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Reference</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\delta_x (10^{-3} \text{ ft})$</td>
<td>1.0787</td>
<td>1.0787</td>
<td>1.0787</td>
</tr>
<tr>
<td>$\delta_y (10^{-3} \text{ ft})$</td>
<td>1.7873</td>
<td>-4.8205</td>
<td>-0.1803</td>
</tr>
<tr>
<td>$\theta_z (10^{-3} \text{ rad})$</td>
<td>0.0992</td>
<td>0.3615</td>
<td>-0.4443</td>
</tr>
<tr>
<td><strong>COSMOS/M</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\delta_x (10^{-3} \text{ ft})$</td>
<td>1.0794</td>
<td>1.0794</td>
<td>1.0794</td>
</tr>
<tr>
<td>$\delta_y (10^{-3} \text{ ft})$</td>
<td>1.7869</td>
<td>-4.8205</td>
<td>-0.1803</td>
</tr>
<tr>
<td>$\theta_z (10^{-3} \text{ rad})$</td>
<td>0.0992</td>
<td>0.3615</td>
<td>-0.4443</td>
</tr>
</tbody>
</table>
Figure S26-1

Problem Sketch and Finite Element Model

Figure S26-2
S27: Frame Analysis with Combined Loads

TYPE:
Static analysis, beam elements (BEAM3D).

REFERENCE:

PROBLEM:
Determine the forces in the beam members under the loads shown in the figure. Consider two separate load cases represented by the uniform pressure and the concentrated force. Set up the input to solve each one individually and then combine them together to obtain the final result.

GIVEN:
\[ I_{yy} = I_{zz} = 0.3215 \text{ ft}^4 \]
\[ I = 0.6430 \text{ ft}^4 \]
\[ A_1 = 3.50 \text{ ft}^2 \]
\[ A_{2,3} = 4.40 \text{ ft}^3 \]
\[ A_4 = 2.79 \text{ ft}^2 \]
\[ E = 432 \times 10^4 \text{ K/ft}^2 \]
Areas of members were made to be larger than the actual area in order to neglect axial deformation.

COMPARISON OF RESULTS:
The results are shown in the figure below with COSMOS/M results shown in parentheses.
Figure S27-1

Problem Sketch

Finite Element Model

Figure S27-2

2

6.766 K ft (6.76)

10.547 K ft (10.51)

28.256 K ft (28.32)

10.682 K ft (10.67)
S28: Cantilever Unsymmetric Beam

TYPE:
Static analysis, 3D beam element (BEAM3D).

REFERENCE:

PROBLEM:
An unsymmetric cantilever beam is subjected to a concentrated load at the free end. Determine the tip displacement of the beam, the end forces and the stress at y = 8, z = -2 at the clamped end.

GIVEN:

- \( E = 2 \times 10^7 \text{ N/cm}^2 \)
- \( F_y = -8 \text{ N} \)
- \( h_1 = 4 \text{ cm} \)
- \( b_1 = 2 \text{ cm} \)
- \( L = 500 \text{ cm} \)
- \( F_z = -4 \text{ N} \)
- \( h_2 = 8 \text{ cm} \)
- \( b_2 = 6 \text{ cm} \)
- \( A = 19 \text{ cm}^2 \)
- \( I_{yy} = 100.3 \text{ cm}^4 \)
- \( I_{zz} = 278.3 \text{ cm}^4 \)
- \( I_{yz} = 97.3 \text{ cm}^4 \)
- \( I_{xx} = J = 6.333 \text{ cm}^4 \)
- \( t = 1 \text{ cm} \)

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th>Node 6</th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>Translation in Y Dir (cm)</td>
<td>-0.1347</td>
<td>-0.1346</td>
</tr>
<tr>
<td>Translation in Z Dir (cm)</td>
<td>-0.2140</td>
<td>-0.21364</td>
</tr>
<tr>
<td>Rotation about X Axis (rad)</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Node 1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Moment about Y Axis (N-cm)</td>
<td>-2000.0</td>
<td>-2000.0</td>
</tr>
<tr>
<td>Shear in Z Dir (N)</td>
<td>4.0</td>
<td>4.0</td>
</tr>
<tr>
<td>Stress at Y = 8, Z = 2</td>
<td>155.74 Tension</td>
<td>155.8 Tension</td>
</tr>
</tbody>
</table>
Figure S28-1

Unsymmetric Beam Structure

Cross Section

Problem Sketch

Finite Element Model
S29A, S29B: Square Angle-Ply Composite Plate Under Sinusoidal Loading

**TYPE:**
Static analysis, composite shell (SHELL9L), and solid element (SOLIDL).

**REFERENCE:**

**PROBLEM:**
Calculate the maximum deflection of a simply supported square antisymmetric angle-ply under SINUSOIDAL loading. The plate is made up of 6 layers, where the top layer material axis orientation makes 45 degree angle with x-axis. To impose simply-supported boundary conditions, 2 layers of composite solid elements (each has 3 layers of different material orientation) through the thickness are required.

**GIVEN:**
- \(a = b = 20\) in
- \(E_a = 40E6\) psi
- \(h = 0.01\) in
- \(E_b = 1E6\) psi
- \(\nu = 0.25\)
- \(G_{ab} = G_{ac} = G_{bc} = 5E5\) psi
- \(p = \cos (\pi x/a) \cos (\pi y/b)\)
- \(p_0 = 1E-3\)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Max. Deflection</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference Solution</td>
<td>0.256</td>
</tr>
<tr>
<td>COSMOS/M SOLIDL</td>
<td>0.258</td>
</tr>
<tr>
<td>SHELL9L</td>
<td>0.258</td>
</tr>
</tbody>
</table>
S30: Effect of Transverse Shear on Maximum Deflection

**TYPE:**
Static analysis, shell elements (SHELL3).

**REFERENCE:**

**PROBLEM:**
Find the effect of transverse shear on maximum deflection of an isotropic simply supported plate subjected to a constant pressure, q.

**GIVEN:**

<table>
<thead>
<tr>
<th>Thickness Ratio H/a</th>
<th>Thickness H</th>
<th>(\beta) Coefficient*</th>
<th>Difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>H = varies according to thickness ratio (H/a)</td>
<td>Reissner Theory</td>
<td>COSMOS/M</td>
</tr>
<tr>
<td>0.000042</td>
<td>0.001008</td>
<td>0.04436</td>
<td>0.04438**</td>
</tr>
<tr>
<td>0.00042</td>
<td>0.01008</td>
<td>0.04436</td>
<td>0.04438**</td>
</tr>
<tr>
<td>0.0042</td>
<td>0.1008</td>
<td>0.04436</td>
<td>0.04438**</td>
</tr>
<tr>
<td>0.05</td>
<td>1.20</td>
<td>0.044936</td>
<td>0.044772 ***</td>
</tr>
<tr>
<td>0.1</td>
<td>2.40</td>
<td>0.046659</td>
<td>0.046510 ***</td>
</tr>
<tr>
<td>0.15</td>
<td>3.60</td>
<td>0.049533</td>
<td>0.049405 ***</td>
</tr>
<tr>
<td>0.2</td>
<td>4.80</td>
<td>0.053555</td>
<td>0.053458 ***</td>
</tr>
<tr>
<td>0.25</td>
<td>6.00</td>
<td>0.058727</td>
<td>0.058669 ***</td>
</tr>
<tr>
<td>0.3</td>
<td>7.20</td>
<td>0.065048</td>
<td>0.065038 ***</td>
</tr>
</tbody>
</table>

\[ \beta = \frac{EH^3w_{max}}{qa^4} \]

**MODELING HINTS:**
The input data corresponds to \(h = 0.1008\) and the other inputs can be obtained by changing the thickness in the given input data. Due to symmetry, only one quarter of the plate is considered.

**COMPARISON OF RESULTS:**

\(\beta\) = Thin Shell (SHELL3) **Thick Shell (SHELL3T)
Part 2  Verification Problems

Figure S30-1

Problem Sketch

Figure S30-2

Coefficient, $\beta \times 10^{-2}$

Present Finite Element
Reissner's Theory
Classical Theory

$W_{\text{max}}$ = $\frac{b}{qa^6}$
Chapter 2  Linear Static Analysis

S31: Square Angle-Ply Composite Plate Under Sinusoidal Loading

TYPE:
Static analysis, shell element (SHELL4L).

REFERENCE:

PROBLEM:
Calculate the maximum deflection of a simply supported square antisymmetric angle-ply under sinusoidal loading. The plate is made of 4-layers where the top layer material axis orientation makes 15 degree angle with the X-axis.

GIVEN:
\[ a = b = 20 \text{ in} \]
\[ h = 1 \text{ in} \]
\[ E_a = 40E6 \text{ psi} \]
\[ E_b = 1E6 \text{ psi} \]
\[ \nu = 0.25 \]
\[ G_{ab} = G_{ac} = G_{bc} = 5E5 \text{ psi} \]
\[ p = \cos \left( \frac{\pi x}{a} \right) \cos \left( \frac{\pi y}{b} \right) \]

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>( W_{max} ) (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>4.24E-4</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>4.40E-4</td>
</tr>
</tbody>
</table>
Part 2  Verification Problems


**TYPE:**
Static analysis, substructuring using truss elements (TRUSS2D).

**PROBLEM:**
Determine the deflections of a tower loaded at top, using multi-level substructures.

**GIVEN:**

\[
\begin{align*}
E &= 10 \times 10^6 \text{ psi} \\
P &= 1000 \text{ lb} \\
h &= 100 \text{ in} \\
L &= 30 \text{ in}
\end{align*}
\]

Cross-sectional areas of vertical and horizontal bars = 1 in²

Cross-sectional areas of diagonal bars = 0.707 in²

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Node Number</th>
<th>COSMOS/M Using Full Structure X</th>
<th>COSMOS/M Using Substructure X</th>
<th>COSMOS/M Using Full Structure Y</th>
<th>COSMOS/M Using Substructure Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>7.9761</td>
<td>9.1679</td>
<td>7.9761</td>
<td>9.1678</td>
</tr>
<tr>
<td>9</td>
<td>14.6226</td>
<td>25.3632</td>
<td>14.6226</td>
<td>25.3631</td>
</tr>
<tr>
<td>13</td>
<td>19.9360</td>
<td>46.8896</td>
<td>19.9359</td>
<td>46.8893</td>
</tr>
<tr>
<td>17</td>
<td>23.9168</td>
<td>71.9909</td>
<td>23.9167</td>
<td>71.9905</td>
</tr>
<tr>
<td>4</td>
<td>-2.1815</td>
<td>3.4329</td>
<td>-2.1815</td>
<td>3.4328</td>
</tr>
<tr>
<td>6</td>
<td>-4.0240</td>
<td>8.9940</td>
<td>-4.0239</td>
<td>8.9940</td>
</tr>
<tr>
<td>10</td>
<td>-6.7108</td>
<td>25.1829</td>
<td>-6.7108</td>
<td>25.1828</td>
</tr>
<tr>
<td>14</td>
<td>-8.0642</td>
<td>46.7075</td>
<td>-8.0641</td>
<td>46.7072</td>
</tr>
<tr>
<td>18</td>
<td>-8.0834</td>
<td>71.7683</td>
<td>-8.0833</td>
<td>71.7678</td>
</tr>
<tr>
<td>22</td>
<td>-6.7458</td>
<td>98.0790</td>
<td>-6.7457</td>
<td>98.0784</td>
</tr>
</tbody>
</table>
Figure S32A-1

Problem Sketch
(Full Structure)

Finite Element Models

(A) Super Element #1 - Level 3

(B) Super Element #2 - Level 2

(C) Super Element #3 - Level 1

(D) Super Element #4 - Level

(M) Main Element

Super Nodes

Problem Sketch
(Full Structure)
PART 2 VERIFICATION PROBLEMS

S33A, S33M: Substructure of an Airplane (Wing)

**TYPE:**
Static analysis, substructuring using shell, beam and truss elements (SHELL4, BEAM3D, TRUSS3D).

**PROBLEM:**
By using substructure method, determine the deflection of an airplane through the assembly of the calculations concerning separate parts, of the plane.

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Node Number</th>
<th>Deflection (Z-Direction) (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>COSMOS/M Using Full Structure</td>
</tr>
<tr>
<td>16</td>
<td>10.100</td>
</tr>
<tr>
<td>17</td>
<td>8.0671</td>
</tr>
<tr>
<td>18</td>
<td>13.800</td>
</tr>
<tr>
<td>19</td>
<td>10.666</td>
</tr>
<tr>
<td>20</td>
<td>8.7693</td>
</tr>
<tr>
<td>21</td>
<td>12.276</td>
</tr>
</tbody>
</table>

*Figure S33A-1*

Note: Nodes with ⭕ are Super Nodes

Problem Sketch (Full Structure)
Chapter 2  Linear Static Analysis

S34: Tie Rod with Lateral Loading

TYPE:
Static analysis, stress stiffening, beam elements (BEAM3D).

REFERENCE:

PROBLEM:
A tie rod subjected to the action of a tensile force $S$ and a uniform lateral load $q$. Determine the maximum deflection $z$, and the slope at the left end. In addition, determine the same two quantities for the unstiffened tie rod ($S = 0$).

GIVEN:
\[ L = 200 \text{ in} \]
\[ E = 30E6 \text{ psi} \]
\[ S = 21,972.6 \text{ lb} \]
\[ q = 1.79253 \text{ lb/in} \]
\[ b = h = 2.5 \text{ in} \]

CALculated INPUT:
\[ \text{Area} = 6.25 \text{ in}^2 \]
\[ l = 3.2552 \text{ in}^4 \]

MODELING HINTS:
Due to symmetry, only one-half of the beam is modeled.
COMPARISON OF RESULTS:

S ≠ 0 (Stiffened)

<table>
<thead>
<tr>
<th></th>
<th>$Z_{\text{max}}$ in</th>
<th>$\theta$ rad</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>-0.2</td>
<td>0.0032352</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>-0.19701</td>
<td>0.0031776</td>
</tr>
</tbody>
</table>

S = 0 (Unstiffened)

<table>
<thead>
<tr>
<th></th>
<th>$Z_{\text{max}}$ in</th>
<th>$\theta$ rad</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>-0.382406</td>
<td>0.006115</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>-0.37763</td>
<td>0.0060229</td>
</tr>
</tbody>
</table>

Figure S34-1

Problem Sketch

Finite Element Model
S35A, S35B: Spherical Cap Under Uniform Pressure (Solid)

**TYPE:**
Static analysis, solid and composite solid elements (SOLID, SOLIDL).

**REFERENCE:**

**PROBLEM:**
Calculate the center deflection of a simply supported spherical cap under uniform pressure \( (q = 1.0) \) in the direction normal to the cap surface. To impose simply-supported boundary conditions by solid elements, 2 layers of elements through the thickness are required.

Two types of material properties are being tested, each by a different solid element.

A. Isotropic material is handled by SOLID element (S35A).

B. Composite material, 4 layers with the orientation \( 0^\circ/90^\circ/90^\circ/0^\circ \), is analyzed by SOLIDL element (S35B). The lower layer of element is modeled by 2 layers of material orientation \( 0^\circ/90^\circ \) and the upper one is by \( 90^\circ/0^\circ \).

To capture the geometry of a curved surface by a bi-linear shape function accurately, at least 8 elements per side have to be used. The model used below is an 8x8x2 mesh.

**GIVEN:**

**Geometry:**
\[
R = 96 \\
h = 0.32 \text{ in} \\
\text{Length of side } a = b = c = d = 32 \text{ in}
\]

**Material Properties:**

1. S35A: Isotropic
   \[
   E = 1E7 \text{ psi} \\
   v = 0.3
   \]

2. S35B: Composite \( 0^\circ/90^\circ/90^\circ/0^\circ \)
   \[
   E_x = 25E6 \text{ psi} \\
   E_y = E_z = 1E6 \text{ psi} \\
   v_{xy} = 0.25 \\
   v_{yx} = v_{xy} = 0 \\
   G_{yz} = 0.2E6 \text{ psi} \\
   G_{xz} = G_{xz} = 0.5E6 \text{ psi}
   \]
MODELING HINTS:

*Boundary Conditions*

Due to symmetry:

1. All nodes on plane A, \( U_y = 0 \)
2. All nodes on plane B, \( U_x = 0 \)

Simply supported:

1. All nodes on side C, radial displacement \( = 0 \), Disp. on plane C = 0
2. All nodes on side D, radial displacement \( = 0 \), Disp. on plane D = 0
S36A, S36M: Substructure of a Simply Supported Plate

TYPE:
Static analysis, substructuring using plate elements (SHELL4).

REFERENCE:

PROBLEM:
Calculate the deflections of a simply supported isotropic plate subjected to uniform pressure \( p \) using the substructuring technique. Nodes 11 through 15 are super nodes which connect the substructure to the main structure.

GIVEN:
\[
\begin{align*}
E &= 30 \times 10 \text{ psi} \\
\nu &= 0.3 \\
h &= 0.5 \text{ in} \\
p &= 5 \text{ psi} \\
a &= 16 \text{ in} \\
b &= 10 \text{ in}
\end{align*}
\]

MODELING HINTS:
Due to symmetry, a quarter of a plate is taken for modeling.

COMPARISON OF RESULTS:
Timoshenko gives the expression for deflection \( w \) in the z-direction with origin at the corner of the plate.
### Part 2  Verification Problems

#### Figure S36A-1

<table>
<thead>
<tr>
<th>Node No.</th>
<th>X (inch)</th>
<th>Y (inch)</th>
<th>W (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X</td>
<td>Y</td>
<td>Theory</td>
</tr>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0.0012103</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>0</td>
<td>0.0011338</td>
</tr>
<tr>
<td>3</td>
<td>4</td>
<td>0</td>
<td>0.0009043</td>
</tr>
<tr>
<td>4</td>
<td>6</td>
<td>0</td>
<td>0.0005150</td>
</tr>
</tbody>
</table>

**Problem Sketch**

(A) Main Structure  (B) Substructure

**Finite Element Models**
**S37: Hyperboloidal Shell Under Uniform Ring Load Around Free Edge**

**TYPE:**
Static analysis, axisymmetric shell elements (SHELLAX).

**REFERENCE:**

**PROBLEM:**
Determine the horizontal displacement of a hyperboloidal shell under uniform ring load around free edge.

**GIVEN:**
\[
\begin{align*}
R_0 &= 600 \text{ in} \\
R_1 &= 1200 \text{ in} \\
H &= 2400 \text{ in} \\
H_0 &= 1500 \text{ in} \\
t &= 8 \text{ in} \\
E &= 3000 \text{ kip/sq in} \\
\nu &= 0.3 \\
P &= 1 \text{ kip/in} \\
\end{align*}
\]

Equation of the hyperboloid:
\[X^2 = 0.48(Y - H_0)^2 + R^2\]

**MODELING HINTS:**
Nodes at the top of the tower are spaced closely because of the concentrated ring load. Nodal spacing is as follows:

<table>
<thead>
<tr>
<th>Nodes</th>
<th>1-11</th>
<th>11-21</th>
<th>21-29</th>
<th>29-39</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dy (in)</td>
<td>10</td>
<td>20</td>
<td>75</td>
<td>150</td>
</tr>
</tbody>
</table>

And it is to be noted that the ring load should be input per radian length, since the radius at the top of the shell is 865.3323 in, the load is 865.33 kip/rad.
Part 2  Verification Problems

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>Maximum Displacement at Node 41 (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>-0.904</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>-0.89705</td>
</tr>
</tbody>
</table>

Figure S37-1
S38: Rotating Solid Disk

**TYPE:**
Static analysis, axisymmetric (PLANE2D) elements, centrifugal loading.

**REFERENCE:**

**PROBLEM:**
A solid disk rotates about center 0 with angular velocity $\omega$. Determine the stress distribution in the disk.

**GIVEN:**
- $E = 30 \times 10^6$ psi
- $\nu = 0.3$
- $DENS = 0.02$ lb sec$^2$/in$^4$
- $h = 1$ in
- $\omega = 25$ rad/sec
- $R = 9$ in

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Location</th>
<th>$\sigma_r$ psi</th>
<th>$\sigma_\theta$ psi</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>416.37</td>
<td>416.91</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>416.82</td>
<td>416.82</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Location</th>
<th>$\sigma_r$ psi</th>
<th>$\sigma_\theta$ psi</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element 1 (r = 0.5 inch)</td>
<td>416.37</td>
<td>416.91</td>
</tr>
<tr>
<td>Element 9 (r = 8.5 inch)</td>
<td>45.12</td>
<td>203.16</td>
</tr>
</tbody>
</table>

Figure S38-1

Problem Sketch and Finite Element Model
S39: Unbalanced Rotating Flywheel

**TYPE:**
Static analysis, centrifugal loading, beam and mass elements (BEAM3D, MASS).

**PROBLEM:**
The model shown in the figure is assumed to be rotating about the y-axis at a constant angular velocity of 25 rad/sec. Determine the axial forces and bending moments in the supporting beams and columns due to self-weight and rotational inertia.

**GIVEN:**

\[
\begin{align*}
A &= 100 \text{ in}^2 \\
I_{yy} &= I_{zz} = 1000 \text{ in}^4 \\
I_{xx} &= 2000 \text{ in}^4 \\
E &= 30 \times 10^6 \text{ psi} \\
G &= 10 \times 10^6 \text{ psi} \\
\rho &= 0.01 \text{ lb-sec}^2/\text{in}^4 \\
\omega &= 25 \text{ rad/sec} \\
m_3 &= m_4 = 10 \\
a_y &= -100 \text{ in/sec}^2 \\
w &= 25 \text{ rad/sec}
\end{align*}
\]

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element 2, Node 2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Axial Force</td>
<td>58,800</td>
<td>58,800</td>
</tr>
<tr>
<td>Bending Moment</td>
<td>412,000</td>
<td>412,000</td>
</tr>
<tr>
<td>Element 3, Node 2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Axial Force</td>
<td>61,200</td>
<td>61,200</td>
</tr>
<tr>
<td>Bending Moment</td>
<td>388,000</td>
<td>388,000</td>
</tr>
</tbody>
</table>
**S40: Truss Structure Subject to a Concentrated Load**

**TYPE:**
Static analysis, truss elements (TRUSS2D).

**REFERENCE:**

**PROBLEM:**
Calculate the reactions and the vertical deflection of joint 2 of the loaded truss shown below subject to a concentrated load.

**GIVEN:**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( E )</td>
<td>30,000 kips/in(^2)</td>
</tr>
<tr>
<td>( P )</td>
<td>64 kips</td>
</tr>
<tr>
<td>( L / A )</td>
<td>1 for all members</td>
</tr>
</tbody>
</table>

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection of Joint 2</td>
<td>0.006733 in</td>
<td>0.006733 in</td>
</tr>
<tr>
<td>Reaction at Node 1</td>
<td>48 K</td>
<td>48 K</td>
</tr>
<tr>
<td>Reaction at Node 5</td>
<td>16 K</td>
<td>16 K</td>
</tr>
</tbody>
</table>

**Problem Sketch and Finite Element Model**
S41: Reactions of a Frame Structure

**TYPE:**
Static analysis, beam element (BEAM2D).

**REFERENCE:**

**PROBLEM:**
Determine the reactions for the frame shown below.

**GIVEN:**
- \( E = 30 \times 10^6 \text{ psi} \)
- \( A = 0.1 \text{ in}^2 \)
- The relative values of \( 2EI/L \):
  - for element 1 = 1 lb-in
  - for elements 2, 3 = 2 lb-in

**COMPARISON OF RESULTS:**
The free body diagram for the structural system is given below and COSMOS/M results are given in parentheses.
S42A, S42B: Reactions and Deflections of a Cantilever Beam

**TYPE:**
Static analysis, shell elements (SHELL4, SHELL6).

**PROBLEM:**
Calculate reactions and deflections of a cantilever beam subject to a concentrated load at tip.

**GIVEN:**
- $E = 30 \text{E}6 \text{ psi}$
- $h = 1 \text{ in}$
- $L = 10 \text{ in}$
- $W = 4 \text{ in}$
- $P = 8 \text{ lb}$

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>SHELL4</td>
</tr>
<tr>
<td>Tip Deflection</td>
<td></td>
<td>$-2.667 \times 10^{-4}$</td>
</tr>
<tr>
<td>Total Force Reaction</td>
<td>8 lb</td>
<td>8 lb</td>
</tr>
<tr>
<td>Total Moment Reaction</td>
<td>-80 lb-in</td>
<td>-80 lb-in</td>
</tr>
</tbody>
</table>

Figure S42-1
S43: Bending of a T Section Beam

**TYPE:**
Static analysis, shell element, beam element with offset (SHELL4L, BEAM3D).

**PROBLEM:**
Calculate the deflections and stresses of a cantilever T beam subjected to a concentrated load at the free end.

**GIVEN:**
- \( L = 2000 \text{ in} \)
- \( y = 49 \text{ in} \)
- \( I = 480833.33 \text{ in}^4 \)
- \( E = 10\times10^8 \text{ psi} \)
- \( D_y = -24 \text{ in} \)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>Free End (at Node 12)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Y-Displacement (in)</td>
<td>-5.546E-6</td>
<td>-5.588E-6</td>
</tr>
<tr>
<td>( \theta_z )- Rotation</td>
<td>-4.159E-9</td>
<td>-4.161E-9</td>
</tr>
<tr>
<td>Clamped End</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( \sigma ) top (psi)</td>
<td>4.57360</td>
<td>4.377</td>
</tr>
<tr>
<td>( \sigma ) bottom (psi)</td>
<td>20.3813</td>
<td>21.302</td>
</tr>
</tbody>
</table>

**ANALYTICAL SOLUTION:**
- \( \delta = \frac{PL^3}{3EI} \)
- \( \phi = \frac{PL^2}{2EI} \)
- \( \sigma = \frac{Mc}{I} \)

**NOTE:**
The maximum stress occurs in the beam. The point at which stresses are calculated for unsymmetric beams should be specified in the real constant set (real constants 25 and 26).
Chapter 2  Linear Static Analysis

S44A, S44B: Bending of a Circular Plate with a Center Hole

TYPE:
Static analysis, shell elements (SHELL4), coupled points (file S44A) and/or constraint equations (file S44B).

REFERENCE:

PROBLEM:
A circular plate with a center hole is built-in along the inner edge and unsupported along the outer edge. The plate is subjected to bending by a moment M applied along the outer edge. Determine the maximum deflection and the maximum slope of the plate. In addition, determine the moment M and the corresponding stress at the center of the first and the last elements.

GIVEN:
\[
\begin{align*}
E &= 30E6 \text{ psi} \\
\nu &= 0.3 \\
h &= 0.25 \text{ in} \\
a &= 30 \text{ in} \\
b &= 10 \text{ in} \\
M &= 10 \text{ in lb/in} \\
\theta &= 10^\circ
\end{align*}
\]

CALCULATED INPUT:
\[
M_{1a} = 10 \text{ in-lb/in} = 52.359 \text{ in lb/10^\circ segment}
\]

MODELING HINTS:
Since the problem is axisymmetric, only a small sector of elements is needed. A small angle \( \theta \) is used for approximating the circular boundary with a straight-side element. A radial grid with nonuniform spacing (3:1) is used. The load is applied equally to the outer nodes. Coupled nodes (CPDOF) and/or constraint equations (CEQN) are used to ensure symmetry for S44A and S44B, respectively. Note that all constraint and load commands are active in the cylindrical coordinate system.
**COMPARISON OF RESULTS***:

At the outer edge (node 14).

<table>
<thead>
<tr>
<th></th>
<th>$\delta_x$, inch</th>
<th>$\theta_y$, rad</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>0.0490577</td>
<td>-0.0045089</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>0.0492188</td>
<td>-0.0044562</td>
</tr>
<tr>
<td>Difference</td>
<td>0.3%</td>
<td>1.17%</td>
</tr>
</tbody>
</table>

* The above results are tabulated for S44A. Identical results will be obtained for S44B.

<table>
<thead>
<tr>
<th>X = 10.86 inch (First Element)</th>
<th>X = 27.2 inch (Sixth Element)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment in-lb/in</td>
<td>$\sigma_r$, psi</td>
</tr>
<tr>
<td>Theory</td>
<td>-13.7</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>-13.7</td>
</tr>
<tr>
<td>Difference</td>
<td>0%</td>
</tr>
</tbody>
</table>

* The above results are tabulated for S44A. Identical results will be obtained for S44B.

Figure S44-1
TYPE: Static analysis, beam elements (BEAM3D) and point-to-point constraints (CPCNS command).


PROBLEM: Two vertical beams constitute an eccentric portal frame with the aid of 3 horizontal rigid bars. Find the deformations resulting from the horizontal forces.

GIVEN: \( h = 1 \text{ in} \quad S = 2.5 \)  
\( W = 1 \text{ in} \quad E = 1E6 \text{ psi} \)  
\( L = 10 \text{ in} \quad P = 1 \text{ lb} \)  
\( I = 1/12 \text{ in}^4 \) (about y and z axis)

MODELING HINT: Point-to-point constraint elements are used (i.e., points 2-3, 3-4 and 4-5) to ensure the frame 2-3 - 4-5 is rigid when the horizontal forces are loaded. Each of the beams (1) and (2) will deform as shown:

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th>Deflection along X-axis at points A and B:</th>
<th>( \delta_x ) (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>1.0000E</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>1.0104E</td>
</tr>
<tr>
<td>Difference</td>
<td>1%</td>
</tr>
</tbody>
</table>
Part 2  Verification Problems

S46, S46A, S46B: Bending of a Cantilever Beam

TYPE:
Static analysis, plane stress elements (PLANE2D), beam elements (BEAM2D),
shell elements (SHELL9) and constraint elements.

PROBLEM:
Calculate the maximum deflection and the maximum rotation of a cantilever beam
loaded by a shear force at the free end.

GIVEN:

\[
\begin{align*}
  h &= 1 \text{ in} \\
  L &= 10 \text{ in} \\
  I &= 1/12 \text{ in}^4 \\
  E &= 1\text{E}6 \text{ psi} \\
  \nu &= 0.3 \\
  p &= -1 \text{ lb}
\end{align*}
\]

MODELING HINTS: Continuum-to-Structure Constraint

Problem 1 (S46):
The plane stress elements are defined by nodes 1 through 12. The beam element is
defined by nodes 13 and 14. Each plane stress element is theoretically equivalent to
a beam element where \( I = 1/12 \text{ in}^4 \). Node 14 is attached to line 11-12, so displace-
ments and rotations are constrained to be compatible.

Problem 2 (S46A):
Two groups of PLANE2D, plane stress, 8-node elements are coupled together as
shown in Figure S46–2 where the geometry and material properties are the same as
those in Problem 1. The focus of interest is on the continuum-to-continuum
constraint and the location of the primary point which is no longer located at the
middle of the 3-point curve, but at any arbitrary position.

Problem 3 (S46B):
Two groups of SHELL9 elements are coupled together as shown in Figure S46–3
where the geometry and material properties are the same as those in Problem 1 and
2. The primary deformation is located in the x-y plane. This problem is provided to
verify the accuracy of the structure-to-structure constraint.

ANALYTICAL SOLUTION:

\[
\begin{align*}
  \delta_y &= -pL^3 / 3EI \\
  \theta_x &= -pL^2 / 2EI
\end{align*}
\]
Chapter 2  Linear Static Analysis

COMPARISON OF RESULTS:

At the free end:

<table>
<thead>
<tr>
<th></th>
<th>$\delta_y$ inch</th>
<th>$\theta_z$ rad</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>-4.000E-3</td>
<td>-6.000E-4</td>
</tr>
<tr>
<td>Beam Element</td>
<td>-4.000E-3</td>
<td>-6.000E-4</td>
</tr>
<tr>
<td>Plane Stress Element</td>
<td>-4.006E-3</td>
<td>-6.000E-4</td>
</tr>
<tr>
<td>Beam/Plane Stress Element (S46)</td>
<td>-4.008E-3</td>
<td>-6.000E-4</td>
</tr>
<tr>
<td>Plane Stress/Plane Stress Element (S46A)</td>
<td>-4.009E-3</td>
<td>-5.985E-4 *</td>
</tr>
<tr>
<td>SHELL9/SHELL9 Element (S46B)</td>
<td>-4.014E-3</td>
<td>-5.990E-4 *</td>
</tr>
</tbody>
</table>

* Computed using displacements at the free end.

**Figure S46-1**

![Problem Sketch](image)

**Finite Element Model**

**Figure S46-2**

![Finite Element Model - 2](image)

**Figure S46-3**

![Finite Element Model - 3](image)
Part 2 Verification Problems

S47, S47A, S47B: Bending of a Cantilever Beam

TYPE:
Static analysis, SOLID elements, TETRA10 elements, BEAM elements, and point-to-surface constraint elements (attachment).

PROBLEM:
Calculate the maximum deflection and the maximum rotation $\theta$ of a cantilever beam loaded by a shear force at the free end.

GIVEN:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$h$</td>
<td>1 in</td>
</tr>
<tr>
<td>$w$</td>
<td>1 in</td>
</tr>
<tr>
<td>$L$</td>
<td>10 in</td>
</tr>
<tr>
<td>$I$</td>
<td>$1/12$ in$^4$</td>
</tr>
<tr>
<td>$E$</td>
<td>1E6 psi</td>
</tr>
<tr>
<td>$\nu$</td>
<td>0</td>
</tr>
<tr>
<td>$p$</td>
<td>1 lb</td>
</tr>
</tbody>
</table>

MODELING HINTS: Continuum-to-Structure Constraint

Problem 1 (S47):
The solid elements are defined by nodes 1 through 24. The beam element is defined by nodes 25 and 26. Each solid element is theoretically equivalent to a beam element where $I = 1/12$ in$^4$ about y and z axes. Node 25 is attached to surface 21-22-24-23, so displacements and rotations are constrained to be compatible.

Problem 2 (S47A):
Two groups of SOLID 20-node elements are coupled together as shown in Figure S47-2 where the geometry and material properties are the same as those in Problem 1. The focus of interest is on the continuum-to-continuum constraint and the location of the primary point which is no longer located at the middle of the 8-point surface, but at an arbitrary position.

Problem 3 (S47B):
Two groups of TETRA10 elements are coupled together as shown in Figure S47-3 where the geometry and material properties are the same as those in Problem 1 and 2. This problem is provided to verify the accuracy of the continuum-to-continuum constraint with the primary point located at any arbitrary position of a 6-node surface.

ANALYTICAL SOLUTION:

\[
\delta = -\frac{PL^3}{3EI} \\
\theta = -\frac{PL^2}{2EI}
\]
Chapter 2  Linear Static Analysis

COMPARISON OF RESULTS:
At the free end.

<table>
<thead>
<tr>
<th></th>
<th>Deflection δ inch</th>
<th>Rotation θ rad</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>-4.000E-3</td>
<td>-6.000E-4</td>
</tr>
<tr>
<td>Beam Element</td>
<td>-4.000E-3</td>
<td>-6.000E-4</td>
</tr>
<tr>
<td>Plane Stress Element</td>
<td>-4.010E-3</td>
<td>-6.000E-4</td>
</tr>
<tr>
<td>Beam/Solid Element (S47)</td>
<td>-4.005E-3</td>
<td>-6.000E-4</td>
</tr>
<tr>
<td>Solid/Solid Element (S47A)</td>
<td>-3.986E-3</td>
<td>-5.962E-4 *</td>
</tr>
<tr>
<td>Tetra10/Tetra10 Element (S47B)</td>
<td>-3.969E-3</td>
<td>-5.950E-4 *</td>
</tr>
</tbody>
</table>

* Computed using displacements at the free end.

Figure S47-1

Finite Element Model – 1

Problem Sketch

Figure S47-2

Finite Element Model - 2

Point-to-surface Constraints

SOLID Elements

Figure S47-3

Finite Element Model - 3

Point-to-surface Constraints

TETRA10 Elements
S48: Rotation of a Tank of Fluid (PLANE2D Fluid)

TYPE:
Static analysis, axisymmetric elements (PLANE2D).

REFERENCE:

PROBLEM:
A large cylindrical tank is partially filled with an incompressible liquid. The tank rotates at a constant angular velocity about its vertical axis as shown. Determine the elevation of the liquid surface relative to the center (lowest) elevation for various radial positions. Also, determine the pressure $p$ in the fluid near the bottom corner of the tank.

GIVEN:
\[
\begin{align*}
  w &= 1 \text{ rad/sec} \\
  r &= 48 \text{ in} \\
  h &= 20 \text{ in} \\
  \rho &= 0.9345 \times 10^{-4} \text{ lb-sec}^2/\text{in}^4 \\
  g &= 386.4 \text{ in/sec}^2 \\
  b &= 30 \times 10^4 \text{ psi}
\end{align*}
\]

Where:
\[
\begin{align*}
  b &= \text{bulk modulus} \\
  g &= \text{acceleration due to gravity} \\
  \rho &= \text{density}
\end{align*}
\]

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th>Displacements * $\delta_y$ inch</th>
<th>Pressure (psi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node 4</td>
<td>Node 7</td>
</tr>
<tr>
<td>Theory</td>
<td>-1.86335</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>-1.8627</td>
</tr>
<tr>
<td>Difference</td>
<td>0.036%</td>
</tr>
</tbody>
</table>

* After subtracting from the displacement at Node 1 (-1.4798 in)
Figure S48-1

Problem Sketch

Finite Element Model
**S49A, S49B: Acceleration of a Tank of Fluid (PLANE2D Fluid)**

**TYPE:**
Static analysis plane strain (PLANE2D) or SOLID elements.

**REFERENCE:**

**PROBLEM:**
Large rectangular tank is partially filled with an incompressible liquid. The tank has a constant acceleration to the right, as shown. Determine the elevation of the liquid surface relative to the zero acceleration elevation for various Y-axis positions. Also, determine the slope of the surface and the pressure \( p \) in the fluid near the bottom right corner of the tank.

**GIVEN:**

\[
\begin{align*}
    a &= 45 \text{ in/sec}^2 \\
    b &= 48 \text{ in} \\
    h &= 20 \text{ in} \\
    g &= 386.4 \text{ in/sec}^2 \\
    p &= 30\text{E}4 \text{ psi} \\
    \rho &= 0.9345\text{E}-4 \text{ lb-sec}^2/\text{in}^4
\end{align*}
\]

Where:
- \( b \) = bulk modulus
- \( g \) = acceleration due to gravity
- \( \rho \) = density

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Displacements</th>
<th>( \delta_y \text{ inch} )</th>
<th>Pressure (psi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node 3</td>
<td>Node 7</td>
<td>Node 11</td>
</tr>
<tr>
<td>Theory</td>
<td>-1.86335</td>
<td>0</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>-1.8627</td>
<td>0</td>
</tr>
<tr>
<td>Difference</td>
<td>0.036%</td>
<td>0%</td>
</tr>
</tbody>
</table>
Figure S49A-1

Problem Sketch

Finite Element Model
(Use PLANE2D Elements)
S50A, S50B, S50C, S50D, S50F, S50G, S50H, S50I: Deflection of a Curved Beam

TYPE:
Static analysis, multi-field elements (4-node PLANE2D, 8-node PLANE2D, SHELL4T, 6-node TRIANG, 8-node SOLID, 20-node SOLID, TETRA4R and SHELL6 elements).

REFERENCE:

PROBLEM:
A curved beam is clamped at one end and subjected to a shear force \( P \) at the other end. Determine the deflection at the free end.

GIVEN:
\[
E = 10 \times 10^6 \text{ psi} \\
\nu = 0.25 \\
R_1 = 4.12 \text{ in}
\]

COMPARISON OF RESULTS:
Deflections at free end by theoretical solution is equal to 0.08854 in

<table>
<thead>
<tr>
<th>Element</th>
<th>COSMOS/M ( \delta_y ) in</th>
<th>Difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>PLANE2D (4-Node) (S50A)</td>
<td>0.08761</td>
<td>1.05%</td>
</tr>
<tr>
<td>PLANE2D (8-Node) (S50B)</td>
<td>0.08850</td>
<td>0%</td>
</tr>
<tr>
<td>SHELL4T (S50C)</td>
<td>0.08827</td>
<td>0.26%</td>
</tr>
<tr>
<td>TRIANG (6-Node) (S50D)</td>
<td>0.07049</td>
<td>11.6%</td>
</tr>
<tr>
<td>TETRA4R (4-Node) (S50H)</td>
<td>0.08785</td>
<td>0.8%</td>
</tr>
<tr>
<td>SOLID (8-Node) (S50F)</td>
<td>0.08726</td>
<td>1.45%</td>
</tr>
<tr>
<td>SOLID (20-Node) (S50G)</td>
<td>0.08848</td>
<td>0.07%</td>
</tr>
<tr>
<td>SHELL6 (Curved) (S50I)</td>
<td>0.07498</td>
<td>15.32%</td>
</tr>
<tr>
<td>SHELL6 (Assembled) (S50I)</td>
<td>0.062679</td>
<td>29.2%</td>
</tr>
</tbody>
</table>
Chapter 2  Linear Static Analysis

S51: Gable Frame with Hinged Supports

**TYPE:**
Static analysis, beam elements (BEAM2D).

**REFERENCE:**

**PROBLEM:**
Determine the support reactions for frame shown in the figure.

**GIVEN:**
- \( L = 16 \) ft
- \( h = 8 \) ft
- \( E = 4.32 \times 10^6 \) lb/ft²
- \( f = 6 \) ft
- \( q = 10 \) lb/ft
- \( I_{12} = I_{23} = I_{34} = I_{45} \)
- \( A_{12} = A_{23} = A_{34} = A_{34} = A_{45} \)
- Total Load = 4 lbs

**MODELING HINTS:**
Find load intensity along the frame from
\[
W = \frac{\text{Total load}}{q} = 4 \text{ lb/ft}
\]
Then use beam loading commands to solve the problem

**COMPARISON OF RESULTS:**
Reactions (lb):

<table>
<thead>
<tr>
<th>Node No.</th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FX</td>
<td>FY</td>
</tr>
<tr>
<td>1</td>
<td>4.44</td>
<td>30.00</td>
</tr>
<tr>
<td>5</td>
<td>-4.44</td>
<td>10.00</td>
</tr>
</tbody>
</table>

Figure S51-1

![Problem Sketch](Figure S51-1.png)

Finite Element Model
**S52: Support Reactions for a Beam with Intermediate Forces and Moments**

**TYPE:**
Static analysis, beam elements (BEAM2D).

**REFERENCE:**

**PROBLEM:**
Determine the support reactions for the simply supported beam with intermediate forces and moments.

**Figure S52-1**

![Problem Sketch](image)

![Finite Element Sketch](image)
Chapter 2  Linear Static Analysis

**GIVEN:**

\[ A = 0.3472 \text{ ft}^2 \]
\[ I_y = I_z = 0.02009 \text{ ft}^4 \]
\[ I_x = 0.04019 \text{ ft}^4 \]
\[ E = 4320 \times 10^3 \text{ K/ft}^2 \]

**NOTE:**

The sign convention for the intermediate loads follows the local coordinate system for the beam (defined by the I, J, K nodes).

**COMPARISON OF RESULTS:**

*Figure S52-2*

\[ R_{x^p} = -45K (-45K) \]
\[ R_y = 57K (57K) \]
\[ R_{2y^p} = 89K (89K) \]

**NOTE:**

The results obtained with COSMOS/M are compared with those given in reference. The numbers shown in parenthesis are from COSMOS/M.
S53: Beam Analysis with Intermediate Loads

**TYPE:**
Static analysis, beam elements (BEAM2D).

**REFERENCE:**

**PROBLEM:**
Find the reactions in the support and forces and moments in the beam.

**Figure S53-1**

**GIVEN:**
\[ I_{yy} = I_{zz} = 1 \text{ ft}^4 \]
\[ I_{xx} = 2 \text{ ft}^4 \]
\[ A = 3.464 \text{ ft}^2 \]
\[ E = 432 \times 10^4 \text{ k/ft}^2 \]
NOTE:
The sign convention for intermediate loads, follows the local coordinate system, for the beam (defined by I, J, K nodes).

COMPARISON OF RESULTS:

Figure S53-2

Use the BEAMRESLIST (Results, List, Beam End Force) command to list the results

NOTE:
COSMOS/M results are given in parentheses.
**S54: Analysis of a Plane Frame with Beam Loads**

**TYPE:**
Static analysis, beam elements (BEAM2D).

**REFERENCE:**

**PROBLEM:**
Find the deformations and forces in the plane frame subjected to intermediate forces and moments.

**Figure S54-1**

**GIVEN:**
- \( A_x = 0.04 \text{ m}^2 \)
- \( I_y = I_z = 2 \times 10^3 \text{ m}^4 \)
- \( I_x = 4 \times 10^3 \text{ m}^4 \)
- \( E = 200 \times 10^6 \text{ KN/m}^2 \)
- \( L = 3 \text{ m} \)
- \( P_1 = 30 \times 2^{(1/2)} \text{ KN} \)
- \( P_2 = 60 \text{ KN} \)
- \( M = 180 \text{ KN-m} \)

**Figure S54-2**

\* Results obtained from reference
\+ Results obtained from COSMOS/M

Moments are in KNm units.
S55: Laterally Loaded Tapered Beam

TYPE:
Static analysis, beam element (BEAM3D).

REFERENCE:

PROBLEM:
A cantilever beam of width \( b \) and length \( L \) has a depth which tapers uniformly from \( d \) at the tip to \( 3d \) at the wall. It is loaded by a force \( P \) at the tip. Find the maximum bending stress at \( x = L \) (midspan).

GIVEN:
- \( P = 4000 \text{ lb} \)
- \( L = 50 \text{ in} \)
- \( d = 3 \text{ in} \)
- \( b = 2 \text{ in} \)
- \( E = 30E6 \text{ psi} \)

COMPARISON OF RESULTS:
<table>
<thead>
<tr>
<th>( \sigma_x ) (psi) (at node 2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
</tr>
<tr>
<td>COSMOS/M</td>
</tr>
</tbody>
</table>

Figure S55-1
S56: Circular Plate Under a Concentrated Load (SHELL9 Element)

**Type:**
Static analysis, 9-node shell element (SHELL9).

**Reference:**

**Problem:**
A circular thick plate clamped at the boundary is subjected to a point load at its center. (Shown in Figure S56-1).

Determine the transverse displacement along the radius r.

**Given:**
- \( E = 1.09 \times 10^6 \)
- \( v = 0.3 \)
- \( t = 2 \) (thickness) in
- \( P = 4 \) lb
- \( R = 5 \) in

**Analytical Solution:**

\[
W_r = \frac{PR^2}{16\pi D} \left[ 1 - \left( \frac{r}{R} \right)^2 \right] - \frac{2r^2}{R^2} \ln \frac{R}{r} - \frac{8D}{KGr^2} \ln \frac{R}{r}
\]

Where:
- \( D = \frac{Et^3}{12(1-v^2)} \)
- \( G = \frac{E}{2(1+v)} \)
- \( K = 0.8333 \) (shear correction factor)
**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Node</th>
<th>r (in)</th>
<th>$W_{\text{max}}$ (in) $\times 10^{-6}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Analytical</td>
</tr>
<tr>
<td>1</td>
<td>0.0</td>
<td>—</td>
</tr>
<tr>
<td>2</td>
<td>.625</td>
<td>4.185</td>
</tr>
<tr>
<td>3</td>
<td>1.250</td>
<td>3.1670</td>
</tr>
<tr>
<td>4</td>
<td>1.875</td>
<td>2.3474</td>
</tr>
<tr>
<td>5</td>
<td>2.500</td>
<td>1.6366</td>
</tr>
<tr>
<td>6</td>
<td>3.125</td>
<td>1.0316</td>
</tr>
<tr>
<td>7</td>
<td>3.750</td>
<td>0.5458</td>
</tr>
<tr>
<td>8</td>
<td>4.375</td>
<td>0.1962</td>
</tr>
</tbody>
</table>

Figure S56-1

Problem Sketch

Finite Element Model
(12 Elements)
S57: Test of a Pinched Cylinder with Diaphragm (SHELL9 Element)

**TYPE:**
Static analysis, 9-node shell element (SHELL9).

**REFERENCE:**

**PROBLEM:**
A cylindrical shell with both ends covered with rigid diaphragms which allow displacement only in the axial direction of the cylinder is subjected to a concentrated load on the center (shown in the figure below). Determine the radial deflection of point P.

**GIVEN:**
- \( R = 300 \text{ in} \)
- \( L = 600 \text{ in} \)
- \( E = 3E6 \text{ psi} \)
- \( \nu = 0.3 \)
- \( h = 3 \text{ (thickness) in} \)
- \( P = 1 \text{ lb} \)

**COMPARISON OF RESULTS**

<table>
<thead>
<tr>
<th></th>
<th>( \delta_x ) (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>0.18248E-4</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>0.17651E-4</td>
</tr>
</tbody>
</table>

**MODELING HINTS:**
Due to symmetry, only one-eighth of the cylinder is modeled. To simulate the rigid diaphragm, on the boundary of the cylinder with \( z = 0 \), no rotation along the axial direction (z-axis) is allowed.
Chapter 2  Linear Static Analysis

Figure S57-1

Problem Sketch

Finite Element Model

(4 x 4)
S58A, S58B, S58C: Deflection of a Twisted Beam with Tip Force

**TYPE:**
Static analysis, 9-node shell element (SHELL9), 4-node tetrahedral element (TETRA4R).

**REFERENCE:**

**PROBLEM:**
A twisted beam is subjected to a concentrated load at the tip in the in-plane and out-of-plane directions (shown in the figure below). Determine the deflections coincident with the load.

**GIVEN:**
- \( L = 12 \) in
- \( W = 1.1 \) in
- \( h = 0.32, 0.0032 \) (thickness) in
- \( F = 1 \) lb for \( h = 0.32 \) and \( 1e-6 \) lb for \( h = 0.0032 \)
- \( E = 29E6 \) psi
- \( \nu = 0.22 \)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Thickness (in)</th>
<th>Force Direction</th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>( h = 0.32 ) in ( (S58A: SHELL9) ) ( F = 1.0 ) lb</td>
<td>In-Plane (Load case 1) Out-of-Plane (Load Case 2)</td>
<td>0.5240E-2 0.1754E-2</td>
<td>0.5397E-2 0.1759E-2</td>
</tr>
<tr>
<td>( h = 0.0032 ) in ( (S58B: SHELL9) ) ( F = 1e-6 ) lb</td>
<td>In-Plane (Load case 1) Out-of-Plane (Load Case 2)</td>
<td>0.5256E-2 0.1794E-2</td>
<td>0.4704E-2 0.1255E-2</td>
</tr>
<tr>
<td>( h = 0.32 ) ( (S58C: TETRA4R) ) ( F = 1.0 ) lb.</td>
<td>In-Plane (Load case 1) Out-of-Plane (Load Case 2)</td>
<td>0.5240E-2 0.1754E-2</td>
<td>0.4967E-2 0.1600E-2</td>
</tr>
</tbody>
</table>
Figure S58-1

Problem Description and Finite Element Mesh (1 x 6)

- Twist = 90°
- F (out-of-plane)
- F (in-plane)
- W
**S59A, S59B, S59C: Sandwich Square Plate Under Uniform Loading (SHELL9L)**

**TYPE:**
Static analysis, 4- and 9-node composite shell elements (SHELL4L, SHELL9L), solid composite element (SOLIDL).

**REFERENCE:**

**PROBLEM:**
A square sandwich plate consisting of two identical facings and an aluminum honeycomb core is subjected to uniform loading as shown in the figure below. Determine the central deflection of the plate at point A.

**GIVEN:**

*Facing:*
- \( E = 10.5 \times 10^6 \) ksi
- \( v = 0.3 \)
- \( h_f = 0.015 \) in (thickness)

*Core:*
- \( E = 0 \) ksi
- \( a = 25 \) in
- \( G_{xx} = G_{yy} = 50 \) ksi
- \( P = 9.2311 \) psi
- \( h_c = 1 \) in (thickness)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Reference</th>
<th>( W_{\text{max at the Center}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>SHELL4L (S59B)</td>
<td>0.846</td>
</tr>
<tr>
<td>SHELL9L (S59A)</td>
<td>0.868</td>
</tr>
<tr>
<td>SOLIDL (S59C)</td>
<td>0.849</td>
</tr>
<tr>
<td><strong>COSMOS/M</strong></td>
<td></td>
</tr>
<tr>
<td>SHELL4L (S59B)</td>
<td>0.851</td>
</tr>
<tr>
<td>SHELL9L (S59A)</td>
<td>0.866</td>
</tr>
<tr>
<td>SOLIDL (S59C)</td>
<td>0.849</td>
</tr>
</tbody>
</table>

**MODELING HINTS:**
Due to symmetry, one quarter of the plate is modeled. To ensure computational stability, a small elastic modulus (\( E = 1.0 \times 10^{-12} \)) for the core is used.
Figure S59A-1

Problem Sketch and Finite Element Model
**S60: Clamped Square Plate Under Uniform Loading**

**TYPE:**
Static analysis, 9-node shell element (SHELL9).

**REFERENCE:**

**PROBLEM:**
Determine the maximum deflection (at point A) of a clamped-clamped plate (shown in the figure below) with uniform loading and modeled by a skewed mesh. Various span-to-depth ratios are investigated.

**GIVEN:**
- \( E = 1 \times 10^7 \) psi
- \( \nu = 0.3 \)
- \( a = 2 \) in
- \( q = 1 \) psi (0.01 psi is used for thickness 0.002)
- \( t = \) thickness = 0.2, 0.02, and 0.002 in

**MODELING HINTS:**
Due to symmetry, only one quarter of the plate is modeled.

**ANALYTICAL SOLUTION:**

\[
U_a = 0.00126 \frac{qa^4}{D}
\]

*Where:*

\[
D = \frac{Et^3}{12(1 - \nu^2)}
\]
Chapter 2  Linear Static Analysis

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th>Span/Thickness Ratio *</th>
<th>Deflection (inch)</th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 (q = 1.0 psi)</td>
<td>-2.7518E-6</td>
<td>-3.4758E-6</td>
<td></td>
</tr>
<tr>
<td>100 (q = 1.0 psi)</td>
<td>-2.7518E-3</td>
<td>-3.0649E-3</td>
<td></td>
</tr>
<tr>
<td>1,000 (q = 0.01 psi)</td>
<td>-2.7518E-2</td>
<td>-2.79259E-2</td>
<td></td>
</tr>
</tbody>
</table>

* The input file provided (S60.GEO) is for a span/thickness ratio of 10. You need to redefine the thickness for other ratios using the RCONST command.

Better accuracy can be obtained with a finer mesh.

Figure S60-1

Problem Sketch

Finite Element Model (2 x 2 Skew)
S61: Single-Edge Cracked Bend Specimen, Evaluation of Stress Intensity Factor Using Crack Element

TYPE:
Static analysis, crack element, stress intensity factor, 8-node plane continuum element (PLANE2D).

REFERENCE:

PROBLEM:
Determine the stress intensity factor of a single-edge-cracked bend specimen using the crack element.

GIVEN:
\( E = 30 \times 10^6 \) psi  
\( \nu = 0.3 \)  
Thickness = 1 in  
\( a = 2 \) in  
\( b = 4 \) in  
\( L = 32 \) in  
\( P = 1 \) lb

COMPARISON OF RESULTS:
<table>
<thead>
<tr>
<th>( K_I )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory ( 10.663 )</td>
</tr>
<tr>
<td>COSMOS/M ( 9.855 )</td>
</tr>
</tbody>
</table>
Chapter 2  Linear Static Analysis

S62: Plate with Central Crack

TYPE:
Static analysis, crack stress intensity factor, 8-node plane continuum element (PLANE2D).

REFERENCE:

PROBLEM:
Determine the stress intensity factor of the center-cracked plate.

GIVEN:
\( E = 30 \times 10^6 \) psi
\( \nu = 0.3 \)
Thickness = 1 in
\( W = 20 \) in
\( a = 2 \) in
\( p = 1 \) lb/in

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>( K_I )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>2.5703</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>2.668</td>
</tr>
</tbody>
</table>
**S63: Cyclic Symmetry Analysis of a Hexagonal Frame**

**TYPE:**
Static analysis, cyclic symmetry, truss elements (TRUSS2D).

**REFERENCE:**

**PROBLEM:**
The pin-jointed plane hexagon is loaded by equal forces $P$, each radial from center $0$. All lines are uniform and identical. Find the radial displacement of a typical node.

**GIVEN:**
- $r = 120$ in
- $L = 120$ in
- $A = 10$ in$^2$
- $P = 3000$ lb
- $E = 30E6$ psi

**MODELING HINTS:**
Taking advantage of the cyclic symmetry of the model and noting that the model displaces radically the same amount at all six nodes, only one element is considered with the radial degree of freedom coupled in the cylindrical coordinate system.

**COMPARISON OF RESULTS:**
Radial Displacement = $\frac{2PL}{AE} = \frac{(3000)(120)}{(10)(30E6)} = 0.0012$ in

<table>
<thead>
<tr>
<th>Radial Displacement</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Theory</strong></td>
</tr>
<tr>
<td><strong>COSMOS/M</strong></td>
</tr>
</tbody>
</table>
S64A, S64B: Cyclic Symmetry

**TYPE:**
Static analysis, cyclic symmetry, 3-node triangular elements (TRIANG).

**REFERENCE:**

**PROBLEM:**
A hexagonal shaped plate is loaded by a set of radial forces as shown in the figure below. Calculate the deformation of the structure at the point where the load is applied. The plate is considered as a plane stress problem and modeled with 3-node triangular plane elements.

**GIVEN:**
- \( R = 10 \text{ in} \)
- \( t = 1 \text{ in} \)
- \( P = 3000 \text{ lb} \)
- \( E = 30E6 \text{ psi} \)

**MODELING HINTS:**
This plate is built by combining six sub-structures at 60 degree angles relative to one another. Taking advantage of the cyclic sub-structure may be considered for analysis. Note that for the sub-structure shown, the displacements of nodes along A-A and B-B must be the same in the radial directions. Therefore, these nodes will be coupled radially in the cylindrical coordinate system. All degrees of freedom in the circumferential direction will be fixed.

**COMPARISON OF RESULTS:**
The problem is solved for both the full structure and the sub-structure with the displacements coming out identical for the corresponding nodes.

<table>
<thead>
<tr>
<th>Displacement at Point A</th>
<th>X-Displacement</th>
<th>Y-Displacement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full Model (S64A)</td>
<td>-9.445E-5</td>
<td>-1.636E-4</td>
</tr>
<tr>
<td>Cyclic Part (S64B)</td>
<td>-9.450E-5</td>
<td>-1.637E-4</td>
</tr>
</tbody>
</table>

Figure S64-1
**PROBLEM:**
A large cylindrical tank is filled with an incompressible liquid. The tank rotates at a constant velocity about its vertical axis as shown. Determine the deflection of the tank wall and the bending and shear stresses at the bottom of the tank wall.

**GIVEN:**
- \( r = 48 \text{ in} \)
- \( h = 20 \text{ in} \)
- \( t = 1 \text{ in} \)

**Fluid:**
- \( \rho = 0.9345 \times 10^{-4} \text{ lb-sec}^2/\text{in}^4 \)
- \( b = 30 \times 10^4 \text{ psi} \)

**Tank:**
- \( E = 3 \times 10^7 \text{ psi} \)
- \( \nu = 0.3 \)
- \( \omega = 1 \text{ rad/sec} \)
- \( g = 386.4 \text{ in/sec}^2 \)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Y (in)</th>
<th>Point</th>
<th>Deflection in x-direction ((10^{-6} \text{ in}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>A</td>
<td>7.381</td>
</tr>
<tr>
<td>16</td>
<td>B</td>
<td>19.544</td>
</tr>
<tr>
<td>12</td>
<td>C</td>
<td>27.805</td>
</tr>
<tr>
<td>8</td>
<td>D</td>
<td>27.161</td>
</tr>
<tr>
<td>4</td>
<td>E</td>
<td>13.604</td>
</tr>
<tr>
<td>0</td>
<td>F</td>
<td>0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \sigma_{yy} ) (\text{max}), psi</td>
<td>52.24</td>
</tr>
<tr>
<td>( Q_0 ), lb/in</td>
<td>3.76</td>
</tr>
</tbody>
</table>

**Where:**
- \( b \) = Bulk modulus
- \( g \) = Accel. due to gravity
- \( \rho \) = Density
- \( E \) = Young’s modulus
- \( \nu \) = Poisson’s ratio

- **Note:** Compatibility is imposed along the direction normal to the interface using the CPDOF command (LoadsBC, Structural, Coupling, Define DOF Set).
**ANALYTICAL SOLUTIONS:**

1. **Deflection** $w(y)$:

$$w = \frac{\gamma r^2}{E_l} \left[ (h_y-y) e^{-\beta y} \left( h \cos \beta y + \left( h - \frac{1}{\beta} \right) \sin \beta y \right) \right] \frac{P r^2}{E_l} \left[ 1 - e^{-\beta y} (\cos \beta y + \sin \beta y) \right]$$

$$\beta^4 = \frac{3(1 - \nu^2)}{r^2 t^2}$$

$\lambda = \rho g$

$P$ = pressure applied on the tank wall due to an angular velocity

2. **End Moment** $M_0$:

$$M_0 = \left( 1 - \frac{1}{\beta} \right) \frac{\gamma r h t}{\sqrt{12(1 - \nu^2)}} + \frac{pr t}{\sqrt{12(1 - \nu^2)}}$$

$$\sigma_{yy}(\text{max}) = \frac{M_0 t}{2I}$$

3. **End Shear force** $Q_0$:

$$Q_0 = \frac{\gamma r h t}{\sqrt{12(1 - \nu^2)}} \left( 2\beta - \frac{1}{h} \right) + \frac{pr t}{\sqrt{3(1 - \nu^2)}} \beta$$
**S66: Fluid-Structure Interaction, Acceleration of a Tank of Fluid**

**TYPE:**
Static analysis, plane strain solid (PLANE2D) and plane fluid (PLANE2D) elements.

**REFERENCE:**

**PROBLEM:**
A large rectangular tank is filled with an incompressible liquid. The tank has a constant acceleration to the right, as shown. Determine the deflection of the tank walls and the bending and shear stresses at the bottom of the right tank wall.

**GIVEN:**

- \( r = 48 \text{ in} \)
- \( h = 20 \text{ in} \)
- \( t = 1 \text{ in} \)
- Fluid:
  - \( \rho = 0.9345 \times 10^{-4} \text{ lb-sec}^2/\text{in}^4 \)
  - \( b = 30E4 \text{ psi} \)
- Tank:
  - \( E = 3E7 \text{ psi} \)
  - \( v = 0.3 \)
  - \( a = 45 \text{ in/sec}^2 \)
  - \( g = 386.4 \text{ in/sec}^2 \)

**Where:**

- \( b \) = Bulk modulus
- \( g \) = Gravity Accel.
- \( \rho \) = Density
- \( E \) = Young's modulus
- \( v \) = Poisson’s ratio

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Point</th>
<th>Y (in)</th>
<th>( W_R ) (inch)</th>
<th>( W_L ) (inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Theory</td>
<td>COSMOS/M</td>
<td>Theory</td>
</tr>
<tr>
<td>A</td>
<td>20</td>
<td>2.137E-3</td>
<td>2.150E-3</td>
</tr>
<tr>
<td>B</td>
<td>16</td>
<td>1.591E-3</td>
<td>1.602E-3</td>
</tr>
<tr>
<td>C</td>
<td>12</td>
<td>1.054E-3</td>
<td>1.062E-3</td>
</tr>
<tr>
<td>D</td>
<td>8</td>
<td>5.564E-4</td>
<td>5.619E-4</td>
</tr>
<tr>
<td>E</td>
<td>4</td>
<td>1.662E-4</td>
<td>1.689E-5</td>
</tr>
<tr>
<td>F</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>( \sigma_{yy} ) (max), psi (at ( y = 0 ))</th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>( V_0 ), lb/in (at ( y = 0 ))</td>
<td>410.02</td>
<td>386.39</td>
</tr>
<tr>
<td>9.24</td>
<td>8.84*</td>
<td></td>
</tr>
</tbody>
</table>

*This value is calculated by averaging TXY at the nodes located at the bottom of the right wall giving half weight to corner nodes.

- Note: Compatibility is imposed along the direction normal to the interface using the CPDOF command (LoadsBC, Structural, Coupling, Define DOF Set).
ANALYTICAL SOLUTIONS:

1. Deflections of the right wall $W_R(y)$ and the left wall $W_L(y)$:

$$W_R = \frac{p_0 y^2}{120EI} \left(10h^3 - 10h^2 + 5hy^2 - y^3\right) + \frac{p_1 y^2}{24EI} \left(6h^2 - 4hy + y^2\right)$$

$$W_L = \frac{p_0 y^2}{120EI} \left(10h^3 - 10h^2 + 5hy^2 - y^3\right) + \frac{p_2 y^2}{24EI} \left(6h^2 - 4hy + y^2\right)$$

Where:

- $p_0 = \rho gh$
- $p_1$ = pressure applied on the right wall due to acceleration
- $p_2$ = pressure applied on the left wall due to acceleration
- $E = E/(1-\nu^2)$

2. End Moment $M_0$

$$M_0 = -EI \frac{d^2 W}{dy^2}$$

3. End Shear Force $V_0$

$$V_0 = \tau A$$

$$V_0 = -EI \frac{d^3 W}{dy^3}$$

Figure S66-1

Problem Sketch

Finite Element Model
S67: MacNeal-Harder Test

**TYPE:**
Static analysis, plane stress quadrilateral p-element (8-node PLANE2D) with the polynomial order of shape functions equal to 5.

**PROBLEM:**
Calculate the maximum deflection of a cantilever beam loaded by a concentrated end force.

**GIVEN:**

<table>
<thead>
<tr>
<th>Geometric Properties</th>
<th>Material Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>h = 0.2 in</td>
<td>E = 1 x 10^7 psi</td>
</tr>
<tr>
<td>t = 0.1 in</td>
<td>v = 0.3</td>
</tr>
<tr>
<td>L = 6 in</td>
<td>Loading:</td>
</tr>
<tr>
<td>I = 2/3 x 10^{-4} in^4</td>
<td>P = 1 lb</td>
</tr>
</tbody>
</table>

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tip Displacement</td>
<td>0.1081 in</td>
<td>0.10807 in</td>
</tr>
</tbody>
</table>

Figure S67-1
S68: P-Method Solution of a Square Plate with Hole

**TYPE:**
Static analysis, plane stress quadrilateral (8-node PLANE2D) and triangular (6-node TRIANG) p-elements with the polynomial order of shape functions equal to 5.

**PROBLEM:**
Calculate the maximum stress of a plate with a circular hole under a uniformly distributed tension load.

**GIVEN:**

*Geometric Properties:*
- \( L = 12 \text{ in} \)
- \( d = 1 \text{ in} \)
- \( t = 1 \text{ in} \)

*Material Properties:*
- \( E = 30 \times 10^6 \text{ psi} \)
- \( \nu = 0.3 \)

*Loading:*
- \( p = 1000 \text{ psi} \)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Theory</th>
<th>COSMOS/M (PORD = 5)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max. Stress in X-Direction</td>
<td>3018</td>
</tr>
</tbody>
</table>

Figure S68-1
S69: P-Method Analysis of an Elliptic Membrane Under Pressure

**TYPE:**
Static analysis, plane stress triangular p-element (6-node TRIANG).

**REFERENCE:**

**PROBLEM:**
Calculate the stresses at point D of an elliptic membrane under a uniform outward pressure.

**GIVEN:**
- $E = 210 \times 10^3$ MPa
- $\nu = 0.3$
- $t = 0.1$
- $p = 10$ MPa

**COMPARISON OF RESULTS**

<table>
<thead>
<tr>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>92.70</td>
<td>93.72</td>
</tr>
</tbody>
</table>

**Figure S69-1**

All dimensions in meters
Thickness = 0.1
S70: Thermal Analysis with Temperature Dependent Material

**TYPE:**
Linear thermal stress analysis, plane continuum element (PLANE2D).

**PROBLEM:**
A flat plate consists of different material properties through its length. Determine the deflections and thermal stresses in the plate due to uniform changes of temperature equal to 100° F and 200° F.

**GIVEN:**
- \( t = 0.1 \) in
- \( x = 0.00001 \) in/in/°F
- \( \nu = 0 \)
- \( E = 30,000 \) ksi

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>( T = 100^\circ ) F</th>
<th>( T = 200^\circ ) F</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>-30 ksi</td>
<td>-48 ksi</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>-30 ksi</td>
<td>-48 ksi</td>
</tr>
</tbody>
</table>

* The temperature in the input file corresponds to \( T = 200^\circ \) F. You need to delete the applied temperature using the NTNDEL command and apply temperature of 100° F using the NTND command.

---

Figure S70-1

Finite Element Model

\[ E = 30E3 \text{ ksi} \] (CONSTANT)

\[ E \text{ (ksi)} \]

\[ 30000 \]

\[ 20000 \]

\[ 150 \]

\[ 200 \]

\[ \text{Temperature} \]

Elements 1 & 2

Elements 3 & 4
**S71: Sandwich Beam with Concentrated Load**

**TYPE:**
Static analysis, composite shell element (SHELL4L).

**PROBLEM:**
Determine the total deflection of the sandwich beam subjected to a concentrated load.

**GIVEN:**

\[
\begin{align*}
E_t &= 7000 \text{ N/mm}^2 \\
t &= 3 \text{ mm} \\
E_c &= 20 \text{ N/mm}^2 \\
c &= 25 \text{ mm} \\
G_c &= 5 \text{ N/mm}^2 \\
d &= 28 \text{ mm} \\
L &= 1000 \text{ mm} \\
b &= 100 \text{ mm} \\
W &= 250 \text{ N}
\end{align*}
\]

**THEORY:**

\[
D = E_t b t^3/6 + E_t b t d^2/2 + E_c b c^3/12 = 8.28 \times 10^6 \text{ N.mm}^2
\]

\[
\delta = W L^3/48D + W L c/4 b d G = 6.2902 + 3.986 = 10.276 \text{ mm}
\]

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Midspan Deflection (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>10.276</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>10.323</td>
</tr>
</tbody>
</table>

---

**Figure S71-1**
S74: Constant Stress Patch Test (TETRA4R)

**TYPE:**
Static analysis, tetrahedral elements (TETRA4, TETRA4R).

**PROBLEM:**
Constraint displacements at one end and prescribed displacements at the other end of the plate to produce a constant stress state with $\sigma_x = 0.1667E5$ and $\sigma_y = \sigma_z = \tau_{xy} = \tau_{yz} = \tau_{zx} = 0$.

Patch test model.

**GIVEN:**
- $E_x = 1E8$
- $\nu = 0.25$
- $\delta x = 0.4E-2$
- $t = 0.024$
- $a = 0.12$
- $b = 0.24$

**RESULTS:**
All the above elements pass the patch test. The nodal stresses show that $\sigma_x = 0.1667E5$ and $\sigma_y = \sigma_z = \tau_{xy} = \tau_{yz} = \tau_{zx} = 0$.

Figure S74-1

Finite Element Model for Patch Test
**S75: Analysis of a Cantilever Beam with Gaps, Subject to Different Loading Conditions**

**TYPE:**
Linear static analysis, beam and gap elements (BEAM2D, GAP).

**PROBLEM:**
The problem is modeled using BEAM2D elements. Five gap elements with zero gap distances are used. Two different load cases were selected, and the analysis was performed.

**GIVEN:**

\[
\begin{align*}
E_{\text{beam}} &= 30 \times 10^6 \text{ psi} \\
 b &= 1.2 \text{ in} \\
 h &= 10 \text{ in} \\
 L_1 &= 100 \text{ in} \\
 L_2 &= 50 \text{ in}
\end{align*}
\]

**COMPARISONS OF RESULTS:**
The deformation state of gaps for each load case agrees with the beam deformed shape corresponding to that load case. The results can be compared with the solution obtained from linear static analysis, where the gaps are removed and the nodes at the closed gaps are fixed.

**OBTAINED RESULTS:**

<table>
<thead>
<tr>
<th>Applied Forces</th>
<th>Load Case</th>
<th>Gap 1</th>
<th>Gap 2</th>
<th>Gap 3</th>
<th>Gap 4</th>
<th>Gap 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>[ Fa = -1000 ] [ Fc = -2000 ]</td>
<td>1</td>
<td>-361.84</td>
<td>-1197.4</td>
<td>-842.11</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>[ Fa = -1000 ] [ Fb = -1000 ]</td>
<td>2</td>
<td>-1206.3</td>
<td>-275.0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
Chapter 2  Linear Static Analysis

Figure S75-1

Problem Sketch

Finite Element Model

Load Case 1  Load Case 2
\( F_a = -1000 \quad F_a = -1000 \)
\( F_c = -2000 \quad F_b = -1000 \)
S76: Simply Supported Beam Subject to Pressure from a Rigid Parabolic Shaped Piston

**TYPE:**  
Linear static analysis, beam, plane and gap elements (BEAM2D, PLANE2D, TRUSS2D and GAP).

**PROBLEM:**  
The shape of the piston is simulated through gap distances. In order to avoid singularities in the structure stiffness, two soft truss elements are used to hold the piston. The problem is analyzed for two different pressure values.

**GIVEN:**  
*Gap Distances:*  
\[ g_1 = g_7 = 0.027 \text{ in} \]  
\[ g_2 = g_6 = 0.001 \text{ in} \]  
\[ g_3 = g_5 = 0.008 \text{ in} \]  
\[ g_4 = 0 \text{ in} \]  
\[ h = 10 \text{ in} \]  
\[ b = 1.2 \text{ in} \]  
\[ k = 1 \text{ lb/in} \]  
\[ E = 30 \times 10^6 \text{ psi} \]  
Load case 1: \( P = 52.5 \text{ psi} \)  
Load case 2: \( P = 90.8 \text{ psi} \)

**COMPARISON OF RESULTS:**  
The forces in the gap elements at a particular of time are in good agreement with the total force applied to the piston at that time. The deformed shape of the beam for each load case is compatible with the forces and location of closed gaps for that load case.
### Forces in Gap Elements (lb)

<table>
<thead>
<tr>
<th>No. of Closed Gaps</th>
<th>Pressure</th>
<th>Gap Forces</th>
<th>Total Force (Theory)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>p = 52.5</td>
<td>-215.5 -215.5 -1668.0 -1668.0</td>
<td>-3767 -3780</td>
</tr>
<tr>
<td>2</td>
<td>p = 90.8</td>
<td>-3259.0 -3259.0</td>
<td>-6518 -6540</td>
</tr>
</tbody>
</table>

**Figure S76-1**

- **Problem Sketch**
  - y = ((x/100)^3
  - 120 in 60 in 120 in

- **Finite Element Model**
  - 24 25 11 10 9 8 7 6 5 4 3 2
S77: Bending of a Solid Beam Using Direct Material Matrix Input

TYPE:
Static analysis, direct material property input, hexahedral solid element (SOLID).

REFERENCE:

PROBLEM:
A beam of length L, width b, and height h is built-in at one end and loaded at the free end with a shear force F. Determine the deflection at the free end.

GIVEN:
L = 10 in
E = 30E6 psi
b = 1 in
υ = 0.3
h = 2 in
F = 300 lb

MODELING HINTS:
Instead of specifying the elastic material properties by E and υ, the elastic matrix [D] shown below is provided by direct input of its non-zero terms.

\[
[D] = \begin{bmatrix}
MC11 & MC12 & MC13 & MC14 & MC15 & MC16 \\
MC22 & MC23 & MC24 & MC25 & MC26 \\
MC33 & MC34 & MC35 & MC36 \\
MC44 & MC45 & MC46 \\
MC55 & MC56 \\
\text{Sym.} & MC66
\end{bmatrix}
\]
RESULTS:
Displacement in Z-direction at the tip using E and $\nu$, is compared with those obtained with direct input of elastic coefficients in matrix [D].

<table>
<thead>
<tr>
<th></th>
<th>Using E and $\nu$</th>
<th>Using Direct Matrix Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>0.005</td>
<td>0.005</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>0.00496</td>
<td>0.00496</td>
</tr>
</tbody>
</table>

Figure S77-1

Problem Geometry

Finite Element Model
S78: P-Adaptive Analysis of a Square Plate with a Circular Hole

**TYPE:**
P–adaptive analysis, plane stress triangular p-element (TRIANG).

**PROBLEM:**
Calculate the maximum stress of a plate with a circular hole under a uniform distributed tension load.

**GIVEN:**

*Geometric Properties:*
- \( L = 200 \text{ in} \)
- \( d = 20 \text{ in} \)
- \( E = 30 \times 10^6 \text{ psi} \)
- \( t = 1 \text{ in} \)

*Loading:*
- \( p = 1 \text{ psi} \)

**RESULTS:**
Nodes: 33, elements: 12, allowable local displacement error: 5%.

<table>
<thead>
<tr>
<th>Iter. No.</th>
<th>Min. ( p )</th>
<th>Max. ( p )</th>
<th>d.o.f.</th>
<th>Energy ( \times 10^{-4} )</th>
<th>Max. Displ. ( \times 10^{-6} )</th>
<th>Max. Stress</th>
<th>Local Displ. Error %</th>
<th>No. of Sides Not Converged</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>16</td>
<td>1.692</td>
<td>3.468</td>
<td>1.586</td>
<td>--</td>
<td>22</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>2</td>
<td>56</td>
<td>1.701</td>
<td>3.480</td>
<td>2.418</td>
<td>29.544</td>
<td>16</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>3</td>
<td>85</td>
<td>1.704</td>
<td>3.489</td>
<td>2.692</td>
<td>12.844</td>
<td>8</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>4</td>
<td>100</td>
<td>1.704</td>
<td>3.490</td>
<td>2.817</td>
<td>1.083</td>
<td>0</td>
</tr>
<tr>
<td>Ref.</td>
<td>4</td>
<td>4</td>
<td>133</td>
<td>1.706</td>
<td>3.502</td>
<td>2.994</td>
<td>--</td>
<td>--</td>
</tr>
</tbody>
</table>
Chapter 2   Linear Static Analysis

Figure S78-1

Plate with a Hole

Figure S78-2

Polynomial Order for Each Side at Iteration No. 4
**S79: Hemispherical Shell Under Unit Moment Around Free Edge**

**TYPE:**
Static linear analysis using the asymmetric loading option (SHELLAX).

**REFERENCE:**

**PROBLEM:**
Determine the radial displacement of a hemispherical shell under a uniform unit moment around the free edge.

**GIVEN:**
- \( R = 100 \text{ in} \)
- \( r = 50 \text{ in} \)
- \( t = 2 \text{ in} \)
- \( E = 1E7 \text{ psi} \)
- \( M = 1 \text{ in-lb/in} \)
- \( \nu \text{ (NUXY)} = 0.33 \)

**MODELING HINTS:**
It is important to note that nodal load is to be specified per unit radian which in this case is 50 in-lb/rad.

\[ M_t = M_x \text{ Arcx} = M_x \text{ Rx} \psi = 1(50) (1) \text{ rad} = 50 \]

*Where:* \( \psi = \text{horizontal angle} \)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Radial Displacement at Node 31</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
</tr>
<tr>
<td>COSMOS/M</td>
</tr>
</tbody>
</table>
**S80: Axisymmetric Hyperbolic Shell Under a Cosine Harmonic Loading on the Free Edge**

**TYPE:**
Static linear analysis using the asymmetric loading option in SHELLAX.

**REFERENCE:**

**PROBLEM:**
Determine the stress of an axisymmetric Hyperbolic shell under loading $F = \cos 2\theta$ on the outward edge, $y = 1$.

**GIVEN:**
- $R_1 = 1$ m
- $H = 1$ m
- $\tan \phi = 2^{(-1/2)}$
- $R_2 = 2^{(1/2)}$ m
- $E = 210E3$ MPa
- $\nu$ (NUXY)$ = 0.3$
- Thickness $= 0.01$

**MODELING HINTS:**
Due to symmetry only half of the shell will be modeled. The Cosine load at the free edge will be applied in terms of its x- and y- components, representing the second term of the even function for a Fourier expansion.

**COMPARISON OF RESULTS:**
The results in the following table correspond to the NXZ component of stress for element 1 as recorded in the output file.

<table>
<thead>
<tr>
<th></th>
<th>Shear Stress ($y = 0, \theta = 45^\circ$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>-81.65 MPa</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>-79.63 MPa</td>
</tr>
</tbody>
</table>
S81: Circular Plate Under Non-Axisymmetric Load

TYPE:
Static linear analysis using the asymmetric loading option in SHELLAX.

REFERENCE:
SHELL4 elements are used for comparison purposes.

PROBLEM:
A circular plate with inner and outer radii of 3 in and 10 in respectively, is subjected to a non-axisymmetric load around outer circumference from $\theta = -54^\circ$ to $\theta = 54^\circ$ perpendicular to the plate surface. The load distribution is:

$$F(\theta) = 5.31 \left[1 + \cos\left(\frac{10\theta}{3}\right)\right]\times10^3$$

GIVEN:
- $R_i = 3$ in
- $R_o = 10$ in
- $E = 3\times10^7$ psi
- $t = 1$ in
- $\nu$ (NUXY) = 0.3

MODELING HINTS:
A total of seven elements are considered in this example. Note that since the load is symmetric about the x-axis, it will be considered only between $\theta = 0^\circ$ and $\theta = 54^\circ$ at $3^\circ$ intervals, and represented by the even (Cosine) terms of the Fourier expansion. Only the first six (Cosine) terms will be included.

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th>Displacement of Outer Edges $(\theta = 180^\circ)$ in the Axial Direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>SHELLAX</td>
</tr>
<tr>
<td>SHELL4</td>
</tr>
</tbody>
</table>
Chapter 2  Linear Static Analysis

Figure S81-1

Problem Sketch

Finite Element Model

Load Distribution

\[ \theta = -54^\circ \]

\[ \theta = +54^\circ \]
S82: Twisting of a Long Solid Shaft

TYPE:
Static analysis, PLANE2D element using asymmetric loading option.

REFERENCE:

PROBLEM:
A long solid circular shaft is built-in at one end and subjected to a twisting moment at the other end. Determine the maximum shear stress, $\tau_{\text{max}}$, at the wall due to the moment.

Figure S82-1
Chapter 2  Linear Static Analysis

GIVEN:
\( E = 30E6 \) psi
\( L = 24 \) in
\( d = 1 \) in
\( M = -200 \text{ in-lb} \)

MODELING HINTS:
Since the geometry is axisymmetric about the y-axis, the finite element model, shown in the figure above, is considered for analysis. The effect of the applied moment is calculated in terms of a tangential force integrated around the circumference of the circular rod.

ANALYTICAL SOLUTION:
\[
M = - \int_0^{2\pi} F_z \frac{d}{2} d \theta = -F_z \pi d \\
F_z = -\frac{M}{\pi d} = 63.661977 \text{ lb}
\]
The load is applied at (node 63) in the z-direction (circumferential). Ux (radial) constraints are not imposed at the wall in order to allow freedom of cross-sectional deformation which corresponds to the assumptions of “negligible shear” stated in the reference.

COMPARISON OF RESULTS:
At clamped edge (node 3).

<table>
<thead>
<tr>
<th>Max Shear Stress (psi) ( \tau_{13} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
</tr>
<tr>
<td>COSMOS/M</td>
</tr>
</tbody>
</table>
S83: Bending of a Long Solid Shaft

TYPE:
Static analysis, PLANE2D element using the asymmetric loading option.

REFERENCE:

PROBLEM:
A long solid circular shaft is built-in at one end and at the other end a vertical force is applied. Determine the maximum axial stress $\sigma_y$ at the wall and at one inch from the wall due to the force.

Figure S83-1
Chapter 2  Linear Static Analysis

GIVEN:

\[ E = 30 \text{E}6 \text{ psi} \]
\[ L = 24 \text{ in} \]
\[ d = 1 \text{ in} \]
\[ F = -25 \text{ lb} \]

MODELING HINTS:

The finite element model is formed as noted in the figure considering the axisymmetric nature of the problem. The force applied at node 63 is calculated based on a Fourier Sine expansion representing its antisymmetric nature.

ANALYTICAL SOLUTION:

\[ F = \int_{0}^{2\pi} F_{x} \sin^{2} \theta \, d \theta = F_{x} \pi \quad F_{x} = \frac{F}{\pi} = 7.9577471 \text{ lb} \]

The load is applied at (node 75) in the z-direction (circumferential). Ux (radial) constraints are not imposed at the wall in order to allow freedom of cross-sectional deformation which corresponds to the assumptions of “negligible shear” stated in the reference.

COMPARISON OF RESULTS:

At element 1 and \( \theta = 90^\circ \).

<table>
<thead>
<tr>
<th>Max Axial Stress (sy psi)</th>
<th>( y = 0 ) (Node 3)</th>
<th>( y = 1 ) in (Node 5)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>6111.6</td>
<td>5856.9</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>6115.1</td>
<td>5856.8</td>
</tr>
</tbody>
</table>
**S84: Submodeling of a Plate**

**TYPE:**
Static analysis TRIANG element using the submodeling option.

**PROBLEM:**
Calculate the maximum von Mises stress for a square plate under a concentrated load at one corner. Compare the displacement and stress results from a fine mesh to the results from an originally coarse mesh improved using submodeling.

**GIVEN:**
- \(a = 25\text{ in}\)
- \(E = 30 \text{ E}6\text{ psi}\)
- \(b = 25\text{ in}\)
- \(t = 0.1\text{ in}\)
- \(F_x = F_y = 1000\text{ lbs}\)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Mesh Type</th>
<th>Max Deflection at Node 1</th>
<th>Max Stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse Mesh</td>
<td>-0.00131</td>
<td>3810</td>
</tr>
<tr>
<td>Coarse Mesh + Submodeling</td>
<td>-0.00156</td>
<td>7626</td>
</tr>
<tr>
<td>Fine Mesh and Theory</td>
<td>-0.00156</td>
<td>7620</td>
</tr>
</tbody>
</table>
S85: Plate on Elastic Foundation

**TYPE:**
Static analysis, SHELL4 plate elements on elastic foundation.

**PROBLEM:**
A simply supported plate is subjected to uniform pressure P. The full plate is supported by elastic foundation. For small flexural rigidity, the calculated pressure applied to the plate from the foundation approaches the applied external pressures. The flexural rigidity decreases by decreasing the thickness and modulus of elasticity.

**GIVEN:**
- $E = 30 \times 10$ psi
- $\nu = 0.3$
- $h = 0.01$ in
- $a = 10$ in
- $b = 10$ in
- $P = 10$ psi

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Foundation Pressure at Element 200</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Theory</strong></td>
</tr>
<tr>
<td><strong>COSMOS/M</strong></td>
</tr>
</tbody>
</table>

**NOTE:**
Foundation pressure is recorded in the output file for each element in the last column of element stress results.
S86: Plate with Coupled Degrees of Freedom

TYPE:
Static analysis, PLANE2D element, coupled degrees of freedom.

PROBLEM:
Determine displacements for the plate shown in the figure below such that translations in the Y-direction are coupled for nodes 5, 10, and 15.

GIVEN:
\[ \text{EX} = 3.0\times10^8, 3.0\times10^9, \text{and } 3.0\times10^8 \text{ psi} \]
\[ \nu = 0.25 \]

COMPARISON OF RESULTS:
Displacements for the coupled D.O.F.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>3.0E10</td>
<td>5.33333E-7</td>
<td>5.33333E-7</td>
<td>5.33333E-7</td>
<td>1.000</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td></td>
<td>5.33333E-7</td>
<td>5.33333E-7</td>
<td>5.33333E-7</td>
<td></td>
</tr>
<tr>
<td>Theory</td>
<td>3.0E09</td>
<td>5.33333E-6</td>
<td>5.33333E-6</td>
<td>5.33333E-6</td>
<td>1.000</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td></td>
<td>5.33333E-6</td>
<td>5.33333E-6</td>
<td>5.33333E-6</td>
<td></td>
</tr>
<tr>
<td>Theory</td>
<td>3.0E08</td>
<td>5.33333E-5</td>
<td>5.33333E-5</td>
<td>5.33333E-5</td>
<td>1.000</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td></td>
<td>5.33333E-5</td>
<td>5.33333E-5</td>
<td>5.33333E-5</td>
<td></td>
</tr>
</tbody>
</table>

ANALYTICAL SOLUTION:
\[ U_{10} = U_5 = \frac{FL}{AE} \]
**Chapter 2  Linear Static Analysis**

**S87: Gravity Loading of ELBOW Element**

**TYPE:**
Linear static analysis, ELBOW element with pipe cross-section subjected to gravity loading.

**PROBLEM:**
- Case A: Reduced gravity loading (fixed-end moments ignored)
- Case B: Consistent gravity loading (fixed-end moments considered)

**GIVEN:**
- \( g = -32.2 \text{ in/sec}^2 \)
- \( E = 3.0 \text{E} 7 \text{ psi} \)
- \( \rho = 7.82 \)
- Elbow wall thickness = 0.1 in
- Elbow outer diameter = 1.0 in
- Elbow radius of curvature = 10.0 in

**Figure S87-1**

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Case</th>
<th>Y-Translation at Node 51</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case A</td>
<td>-7.42E-3</td>
</tr>
<tr>
<td>Case B</td>
<td>-6.41E-3</td>
</tr>
</tbody>
</table>
S88A, S88B: Single-Edge Cracked Bend Specimen, Evaluation of Stress Intensity Factor Using the J-integral

TYPE:
Static analysis, J-integral, stress intensity factor, plane stress conditions.

S88A: Using 6-node triangular plane element (TRIANG)
S88B: Using 8-node rectangular plane element (PLANE2D)

REFERENCE:

PROBLEM:
Determine the stress intensity factor for a single-edge-cracked bend specimen using the J-integral.

GIVEN:
E = 30 x 10^6 psi
\(\nu = 0.3\)
Thickness = 1 in
a = 2 in
b = 4 in
L = 32 in
P = 1 lb

MODELING HINTS:
Three circular J-integral paths centered at the crack tip are considered. Due to symmetry, only one half of the model is modeled.
COMPARISON OF RESULTS

<table>
<thead>
<tr>
<th>Theory</th>
<th>$K_i$ (TRIANG)</th>
<th>$K_i$ (PLANE2D)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Path 1</td>
<td>9.9544</td>
<td>9.1692</td>
</tr>
<tr>
<td>Path 2</td>
<td>10.145</td>
<td>10.974</td>
</tr>
<tr>
<td>Path 3</td>
<td>10.240</td>
<td>10.648</td>
</tr>
</tbody>
</table>

Figure S88-1

Figure S88-2
S89A, S89B: Slant-Edge Cracked Plate, Evaluation of Stress Intensity Factors Using the J-integral

**TYPE:**
Static analysis, J-integral, stress intensity factors (combined mode crack), plane strain conditions.

- **S89A:** Using 6-node triangular plane element (TRIANG)
- **S89B:** Using 3-node triangular plane element (TRIANG)
- **S89C:** Using 8-node rectangular plane element (PLANE2D)
- **S89D:** Using 4-node rectangular plane element (PLANE2D)

**REFERENCE:**

**PROBLEM:**
Determine the stress intensity factor for both modes of fracture (opening and shearing) for a rectangular plate with an inclined edge crack subjected to uniform uniaxial tensile pressure at the two ends.

**GIVEN:**
- \( \sigma = 1 \text{ psi} \)
- \( h = 2.5 \text{ in} \)
- \( W = 2.5 \text{ in} \)
- \( a = 1 \text{ in} \)
- \( E = 30 \times 10^6 \text{ psi} \)
- \( \nu = 0.3 \)
- Thickness =1 in
- \( \phi = 45^\circ \)

**MODELING HINTS:**
The full part has to be modeled since the model is not symmetric with respect to the crack. There is no restriction in the type of the mesh to be used and the mesh could
be either symmetric or non-symmetric with respect to the crack. However, the nodes in the two sides of crack should not be merged in order to model the rupture area properly.

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Element Type</th>
<th>Reference</th>
<th>Path 1</th>
<th>Path 2</th>
<th>$K_I$</th>
<th>$K_{II}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>6-node element</td>
<td>S89A.GEO</td>
<td>1.82</td>
<td>1.82</td>
<td>1.85</td>
<td>0.880</td>
</tr>
<tr>
<td>3-node element</td>
<td>S89B.GEO</td>
<td>1.76</td>
<td>1.77</td>
<td>1.85</td>
<td>0.880</td>
</tr>
<tr>
<td>8-node element</td>
<td>S89C.GEO</td>
<td>1.80</td>
<td>1.79</td>
<td>1.85</td>
<td>0.880</td>
</tr>
<tr>
<td>4-node element</td>
<td>S89D.GEO</td>
<td>1.73</td>
<td>1.71</td>
<td>1.85</td>
<td>0.880</td>
</tr>
</tbody>
</table>

**Figure S89-1**

**Figure S89-2**
**S90A, S90B: Penny-Shaped Crack in Round Bar, Evaluation of Stress Intensity Factor Using the J-integral**

**TYPE:**
Static analysis, J-integral, stress intensity factor, axisymmetric geometry.

- **S90A:** Using 8-node rectangular plane element (PLANE2D)
- **S90B:** Using 6-node triangular plane element (TRIANG)

**REFERENCE:**

**PROBLEM:**
Determine the stress intensity factor for a circular crack inside a round bar subjected to uniform axial tensile pressure at the two ends.

**GIVEN:**
- \( \sigma = 1 \) psi
- \( H = 25 \) in
- \( R = 5 \) in
- \( a = 2.5 \) in
- \( E = 30 \times 10^6 \) psi
- \( \gamma = 0.28 \)

**MODELING HINTS:**
Since the model is symmetric with respect to the crack, therefore only one-half of the model (lower half here) is needed for the analysis.
COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>Reference</th>
<th>( K_I )</th>
</tr>
</thead>
<tbody>
<tr>
<td>8-node element (S90A.GEO)</td>
<td>Path 1</td>
<td>1.90</td>
</tr>
<tr>
<td></td>
<td>Path 2</td>
<td>1.91</td>
</tr>
<tr>
<td>6-node element (S90B.GEO)</td>
<td>Path 1</td>
<td>1.89</td>
</tr>
<tr>
<td></td>
<td>Path 2</td>
<td>1.90</td>
</tr>
</tbody>
</table>

Figure S90-1

Figure S90-2

path 1

path 2
PART 2   Verification Problems

S91: Crack Under Thermal Stresses, Evaluation of Stress Intensity Using the J-integral

TYPE:
Static analysis, thermal loading, J-integral, stress intensity factor, plane strain conditions.

REFERENCE:

PROBLEM:
Determine the stress intensity factor for an edge crack strip subjected to thermal loading. The strip is subjected to a linearly varying temperature through its thickness with zero temperature at mid-thickness and temperature \( T_0 \) at the right edge \( (x=w/2) \). The ends are constrained.

GIVEN:
\[
egin{align*}
L & = 20 \text{ in} \\
w & = 10 \text{ in} \\
a & = 5 \text{ in} \\
E & = 30 \times 10^6 \text{ psi} \\
\gamma & = 0.28 \\
\alpha & = 7.4 \times 10^{-6} \text{ in/in}^{-\circ} \text{F} \\
T_0 & = 10 \circ \text{F}
\end{align*}
\]

MODELING HINTS:
Due to symmetry, only one-half of the geometry is modeled (lower half in this problem).

COMPARISON OF RESULTS:
\[
\beta = \frac{K_T}{\left(\sigma_T \sqrt{\pi a}\right)}, \quad \sigma_T = E\alpha T_0 / \left(1 - \gamma\right)
\]

where: \( \beta = K_T / 12220.27 \)
Chapter 2  Linear Static Analysis

<table>
<thead>
<tr>
<th>Path</th>
<th>$K_i$</th>
<th>$K_i/\beta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference</td>
<td>0.5036</td>
<td>*</td>
</tr>
<tr>
<td>Path 1</td>
<td>6141.4</td>
<td>0.5034</td>
</tr>
<tr>
<td>Path 2</td>
<td>6176.3</td>
<td>0.5054</td>
</tr>
</tbody>
</table>

*Average value of the five paths in the reference
S92A, S92B: Simply Supported Rectangular Plate, Using Direct Material Matrix Input

TYPE:
Static analysis, direct material input, SHELL3L element.

REFERENCE:

PROBLEM:
Calculate the deflection and stresses at the center of a simply supported plate subjected to a concentrated load F.

GIVEN:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>30 x 10^6 psi</td>
</tr>
<tr>
<td>G_{xy} = G_{yz} = G_{xz}</td>
<td>11.538 x 10^6 psi</td>
</tr>
<tr>
<td>ν</td>
<td>0.3</td>
</tr>
<tr>
<td>h</td>
<td>1 in</td>
</tr>
<tr>
<td>a = b</td>
<td>40 in</td>
</tr>
<tr>
<td>F</td>
<td>400 lbs</td>
</tr>
</tbody>
</table>

MODELING HINTS:
Instead of specifying the elastic properties by E and ν, the elastic matrix [D] shown below (in the default element coordinate system) is provided by direct input of its non-zero terms.

\[
[D] = \begin{bmatrix}
MC11 & MC12 & 0 & 0 & 0 \\
MC22 & 0 & 0 & 0 & 0 \\
MC44 & 0 & 0 & 0 & 0 \\
Sym & MC55 & 0 & 0 & 0 \\
\end{bmatrix}
\]

where, [D] relates the element strains to the element stresses according to Hook's law:

\[
\begin{bmatrix}
\sigma_x, \sigma_y, \sigma_{xy}, \sigma_{yz}, \sigma_{xz} \\
\end{bmatrix}^T = [D] \begin{bmatrix}
e_x, e_y, \gamma_{xy}, \gamma_{yz}, \gamma_{xz} \\
\end{bmatrix}
\]

Note that the [D] matrix is reduced to a 5x5 matrix from the general form of 6x6 matrix, by considering the fact that \(\sigma_z = 0\) for shell element, thus eliminating the third row and column of the general [D] matrix.
Considering an isotropic property, the terms of $[D]$ matrix are:

$$MC_{11} = MC_{22} = \frac{E}{1-\nu^2} \approx 32,967,000.$$

$$MC_{12} = \frac{E\nu}{1-\nu^2} \approx 9,890,000.$$

$$MC_{44} = G_{xy} = \frac{E}{2(1+\nu)} \approx 11,538,000.$$

$$MC_{55} = K_1 G_{yz} \approx 1,159,600$$

$$MC_{66} = K_2 G_{xy} \approx 1,159,600$$

The terms $K_1$ and $K_2$ are shear correction factors which are chosen to match the plate theory with certain classical solutions and are functions of thickness and material properties. When you input regular material properties ($E$, $\nu$), the shear factors are evaluated internally in the program as $K_1 = K_2 = 0.1005$ (as in S92B). For the sake of consistency, the same values are used for the evaluation of $MC_{55}$ and $MC_{66}$ in S92A.

Due to symmetry in geometry and load, only a quarter of the plate is modeled.

**COMPARISON OF RESULTS:**

Maximum displacement (in Z-direction) at the tip of the plate (Node 25) using $E$ and $\nu$ (S92B) is compared with the result obtained from direct input of the elastic coefficients in matrix $[D]$ (S92A).

<table>
<thead>
<tr>
<th>Theory</th>
<th>Using Direct Matrix Input (S92A)</th>
<th>Using E and $\nu$ (S92B)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum UZ (in)</td>
<td>-0.0270</td>
<td>-0.02746</td>
</tr>
</tbody>
</table>
S93: Accelerating Rocket

**TYPE:**
Static analysis, inertia relief, PLANE2D element, (axisymmetric option).

**PROBLEM:**
A cylinder is accelerating under unbalanced external loads. Find the induced *counter-balance acceleration* and the amount by which the cylinder will be shortened.

**GIVEN:**
- \( E = 3.87 \text{ psi} \)
- \( \gamma = 0.28 \text{ lb sec}^2/\text{in}^4 \)
- \( \rho = 7.3\times10^{-4} \)

---

![Diagram of Actual Model and Finite Element Model](image-url)
MODELING HINTS:
To avoid instability in FEA solution, one node should be constrained in Y-direction. A node on the top end of the cylinder is selected for that purpose rather than on the bottom end. Constraining any node on the surface where the pressure is applied eliminates the components of the load of that node and hence causes inaccuracy in the solution.

ANALYTICAL SOLUTION:

a) The induced counter-balance acceleration:
\[ F + Ma = 0 \]
\[ P \pi R^2 = -\rho \pi R^2 Ha \]
\[ a = -\frac{P}{\rho H} = \frac{-100}{0.00073 (100)} = -1370 \]

b) Length shortening
\[ \delta = \int_0^H \varepsilon d\eta \]
\[ \varepsilon = \frac{\sigma}{E} = \frac{\rho a \eta}{E} \]
\[ \delta = \frac{\rho a}{E} \int_0^H \eta d\eta = \frac{\rho a H^2}{2E} = 0.0001667 \]

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>Acceleration (a)</th>
<th>Displacement ( u_y ) at Node 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>-1370</td>
<td>0.0001667</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>-1370*</td>
<td>0.0001669</td>
</tr>
</tbody>
</table>

* See output file.
Part 2  Verification Problems

S94A, S94B, S94C: P-Method Solution of a Square Plate with a Small Hole

TYPE:

PROBLEM:
Calculate the maximum stress of a plate with a circular hole under a uniformly distributed tension load. Use strain energy to adapt the p-order.

GIVEN:
Geometric Properties:
L = side of the plate = 10.00 in
d = diameter of the hole = 1.00 in
t = thickness of the plate = 0.25 in
Material Properties:
E = 3.0E7 psi
ν = 0.3
Loading:
P = 100 psi

- A coarse mesh is intentionally used to demonstrate the power of the p-method.
Chapter 2  Linear Static Analysis

Figure  S94-2 Meshed Quarter of the Plate.

S94A: TRIANG Elements
S94B: PLANE2D elements
S94C: TETRA10 Elements

Convergence Plots Using Different Element Types
COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
<th>Relative Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max. Stress in X-Direction (TRIANG)</td>
<td>300 psi</td>
<td>308 psi (p-order = 4)</td>
<td>2.7%</td>
</tr>
<tr>
<td>Max. Stress in X-Direction (PLANE2D)</td>
<td>300 psi</td>
<td>308 psi (p-order = 3)</td>
<td>2.7%</td>
</tr>
<tr>
<td>Max. Stress in X-Direction (TETRA10)</td>
<td>300 psi</td>
<td>323 psi (p-order = 5)</td>
<td>7.7%</td>
</tr>
</tbody>
</table>

Reference:
S95A, S95B, S95C: P-Method Solution of a U-Shaped Circumferential Groove in a Circular Shaft

**TYPE:**
Static analysis, axisymmetric triangular (6-node TRIANG) and quadrilateral (8-node PLANE2D) p-elements with the polynomial order of shape function equal to 8.

**PROBLEM:**
Calculate the maximum stress of a circular shaft with a U-shape circumferential groove under a uniformly distributed tension load. P-order is adapted by checking strain energy of the system.

**GIVEN:**
- **Geometric Properties:**
  - L = 0.9 in
  - D = 2 in
  - d = 0.2 in
- **Material Properties:**
  - E = 3.0E7 psi
  - ν = 0.3
- **Loading:**
  - P = 100 psi

![Figure S95-1: The Circular Shaft Model](image)
Part 2  Verification Problems

Figure S95-2: Finite Element Model with Different Element Types

S95A: TRIANG elements

S95B: PLANE2D elements

S95C: TETRA10 elements

Convergence Plots Using Different Element Types
## COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
<th>Relative Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max. Stress in Y-Direction (TRIANG)</td>
<td>305 psi</td>
<td>337 psi (p-order = 4)</td>
<td>10.5%</td>
</tr>
<tr>
<td>Max. Stress in Y-Direction (PLANE2D)</td>
<td>305 psi</td>
<td>333 psi (p-order = 8)</td>
<td>9.2%</td>
</tr>
<tr>
<td>Max. Stress in Y-Direction (TETRA10)</td>
<td>305 psi</td>
<td>339 psi (p-order = 5)</td>
<td>10.5%</td>
</tr>
</tbody>
</table>

## REFERENCE:

Modal (Frequency) Analysis

Introduction

This chapter contains verification problems to demonstrate the accuracy of the Modal Analysis module DSTAR.

<table>
<thead>
<tr>
<th>List of Natural Frequency Verification Problems</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>F1: Natural Frequencies of a Two-Mass Spring System</td>
<td>3-3</td>
</tr>
<tr>
<td>F2: Frequencies of a Cantilever Beam</td>
<td>3-4</td>
</tr>
<tr>
<td>F3: Frequency of a Simply Supported Beam</td>
<td>3-5</td>
</tr>
<tr>
<td>F4: Natural Frequencies of a Cantilever Beam</td>
<td>3-6</td>
</tr>
<tr>
<td>F5: Frequency of a Cantilever Beam with Lumped Mass</td>
<td>3-7</td>
</tr>
<tr>
<td>F6: Dynamic Analysis of a 3D Structure</td>
<td>3-8</td>
</tr>
<tr>
<td>F7A, F7B: Dynamic Analysis of a Simply Supported Plate</td>
<td>3-9</td>
</tr>
<tr>
<td>F8: Clamped Circular Plate</td>
<td>3-10</td>
</tr>
<tr>
<td>F9: Frequencies of a Cylindrical Shell</td>
<td>3-11</td>
</tr>
<tr>
<td>F10: Symmetric Modes and Natural Frequencies of a Ring</td>
<td>3-12</td>
</tr>
<tr>
<td>F11A, F11B: Eigenvalues of a Triangular Wing</td>
<td>3-13</td>
</tr>
<tr>
<td>F12: Vibration of an Unsupported Beam</td>
<td>3-14</td>
</tr>
</tbody>
</table>
### List of Natural Frequency Verification Problems (Concluded)

<table>
<thead>
<tr>
<th>Problem Number</th>
<th>Problem Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>F13</td>
<td>Frequencies of a Solid Cantilever Beam</td>
<td>3-15</td>
</tr>
<tr>
<td>F14</td>
<td>Natural Frequency of Fluid</td>
<td>3-16</td>
</tr>
<tr>
<td>F16A, F16B</td>
<td>Vibration of a Clamped Wedge</td>
<td>3-17</td>
</tr>
<tr>
<td>F17</td>
<td>Lateral Vibration of an Axially Loaded Bar</td>
<td>3-19</td>
</tr>
<tr>
<td>F18</td>
<td>Simply Supported Rectangular Plate</td>
<td>3-20</td>
</tr>
<tr>
<td>F19</td>
<td>Lowest Frequencies of Clamped Cylindrical Shell for Harmonic No. = 6</td>
<td>3-21</td>
</tr>
<tr>
<td>F21</td>
<td>Frequency Analysis of a Right Circular Canal of Fluid with Variable Depth</td>
<td>3-23</td>
</tr>
<tr>
<td>F22</td>
<td>Frequency Analysis of a Rectangular Tank of Fluid with Variable Depth</td>
<td>3-25</td>
</tr>
<tr>
<td>F23</td>
<td>Natural Frequency of Fluid in a Manometer</td>
<td>3-27</td>
</tr>
<tr>
<td>F24</td>
<td>Modal Analysis of a Piezoelectric Cantilever</td>
<td>3-29</td>
</tr>
<tr>
<td>F25</td>
<td>Frequency Analysis of a Stretched Circular Membrane</td>
<td>3-30</td>
</tr>
<tr>
<td>F26</td>
<td>Frequency Analysis of a Spherical Shell</td>
<td>3-31</td>
</tr>
<tr>
<td>F27A, F27B</td>
<td>Natural Frequencies of a Simply-Supported Square Plate</td>
<td>3-32</td>
</tr>
<tr>
<td>F28</td>
<td>Cylindrical Roof Shell</td>
<td>3-33</td>
</tr>
<tr>
<td>F29</td>
<td>Frequency Analysis of a Spinning Blade</td>
<td>3-34</td>
</tr>
</tbody>
</table>
F1: Natural Frequencies of a Two-Mass Spring System

TYPE:
Mode shape and frequency, truss and mass element (TRUSS3D, MASS).

REFERENCES:

PROBLEM:
Determine the normal modes and natural frequencies of the system shown below for the values of the masses and the springs given.

GIVEN:
\[ m_2 = 2m_1 = 1 \, \text{lb-sec}^2/\text{in} \]
\[ k_2 = k_1 = 200 \, \text{lb/in} \]
\[ k_c = 4k_1 = 800 \, \text{lb/in} \]

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>( F_1, \text{Hz} )</th>
<th>( F_2, \text{Hz} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>2.581</td>
<td>8.326</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>2.581</td>
<td>8.326</td>
</tr>
</tbody>
</table>

MODELING HINTS:
Truss elements with zero density are used as springs. Two dynamic degrees of freedom are selected at nodes 2 and 3 and masses are input as concentrated masses at nodes 2 and 3.

Figure F1-1
**F2: Frequencies of a Cantilever Beam**

**TYPE:**
Mode shape and frequency, plane element (PLANE2D).

**REFERENCE:**

**PROBLEM:**
Determine the fundamental frequency, f, of the cantilever beam of uniform cross section A.

**GIVEN:**
\[ E = 30 \times 10^6 \text{ psi} \]
\[ L = 50 \text{ in} \]
\[ h = 0.9 \text{ in} \]
\[ b = 0.9 \text{ in} \]
\[ A = 0.81 \text{ in}^2 \]
\[ v = 0 \]
\[ \rho = 0.734E-3 \text{ lb sec}^2/\text{in}^4 \]

**COMPARISON OF RESULTS**

<table>
<thead>
<tr>
<th></th>
<th>F1, Hz</th>
<th>F2, Hz</th>
<th>F3, Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>11.79</td>
<td>74.47</td>
<td>208.54</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>11.72</td>
<td>73.35</td>
<td>206.68</td>
</tr>
</tbody>
</table>

**Figure F2-1**

![Diagram of a cantilever beam with dimensions and problem sketch]
F3: Frequency of a Simply Supported Beam

**TYPE:**
Mode shapes and frequencies, beam element (BEAM3D).

**REFERENCE:**

**PROBLEM:**
Determine the fundamental frequency, \( f \), of the simply supported beam of uniform cross section \( A \).

**GIVEN:**
\[
\begin{align*}
E &= 30 \times 10^6 \text{ psi} \\
L &= 80 \text{ in} \\
\rho &= 0.7272 \times 10^{-3} \text{ lb-sec}^2/\text{in}^4 \\
A &= 4 \text{ in}^2 \\
I &= 1.3333 \text{ in}^4 \\
h &= 2 \text{ in}
\end{align*}
\]

**ANALYTICAL SOLUTION:**
\[
F_i = (i\pi^2)(EI/mL^4)^{1/2}
\]
\( i \) = Number of frequencies

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>( F_1 ), Hz</th>
<th>( F_2 ), Hz</th>
<th>( F_3 ), Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>28.78</td>
<td>115.12</td>
<td>259.0</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>28.78</td>
<td>114.31</td>
<td>242.7</td>
</tr>
</tbody>
</table>
Chapter 3  Modal (Frequency) Analysis

F4: Natural Frequencies of a Cantilever Beam

TYPE:
Mode shapes and frequencies, beam element (BEAM3D).

REFERENCE:

PROBLEM:
Determine the first three natural frequencies, \( f \), of a uniform beam clamped at one end and free at the other end.

GIVEN:
\[
\begin{align*}
E &= 30 \times 10^6 \text{ psi} \\
I &= 1.3333 \text{ in}^4 \\
A &= 4 \text{ in}^2 \\
h &= 2 \text{ in} \\
L &= 80 \text{ in} \\
\rho &= 0.72723 \times 10^{-3} \text{ lb sec}^2/\text{in}^4
\end{align*}
\]

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>( F_1 ), Hz</td>
<td>10.25</td>
</tr>
<tr>
<td>( F_2 ), Hz</td>
<td>64.25</td>
</tr>
<tr>
<td>( F_3 ), Hz</td>
<td>179.9</td>
</tr>
</tbody>
</table>
F5: Frequency of a Cantilever Beam with Lumped Mass

TYPE:
Mode shape and frequency, beam and mass elements (BEAM3D, MASS).

REFERENCE:

PROBLEM:
A steel cantilever beam of length 10 in has a square cross-section of 1/4 x 1/4 inch. A weight of 10 lbs is attached to the free end of the beam as shown in the figure. Determine the natural frequency of the system if the mass is displaced slightly and released.

GIVEN:
E  = 30 x 10^6 psi
W  = 10 lb
L  = 10 in

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th>F, Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
</tr>
<tr>
<td>COSMOS/M</td>
</tr>
</tbody>
</table>
TYPE:
Mode shapes and frequencies, pipe and mass elements (PIPE, MASS).

REFERENCE:

PROBLEM:
Find the natural frequencies and mode shapes of the 3D structure given below.

GIVEN:
Each member is a pipe.
Outer diameter = 2.375 in
Thickness = 0.154 in
E = 27.9 x 10^6 psi
ν = 0.3
The masses are represented solely by lumped masses as shown in the figure.

M_1 = M_2 = M_4 = M_6 = M_7 = M_8 = M_9 = M_11 = M_13 = M_14 = 0.00894223 lb sec^2/in
M_3 = M_5 = M_10 = M_12 = 0.0253816 lb sec^2/in

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>F_1, Hz</th>
<th>F_2, Hz</th>
<th>F_3, Hz</th>
<th>F_4, Hz</th>
<th>F_5, Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>111.5</td>
<td>115.9</td>
<td>137.6</td>
<td>218.0</td>
<td>404.2</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>111.2</td>
<td>115.8</td>
<td>137.1</td>
<td>215.7</td>
<td>404.2</td>
</tr>
</tbody>
</table>
**F7A, F7B: Dynamic Analysis of a Simply Supported Plate**

**TYPE:**
Mode shapes and frequencies, shell elements (SHELL4 and SHELL6).

**REFERENCE:**
Leissa, A.W. “Vibration of Plates,” NASA, sp-160, p. 44.

**PROBLEM:**
Obtain the first natural frequency for a simply supported plate.

**GIVEN:**
- $E = 30,000$ kips
- $v = 0.3$
- $h = 1$ in
- $a = b = 40$ in
- $\rho = 0.003$ kips sec$^2$/in$^4$

**NOTE:**
Due to double symmetry in geometry and the required mode shape, a quarter of the plate is taken for modeling.

**COMPARISON OF RESULTS**
The first natural frequency of the plate is 5.94 Hz.

<table>
<thead>
<tr>
<th></th>
<th>F7A: SHELL4</th>
<th>F7B: SHELL6 (Curved)</th>
<th>F7B: SHELL6 (Assembled)</th>
</tr>
</thead>
<tbody>
<tr>
<td>COSMOS/M</td>
<td>5.93 Hz</td>
<td>5.94 Hz</td>
<td>5.93</td>
</tr>
</tbody>
</table>
F8: Clamped Circular Plate

TYPE:
Mode shapes and frequencies, thick shell element (SHELL3T).

REFERENCE:

PROBLEM:
Obtain the first three natural frequencies.

GIVEN:
\[ E = 30 \times 10^6 \text{ psi} \]
\[ \nu = 0.3 \]
\[ \rho = 0.00073 \text{ (lb/in}^4 \text{) sec}^2 \]
\[ R = 40 \text{ in} \]
\[ t = 1 \text{ in} \]

NOTE:
Since a quarter of the plate is used for modeling, the second natural frequency is not symmetric \((s = 0, n = 1)\) and will not be calculated. This is an example to show that symmetry should be used carefully.

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th>Frequency No.</th>
<th>(s^*)</th>
<th>(n^*)</th>
<th>Theory (Hz)</th>
<th>COSMOS/M (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
<td>62.30</td>
<td>62.40</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
<td>2</td>
<td>212.60</td>
<td>212.53</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0</td>
<td>242.75</td>
<td>240.30</td>
</tr>
</tbody>
</table>

\(s^*\) refers to the number of nodal circles
\(n^*\) refers to the number of nodal diameters
Part 2  Verification Problems

F9: Frequencies of a Cylindrical Shell

TYPE:
Mode shapes and frequencies, shell element (SHELL4).

REFERENCE:

PROBLEM:
Determining the first three natural frequencies.

GIVEN:
E  = 30 x 10^6 psi
ν  = 0.3
ρ  = 0.00073 (lb-sec^2)/in^4
L  = 12 in
R  = 3 in
t  = 0.01 in

NOTE:
Due to symmetry in geometry and the mode shapes of the first three natural frequencies, 1/8 of the cylinder is considered for modeling.

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>F1, Hz</th>
<th>F2, Hz</th>
<th>F3, Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>552</td>
<td>736</td>
<td>783</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>553.69</td>
<td>718.50</td>
<td>795.60</td>
</tr>
</tbody>
</table>
Chapter 3  Modal (Frequency) Analysis

F10: Symmetric Modes and Natural Frequencies of a Ring

TYPE:
Mode shapes and frequencies, shell element (SHELL4).

REFERENCE:

PROBLEM:
Determine the first two natural frequencies of a uniform ring in symmetric case.

GIVEN:
E = 30E6 psi
v = 0
L = 4 in
h = 1 in
R = 1 in
ρ = 0.25E-2 (lb sec^2)/in^4

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>F1, Hz</th>
<th>F2, Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>135.05</td>
<td>735.14</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>134.92</td>
<td>723.94</td>
</tr>
</tbody>
</table>
F11A, F11B: Eigenvalues of a Triangular Wing

TYPE:
Mode shapes and frequencies, triangular shell elements (SHELL3 and SHELL6).

REFERENCE:

PROBLEM:
Calculate the natural frequencies of a triangular wing as shown in the figure.

GIVEN:
E = 6.5 x 10^6 psi
ν = 0.3541
ρ = 0.166E-3 lb sec^2/in^4
L = 6 in
Thickness = 0.034 in

COMPARISON OF RESULTS:
Natural Frequencies (Hz):

<table>
<thead>
<tr>
<th>Frequency No.</th>
<th>Reference</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>SHELL3</td>
</tr>
<tr>
<td>1</td>
<td>55.9</td>
<td>55.8</td>
</tr>
<tr>
<td>2</td>
<td>210.9</td>
<td>206.5</td>
</tr>
<tr>
<td>3</td>
<td>293.5</td>
<td>285.5</td>
</tr>
</tbody>
</table>
Chapter 3   Modal (Frequency) Analysis

F12: Vibration of an Unsupported Beam

TYPE:
Mode shapes and frequencies, rigid body modes, beam element (BEAM3D).

REFERENCE:

PROBLEM:
Determine the elastic and rigid body modes of vibration of the unsupported beam shown below.

GIVEN:
L = 100 in
E = 1 x 10^8 psi
r = 0.1 in
ρ = 0.2588E-3 lb sec^2/in^4

ANALYTICAL SOLUTION:
The theoretical solution is given by the roots of the equation Cos KL Cosh KL = 1 and the frequencies are given by:

f_i = K_i^2 (EI/ρA)^(1/2)/(2π)  \quad A = area of cross-section
i = Number of natural frequencies  \quad ρ = Mass Density
K_i = (i + 0.5)π/L

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th>Mode</th>
<th>Theory F, Hz</th>
<th>Theory (ki)</th>
<th>COSMOS/M F, Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Mode 1</td>
<td>Mode 2</td>
<td>Mode 3</td>
</tr>
<tr>
<td>Theory F, Hz</td>
<td>0</td>
<td>0</td>
<td>11.07</td>
</tr>
<tr>
<td>Theory (ki)</td>
<td>(0)</td>
<td>(0)</td>
<td>(4.73)</td>
</tr>
<tr>
<td>COSMOS/M F, Hz</td>
<td>0</td>
<td>0</td>
<td>10.92</td>
</tr>
</tbody>
</table>

NOTE:
First two modes are rigid body modes.
F13: Frequencies of a Solid Cantilever Beam

**TYPE:**
Mode shapes and frequencies, hexahedral solid element (SOLID).

**REFERENCE:**

**PROBLEM:**
Determine the first three natural frequencies of a uniform beam clamped at one end and free at the other end.

**GIVEN:**
- $E = 30 \times 10^6$ psi
- $a = 2$ in
- $b = 2$ in
- $L = 80$ in
- $\rho = 0.00072723$ lb-sec$^2$/in$^4$

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>$F_1$, Hz</th>
<th>$F_2$, Hz</th>
<th>$F_3$, Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>10.25</td>
<td>64.25</td>
<td>179.91</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>10.24</td>
<td>63.95</td>
<td>178.38</td>
</tr>
</tbody>
</table>
Chapter 3  Modal (Frequency) Analysis

F14: Natural Frequency of Fluid

TYPE:
Mode shapes and frequencies, truss elements (TRUSS2D).

REFERENCE:

PROBLEM:
A manometer used in a fluid mechanics laboratory has a uniform bore of cross-section area \( A \). If a column of liquid of length \( L \) and weight density \( \rho \) is set into motion, as shown in the figure, find the frequency of the resulting motion.

GIVEN:
\[
\begin{align*}
A &= 1 \text{ in}^2 \\
\rho &= 9.614 \times 10^{-5} \text{ lb sec}^2/\text{in}^4 \\
L &= 51.4159 \text{ in} \\
E &= 1 \times 10^5 \text{ psi}
\end{align*}
\]

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>( F, \text{ Hz} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>0.617</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>0.617</td>
</tr>
</tbody>
</table>

NOTE:
The mass of fluid is lumped at nodes 2 to 28. The boundary elements are applied at nodes 6 to 24.

Figure F14-1
F16A, F16B: Vibration of a Clamped Wedge

**TYPE:**
Mode shapes and frequencies, thick shell elements (SHELL3T, SHELL4T).

**REFERENCE:**

**PROBLEM:**
Determine the fundamental frequency of lateral vibration of a wedge shaped plate. The plate is of uniform thickness \( t \), base \( 3b \), and length \( L \).

**GIVEN:**
- \( E = 30 \times 10^6 \) psi
- \( \rho = 7.28 \times 10^{-4} \) lb sec\(^2\)/in\(^4\)
- \( t = 1 \) in
- \( b = 2 \) in
- \( L = 16 \) in

**MODELING HINTS:**
Only in-plane (in x-y plane) frequencies along y-direction are considered. In order to find better results, out-of-plane displacements (z-direction) are restricted. The effect of different elements and meshes is also considered.

**ANALYTICAL SOLUTION:**
The first in-plane natural frequency calculated by:

\[
f_1 = \frac{5.315b}{2 \pi L^2} \sqrt{\frac{E}{3\rho}}
\]

Using approximate RITZ method, for first and second natural frequencies:

\[
f_1 = \frac{5.319b}{2 \pi L^2} \sqrt{\frac{E}{3\rho}} \quad f_2 = \frac{17.301b}{2 \pi L^2} \sqrt{\frac{E}{3\rho}}
\]
COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>Natural Frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>First</td>
</tr>
<tr>
<td>Reference</td>
<td></td>
</tr>
<tr>
<td>Exact</td>
<td>774.547</td>
</tr>
<tr>
<td>Ritz</td>
<td>775.130</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td></td>
</tr>
<tr>
<td>SHELL3T (F16A)</td>
<td>813.45</td>
</tr>
<tr>
<td>SHELL4T (F16B)</td>
<td>789.12</td>
</tr>
</tbody>
</table>

Figure F16A-1
Part 2  Verification Problems

F17: Lateral Vibration of an Axially Loaded Bar

TYPE:
Mode shapes and frequencies, in-plane effects, beam elements (BEAM3D).

REFERENCE:

PROBLEM:
Determine the fundamental frequency of lateral vibration of a wedge shaped plate. The plate is of uniform thickness t, base 3b, and length L.

GIVEN:
\[ E = 30 \times 10^6 \text{ psi} \]
\[ \rho = 7.2792 \times 10^{-4} \text{ lb sec}^2/\text{in}^4 \]
\[ g = 386 \text{ in/sec}^2 \]
\[ b = h = 2 \text{ in} \]
\[ L = 80 \text{ in} \]
\[ P = 40,000 \text{ lb} \]

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>( F_1, \text{ Hz} )</th>
<th>( F_2, \text{ Hz} )</th>
<th>( F_3, \text{ Hz} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>17.055</td>
<td>105.32</td>
<td>249.39</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>17.055</td>
<td>105.32</td>
<td>249.34</td>
</tr>
</tbody>
</table>

Figure F17-1

Problem Sketch

Finite Element Model
**F18: Simply Supported Rectangular Plate**

**TYPE:**
Mode shapes and frequencies, in-plane effects, shell element (SHELL4).

**REFERENCE:**

**PROBLEM:**
Obtain the fundamental frequency of a simply supported plate with the effect of in-plane forces. \( N_x = 33.89 \text{ lb/in} \) applied at \( x = 0 \) and \( x = a \).

**GIVEN:**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( E )</td>
<td>30,000 psi</td>
</tr>
<tr>
<td>( \nu )</td>
<td>0.3</td>
</tr>
<tr>
<td>( h )</td>
<td>1 in</td>
</tr>
<tr>
<td>( a = b = 40 \text{ in} )</td>
<td></td>
</tr>
<tr>
<td>( \rho )</td>
<td>0.0003 ( \text{lb sec}^2/\text{in}^4 )</td>
</tr>
<tr>
<td>( P )</td>
<td>33.89 psi</td>
</tr>
</tbody>
</table>

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>( F, \text{ Hz} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>4.20</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>4.19</td>
</tr>
</tbody>
</table>

**NOTE:**
Due to double symmetry in geometry, loads and the mode shape, a quarter plate is taken for modeling.

*Figure F18-1*
**F19: Lowest Frequencies of Clamped Cylindrical Shell for Harmonic No. = 6**

**TYPE:**
Mode shapes and frequencies, axisymmetric shell elements (SHELLAX).

**REFERENCE:**

**PROBLEM:**
To find the lowest natural frequency of vibration for the cylinder fixed at both ends.

**GIVEN:**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>R</td>
<td>3 in</td>
</tr>
<tr>
<td>L</td>
<td>12 in</td>
</tr>
<tr>
<td>t</td>
<td>0.01 in</td>
</tr>
<tr>
<td>E</td>
<td>$30 \times 10^6$ psi</td>
</tr>
<tr>
<td>$\nu$</td>
<td>0.35</td>
</tr>
<tr>
<td>$\rho$</td>
<td>0.000730 lb sec$^2$/in$^4$</td>
</tr>
</tbody>
</table>

Range of circumferential harmonics ($n$) = 4 to 7.

**MODELING HINTS:**
All the 21 nodes are spaced equally along the meridian of cylinder. The number of circumferential harmonics (lobes) for each frequency analysis is to be specified and lowest frequency is sought.

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Harmonic No. ($n$)</th>
<th>First Frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Theory</td>
</tr>
<tr>
<td>(4) *</td>
<td>926</td>
</tr>
<tr>
<td>(5) *</td>
<td>646</td>
</tr>
<tr>
<td>(6) *</td>
<td>563</td>
</tr>
<tr>
<td>(7) *</td>
<td>606</td>
</tr>
</tbody>
</table>

* You need to re-execute the analysis by specifying these harmonic numbers under the A_FREQUENCY command. The lowest natural frequency is 549.6 Hz corresponding to harmonic number = 6.

TYPE:
Mode shapes and frequencies, multifield elements, 4- and 8-node PLANE2D, SHELL4T, 6-node TRIANG, TETRA10, 8- and 20-node SOLID, TETRA4R, and SHELL6.

PROBLEM:
Compare the first two natural frequencies of a cantilever beam modeled by each of the above element types.

GIVEN:
\[ E = 10^7 \text{ psi} \]
\[ \rho = 245 \times 10^{-3} \text{ lb-sec}^2/\text{in}^4 \]
\[ b = 0.1 \text{ in} \]
\[ h = 0.2 \text{ in} \]
\[ L = 6 \text{ in} \]
\[ n = 0.3 \]

COMPARISON OF RESULTS:
The theoretical solutions for the first and second mode are: 181.17 and 1136.29 Hz.

<table>
<thead>
<tr>
<th>Input File</th>
<th>Element</th>
<th>1st Mode</th>
<th>Error (%)</th>
<th>2nd Mode</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>F20A</td>
<td>PLANE2D 4-node</td>
<td>180.71</td>
<td>0.2</td>
<td>1127.96</td>
<td>0.7</td>
</tr>
<tr>
<td>F20B</td>
<td>PLANE2D 8-node</td>
<td>181.15</td>
<td>0.0</td>
<td>1153.52</td>
<td>1.53</td>
</tr>
<tr>
<td>F20C</td>
<td>TRIANG 6-node</td>
<td>183.35</td>
<td>1.2</td>
<td>1182.90</td>
<td>4.1</td>
</tr>
<tr>
<td>F20D</td>
<td>TETRA10</td>
<td>183.10</td>
<td>1.0</td>
<td>1184.85</td>
<td>4.3</td>
</tr>
<tr>
<td>F20E</td>
<td>SOLID 8-node</td>
<td>181.64</td>
<td>0.2</td>
<td>1134.67</td>
<td>0.2</td>
</tr>
<tr>
<td>F20F</td>
<td>SOLID 20-node</td>
<td>179.72</td>
<td>0.8</td>
<td>1111.16</td>
<td>2.2</td>
</tr>
<tr>
<td>F20G</td>
<td>TETRA4R</td>
<td>190.24</td>
<td>5.1</td>
<td>1182.72</td>
<td>4.1</td>
</tr>
<tr>
<td>F20H</td>
<td>SHELL6 (Curved)</td>
<td>183.371</td>
<td>1.2</td>
<td>1182.87</td>
<td>4.1</td>
</tr>
<tr>
<td></td>
<td>SHELL6 (Assembled)</td>
<td>183.357</td>
<td>1.2</td>
<td>1182.54</td>
<td>4.1</td>
</tr>
</tbody>
</table>
**F21: Frequency Analysis of a Right Circular Canal of Fluid with Variable Depth**

**TYPE:**
Mode shapes and frequencies, fluid sloshing, plane strain elements (PLANE2D).

**REFERENCE:**

**PROBLEM:**
A right circular canal with radius R is half-filled by an incompressible liquid (see Figure F21-1). Determine the first two natural frequencies with mode shapes antisymmetric about the Y-axis.

**GIVEN**
- \( R = 56.4 \text{ in} \)
- \( H/R = 0 \)
- \( \rho = 0.9345E-4 \text{ lb sec}^2/\text{in}^4 \)
- \( EX = 3E5 \text{ lb/in}^2 \)

*Where:*
- EX = bulk modulus

**NOTES:**
1. A small shear modulus \( S_{XY} = EX \) is used to prevent numerical instability.
2. The radial component of displacements (in local cylindrical coordinate system) is constrained at the curved boundary in order to allow sloshing.
3. The acceleration due to gravity (ACEL command) in the negative direction.
4. PLANE2D plane strain elements are used to solve the current problem since the mode shapes are independent of Z-direction coordinates.
5. For non rectangular geometries, one can expect to obtain some natural frequencies with no significant changes in the free surface profile. This situation is analogous to the rigid modes of a solid structure. Therefore, a negative shift of \( \omega \) is recommended to prevent this type of sloshing modes.
Chapter 3  Modal (Frequency) Analysis

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>Analytical Solution (Hz)</th>
<th>COSMOS/M (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.4858</td>
<td>0.4875</td>
</tr>
<tr>
<td>2</td>
<td>Not Available</td>
<td>0.7269</td>
</tr>
<tr>
<td>3</td>
<td>0.9031</td>
<td>0.8976</td>
</tr>
</tbody>
</table>

Figure F21-1

Constrain only the radial component of displacement to allow sloshing.

Finite Element Model
F22: Frequency Analysis of a Rectangular Tank of Fluid with Variable Depth

**TYPE:**
Mode shapes and frequencies, fluid sloshing, hexahedral solid (SOLID).

**REFERENCE:**

**PROBLEM:**
A rectangular tank with dimensions A and B in X- and Z-directions is partially filled by an incompressible liquid (see Figure F22-1). Determine the first two natural frequencies.

**GIVEN:**
\[
\begin{align*}
A &= 48 \text{ in} \\
B &= 48 \text{ in} \\
H &= 20 \text{ in} \\
\rho &= 0.9345 \times 10^{-4} \text{ lb sec}^2/\text{in}^4 \\
EX &= 3E5 \text{ lb/in}^2 \\
\end{align*}
\]

Where:
\(EX\) = bulk modulus

**NOTE:**
Please refer to notes (1), (2), (3), (4) and (5) in Problem F21.

**COMPARISON OF RESULTS:**
The analytical solution for natural frequencies is as follows:

\[
b_{ij} = \frac{1}{2} \sqrt{\frac{g}{\pi}} \left[ \frac{i^2}{a^2} + \frac{j^2}{b^2} \right] \tan \pi h \left( \frac{i^2}{a^2} + \frac{j^2}{b^2} \right)^{1/2} \text{ Hz}
\]

where \(i\) and \(j\) represent the order number in X- and Z-directions respectively.
The comparison of analytical solutions with those obtained using COSMOS/M for various values of \( i \) and \( j \) are tabulated below.

<table>
<thead>
<tr>
<th>Frequency Number ( i / j )</th>
<th>Analytical Solution (Hz)</th>
<th>COSMOS/M (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 / 0</td>
<td>—</td>
<td>—</td>
</tr>
<tr>
<td>0 / 1</td>
<td>0.7440</td>
<td>0.7422</td>
</tr>
<tr>
<td>1 / 2</td>
<td>0.9286</td>
<td>0.9199</td>
</tr>
</tbody>
</table>

Figure F22-1

Constrain the normal component of displacement to allow sloshing.

Finite Element Model
**F23: Natural Frequency of Fluid in a Manometer**

**TYPE:**
Mode shapes and frequencies, fluid sloshing, plane strain elements (PLANE2D).

**REFERENCE:**

**PROBLEM:**
A manometer used in a fluid mechanics laboratory has a uniform bore of cross-sectional area A. If a column of liquid of length L and weight density r is set into motion as shown in the figures, find the frequency of the resulting motion.

**GIVEN:**
- \( A = 0.5 \text{ in}^2 \)
- \( \rho = 0.9345\times10^{-4} \text{ lb sec}^2/\text{in}^4 \)
- \( L = 26.4934 \text{ in} \) (length of fluid in the manometer)
- \( EX = 3\times10^5 \text{ lb/in}^2 \)

*Where:*
- \( EX \) = bulk modulus

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>( F, \text{ Hz} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical Solution *</td>
<td>0.8596</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>0.8623</td>
</tr>
</tbody>
</table>

**NOTE:**
A small shear modulus \( GXY = EX(1.0E-9) \) is used to prevent numerical instability. Global and local constraints are applied normal to the boundary to prevent leaking of the fluid. Acceleration due to gravity (ACEL command) in the negative y-direction should be included for problems with free surfaces.
Figure F23-1

Problem Sketch

Finite Element Model

Constrain displacement components normal to the surface to allow fluid sloshing.
F24: Modal Analysis of a Piezoelectric Cantilever

TYPE:
Mode shapes and frequencies using solid piezoelectric element (SOLIDPZ).

REFERENCE:

PROBLEM:
A piezoelectric transducer with a polarization direction along its longitudinal direction has electrodes at two ends. Both electrodes are grounded to represent a short-circuit condition. All non-prescribed voltage D.O.F.’s are condensed out after assemblage of stiffness matrix. In this problem, the longitudinal mode of vibration is under consideration.

GIVEN:
L = 80 mm
b = h = 2 mm
Density = 727 Kg/m³

NOTE:
To constrain voltage degrees of freedom for piezoelectric application, use the RX component of displacement in the applicable constraint commands (DND, DCR, DSF, etc.). There is no rotational degree of freedom for SOLID elements in COSMOS/M.

COMPARISON OF RESULTS:
For the sixth mode of vibration in this problem (longitudinal mode):

<table>
<thead>
<tr>
<th>Theory</th>
<th>690 Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>COSMOS/M</td>
<td>685 Hz</td>
</tr>
</tbody>
</table>
Chapter 3  Modal (Frequency) Analysis

F25: Frequency Analysis of a Stretched Circular Membrane

TYPE:
Frequency analysis using the nonaxisymmetric mode shape option (SHELLAX).

REFERENCE:

PROBLEM:
Find the first three frequencies of a stretched circular membrane.

GIVEN:

<table>
<thead>
<tr>
<th>Natural Frequency No.</th>
<th>Theory (Hz)</th>
<th>COSMOS/M (Hz)</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>94.406</td>
<td>93.73</td>
<td>0.72</td>
</tr>
<tr>
<td>2</td>
<td>216.77</td>
<td>212.95</td>
<td>1.76</td>
</tr>
<tr>
<td>3</td>
<td>339.85</td>
<td>329.76</td>
<td>2.97</td>
</tr>
</tbody>
</table>

MODELING HINTS:
A total of 9 elements are considered as shown. The stretching load of 1500 lb for a one radian section of the shell is applied with the inplane loading flag turned on for frequency calculations. All frequencies are found for circumferential harmonic number 0.

Figure F25-1
F26: Frequency Analysis of a Spherical Shell

**TYPE:**
Frequency analysis using the nonaxisymmetric mode shape option (SHELLAX).

**REFERENCE:**

**PROBLEM:**
Find the first eight frequencies of the spherical shell shown here for the circumferential harmonic number 2.

**GIVEN:**
- \( R = 10 \text{ in} \)
- \( E = 1 \times 10^7 \text{ psi} \)
- \( \rho = 0.0005208 \text{ lb-sec}^2/\text{in}^4 \)
- \( v \text{ (NUXY)} = 0.3 \)
- \( t \text{ (Thickness)} = 0.1 \text{ in} \)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th>Natural Frequency No.</th>
<th>Theory (Hz)</th>
<th>COSMOS/M (Hz)</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1620</td>
<td>1622</td>
<td>0.12</td>
</tr>
<tr>
<td>2</td>
<td>1919</td>
<td>1923</td>
<td>0.20</td>
</tr>
<tr>
<td>3</td>
<td>2035</td>
<td>2044</td>
<td>0.44</td>
</tr>
<tr>
<td>4</td>
<td>2093</td>
<td>2110</td>
<td>0.81</td>
</tr>
<tr>
<td>5</td>
<td>2125</td>
<td>2153</td>
<td>1.32</td>
</tr>
<tr>
<td>6</td>
<td>2145</td>
<td>2188</td>
<td>2.00</td>
</tr>
<tr>
<td>7</td>
<td>2159</td>
<td>2224</td>
<td>3.01</td>
</tr>
<tr>
<td>8</td>
<td>2168</td>
<td>2262</td>
<td>4.34</td>
</tr>
</tbody>
</table>

Finite Element Model

![Problem Sketch](image)

![Finite Element Model](image)
F27A, F27B: Natural Frequencies of a Simply-Supported Square Plate

**TYPE:**
Frequency analysis, Guyan reduction, SHELL4 elements.

- Case A: Guyan Reduction
- Case B: Consistent Mass

**PROBLEM:**
Natural frequencies of a simply-supported plate are calculated. Utilizing the symmetry of the model, only one quarter of the plate is modeled and the first three symmetric modes of vibration are calculated. The mass is lumped uniformly at master degrees of freedom.

**GIVEN:**
- \( L = 30 \text{ in} \)
- \( h = 0.1 \text{ in} \)
- \( \rho = 8.29 \times 10^{-4} \text{ (lb sec}^2)/\text{in}^4 \)
- \( v = 0.3 \)
- \( E = 30 \times 10^6 \text{ psi} \)

**ANALYTICAL SOLUTION:**
Theoretical results can be obtained from the equation:

\[
\omega_{mn} = \frac{r^2 D}{L^2 U} \cdot (m^2 + n^2)
\]

*Where:*
- \( D = \frac{E h^3}{12(1 - v^2)} \)
- \( U = \rho h \)

**COMPARISON OF RESULTS:**
Normalized mode shape displacements for the nodes connected by the rigid bar.

<table>
<thead>
<tr>
<th>Natural Frequency (Hz)</th>
<th>First</th>
<th>Second</th>
<th>Third</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Theory</strong></td>
<td>5.02</td>
<td>25.12</td>
<td>25.12</td>
</tr>
<tr>
<td><strong>Case A: Guyan Reduction</strong></td>
<td>5.03</td>
<td>25.15</td>
<td>25.20</td>
</tr>
<tr>
<td><strong>Case B: Consistent Mass</strong></td>
<td>5.02</td>
<td>25.11</td>
<td>25.11</td>
</tr>
</tbody>
</table>

Total Mass = \( \rho \cdot v = 8.29 \times 10^{-4} \times 0.1 \times 30 \times 30 = 0.07461 \)
Lumped Mass at Master Nodes = \( 0.07461/64 = 0.00116 \times 3 \)

**Figure F27-1**

```
\[\text{Simply Supported Plate}\]
```

```
\[\text{Problem Sketch}\]
```

\[\text{Problem Sketch}\]

\[\text{Problem Sketch}\]
F28: Cylindrical Roof Shell

**TYPE:**
Natural mode shape and frequency, shell and rigid bar elements.

**PROBLEM:**
Determine the first frequency and mode shape of the shell roof shown below.

**GIVEN:**
\[ r = 25 \text{ ft} \]
\[ E = 4.32 \times 10^{12}, \ 4.32 \times 10^{11}, \text{ and } 4.32 \times 10^{10} \text{ psi} \]
\[ v = 0 \]

**MODELING HINTS:**
Due to symmetry, a quarter of the shell roof is considered in the modeling. Nodes 8 and 12 are connected by a rigid bar.

**COMPARISON OF RESULTS:**
Normalized mode shape displacements for the nodes connected by the rigid bar.

<table>
<thead>
<tr>
<th>Method</th>
<th>Young's Modulus</th>
<th>Z-Rotation</th>
<th>R8 / R12</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory COSMOS/M</td>
<td>4.32E12</td>
<td>-0.5642901E-2</td>
<td>1.000</td>
</tr>
<tr>
<td>Theory COSMOS/M</td>
<td>4.32E11</td>
<td>-0.5654460E-2</td>
<td>1.000</td>
</tr>
<tr>
<td>Theory COSMOS/M</td>
<td>4.32E10</td>
<td>-0.5693621E-2</td>
<td>1.000</td>
</tr>
</tbody>
</table>
F29A, B, C: Frequency Analysis of a Spinning Blade

TYPE:
Frequency analysis using the Spin Softening and Stress Stiffening Options.

REFERENCE:

PROBLEM:
Find the fundamental frequency of vibration of a blade cantilevered from a rigid spinning rod.

MODELING HINTS:
The blade is cantilevered to a rigid rod. Therefore, the blade may be modeled with a fixed displacement boundary condition at the connection to the rod. The Stress Stiffening effect due to centrifugal load is considered in this model by activating the centrifugal force option in A_STATIC command together with the Inplane Loading Flag in A_FREQUENCY command.
Part 2  Verification Problems

GIVEN:
\[ R = 150 \text{ mm} \quad E = 217 \times 10^9 \text{ Pa} \]
\[ h = 328 \text{ mm} \quad \rho = 7850 \text{ Kg/m}^3 \]
\[ b = 28 \text{ mm} \quad \gamma = 0.3 \]
\[ t = 3 \text{ mm} \quad \omega = 314.159 \text{ rad/sec} \]

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>Fundamental Frequency (Hz)</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>52.75</td>
<td></td>
</tr>
<tr>
<td>A</td>
<td>Stress stiffening with spin softening</td>
<td>51.17</td>
</tr>
<tr>
<td>B</td>
<td>Stress stiffening with no spin softening</td>
<td>71.54</td>
</tr>
<tr>
<td>C</td>
<td>No stress stiffening and no spin softening</td>
<td>23.80</td>
</tr>
</tbody>
</table>
4

**Buckling Analysis**

*Introduction*

This chapter contains verification problems to demonstrate the accuracy of the Buckling Analysis module DSTAR.

<table>
<thead>
<tr>
<th>List of Buckling Verification Problems</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>B1: Instability of Columns</td>
<td>4-2</td>
</tr>
<tr>
<td>B2: Instability of Columns</td>
<td>4-3</td>
</tr>
<tr>
<td>B3: Instability of Columns</td>
<td>4-4</td>
</tr>
<tr>
<td>B4: Simply Supported Rectangular Plate</td>
<td>4-5</td>
</tr>
<tr>
<td>B5A, B5B: Instability of a Ring</td>
<td>4-6</td>
</tr>
<tr>
<td>B6: Buckling Analysis of a Small Frame</td>
<td>4-7</td>
</tr>
<tr>
<td>B7A, B7B: Instability of Frames</td>
<td>4-9</td>
</tr>
<tr>
<td>B8: Instability of a Cylinder</td>
<td>4-10</td>
</tr>
<tr>
<td>B9: Simply Supported Stiffened Plate</td>
<td>4-11</td>
</tr>
<tr>
<td>B10: Stability of a Rectangular Frame</td>
<td>4-12</td>
</tr>
<tr>
<td>B11: Buckling of a Stepped Column</td>
<td>4-13</td>
</tr>
<tr>
<td>B12: Buckling Analysis of a Simply Supported Composite Plate</td>
<td>4-14</td>
</tr>
<tr>
<td>B13: Buckling of a Tapered Column</td>
<td>4-15</td>
</tr>
<tr>
<td>B14: Buckling of Clamped Cylindrical Shell Under External Pressure Using the Nonaxisymmetric Buckling Mode Option</td>
<td>4-16</td>
</tr>
<tr>
<td>B15A, B15B: Buckling of Simply-Supported Cylindrical Shell Under Axial Load</td>
<td>4-18</td>
</tr>
</tbody>
</table>
Chapter 4  Buckling Analysis

B1: Instability of Columns

TYPE:
Buckling analysis, beam element (BEAM3D).

REFERENCE:

PROBLEM:
Find the buckling load and deflection mode for a simply supported column.

GIVEN:
E = 30 x 10^6 psi
h = 1 in
L = 50 in
I = 1/12 in^4

ANALYTICAL SOLUTION:
\[ P_{cr} = \frac{\pi^2 EI}{L^2} = 9869.6 \text{ lb} \]

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>( P_{cr} )</td>
<td>9869.6 lb</td>
<td>9869.6 lb</td>
</tr>
</tbody>
</table>

Figure B1-1
B2: Instability of Columns

**TYPE:**
Buckling analysis, beam element (BEAM3D).

**REFERENCE:**

**PROBLEM:**
Find the buckling load and deflection mode for a clamped-clamped column.

**GIVEN:**
\[ E = 30 \times 10^6 \text{ psi} \]
\[ h = 1 \text{ in} \]
\[ L = 50 \text{ in} \]
\[ I = 1/12 \text{ in}^4 \]

**ANALYTICAL SOLUTION:**
\[ P_{cr} = 4\pi^2 \frac{EI}{L^2} = 39478.4 \text{ lb} \]

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>[ P_{cr} ]</td>
<td>39478.4 lb</td>
<td>39478.8 lb</td>
</tr>
</tbody>
</table>

**Problem Sketch**

![Problem Sketch](image)

**Finite Element Model**

![Finite Element Model](image)
B3: Instability of Columns

**TYPE:**
Buckling analysis, beam element (BEAM3D).

**REFERENCE:**

**PROBLEM:**
Find the buckling load and deflection mode for a clamped-free column.

**GIVEN:**
- $E = 30 \times 10^6$ psi
- $h = 1$ in
- $L = 50$ in
- $I = \frac{1}{12}$ in$^4$

**ANALYTICAL SOLUTION:**
$$ p_{cr} = \frac{\pi^2 EI}{(4L^2)} = 2467.4 \text{ lb}$$

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>$p_{cr}$</td>
<td>2467.4 lb</td>
<td>2467.4 lb</td>
</tr>
</tbody>
</table>

**Figure B3-1**

![Problem Sketch and Finite Element Model](image_url)
**B4: Simply Supported Rectangular Plate**

**TYPE:**
Buckling analysis, shell element (SHELL4).

**REFERENCE:**

**PROBLEM:**
Find the buckling load of a simply supported isotropic plate subjected to inplane uniform load \( p \) applied at \( x = 0 \) and \( x = a \).

**GIVEN:**
\[ E = 30,000 \text{ psi} \]
\[ v = 0.3 \]
\[ h = 1 \text{ in} \]
\[ a = b = 40 \text{ in} \]
\[ p = 1 \text{ lb/in} \]

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>( P_{cr} )</td>
<td>67.78 lb</td>
<td>67.85 lb</td>
</tr>
</tbody>
</table>

**NOTE:**
Due to double symmetry in geometry and loads, a quarter of the plate is taken for modeling.

---

Problem Sketch and Finite Element Model
B5A, B5B: Instability of a Ring

**TYPE:**
Buckling analysis, shell element (SHELL3, SHELL6).

**REFERENCE:**

**PROBLEM:**
Find the buckling load and deflection mode of a ring under pressure loading.

**GIVEN:**
- \( E = 10 \times 10^6 \) psi
- \( R = 5 \) in
- \( h = 0.1 \) in
- \( b = 1 \) in
- \( I = 0.001/12 \) in\(^4\)

**ANALYTICAL SOLUTION:**
Using Donnell Approximations.

\[
P_{cr} = \frac{4EI}{R^3} = 26.667 \text{ lb/in}
\]

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M SHELL3</th>
<th>COSMOS/M SHELL6 (Curved)</th>
<th>COSMOS/M SHELL6 (Assembled)</th>
</tr>
</thead>
</table>

**Problem Sketch:**

- Problem Sketch
- Finite Element Model

**Figure B5-1**
**B6: Buckling Analysis of a Small Frame**

**TYPE:**
Buckling analysis, truss (TRUSS2D) and beam (BEAM3D) elements.

**REFERENCE:**

**GIVEN:**
- \( L = 20 \text{ in} \)
- \( A_B = 4 \text{ in}^2 \)
- \( A_T = 0.1 \text{ in}^2 \)
- \( E = E_B = E_T = 30 \text{E6 psi} \)
- \( I_B = 2 \text{ in}^4 \)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>( P_{1cr} )</td>
<td>1051.392 lb</td>
<td>1051.367 lb</td>
</tr>
<tr>
<td>( P_{2cr} )</td>
<td>1480.44 lb</td>
<td>1481.20 lb</td>
</tr>
</tbody>
</table>

**ANALYTICAL SOLUTION:**
The classical results are obtained from:

\[
P_{1cr} = A_T E \sin \alpha \cos^2 \alpha / (1 + (A_T/A_B) \sin^3 \alpha) \]
\[
P_{2cr} = \pi^2 E I_B / L^2 \]

**MODE SHAPES:**

Figure B6-1

Mode Shape 1  
Mode Shape 2
Chapter 4  Buckling Analysis

Figure B6-2

Problem Sketch and Finite Element Model
B7A, B7B: Instability of Frames

TYPE:
Buckling analysis, shell element (SHELL4 and SHELL6).

REFERENCE:

PROBLEM:
Find the buckling load and deflection mode for the frame shown below.

GIVEN:
- E = 30 x 10^6 psi
- h = 1 in
- L = 25 in
- I = 1/12 in^4

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SHELL4</td>
</tr>
<tr>
<td>P_{cr}</td>
<td>55506.6 lb</td>
</tr>
</tbody>
</table>

ANALYTICAL SOLUTION:

P_{cr} = 1.406\pi^2EI / L^2 = 55506.6 lb

Figure B7-1
B8: Instability of a Cylinder

TYPE:
Buckling analysis, axisymmetric shell element (SHELLAX).

REFERENCE:

PROBLEM:
Find the buckling load and deflection mode for a cylindrical shell that is simply supported at its ends and subjected to uniform lateral pressure.

GIVEN:
\[ E = 10 \times 10^6 \text{ psi} \]
\[ h = 0.2 \text{ in} \]
\[ R = 20 \text{ in} \]
\[ L = 20 \text{ in} \]
\[ \nu = 0.3 \]

ANALYTICAL SOLUTION:

\[
P_{cr} = \frac{Eh}{R} \left\{ \left( \frac{\pi R/L}{n} \right)^2 + n^2 \right\}^{1/2} \left( \frac{h/R}{1 - \nu^2} \right)^2 + \frac{\left( \frac{R/L}{n} \right)^4}{n^2 + \left( \frac{R/L}{n} \right)^2 + \frac{n^3}{L}} \right\}
\]

COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>( P_{cr} )</td>
<td>106 psi</td>
</tr>
</tbody>
</table>
B9: Simply Supported Stiffened Plate

**TYPE:**
Buckling analysis, shell (SHELL4) and beam (BEAM3D) elements.

**REFERENCE:**

**PROBLEM:**
A simply supported rectangular plate is stiffened by a beam of rectangular cross-section as shown in the figure. The stiffened plate is subjected to inplane pressure at edges $x = 0$ and $x = a$. Determine the buckling pressure load.

**GIVEN:**
- $E = 30,000$ kip/in$^2$
- $v = 0$
- $h_p = 1$ in
- $a = 45.5$ in
- $b = 42$ in
- $b_b = 0.42$ in
- $h_b = 10$ in

**ANALYTICAL SOLUTION:**
\[
\sigma_{cr} = \frac{\pi^2 D}{b h} \times \frac{(1 + \beta^2) + 2\gamma}{\beta^2 (1 + 2\delta)}
\]
Where:
- $\beta = a/b$
- $\gamma = E I_b / b D$
- $D = E (h_p)^3 / 12(1-v^2)$
- $A_b = b_b x h_b$
- $\delta = A_b / bh_p$

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_{cr}$</td>
<td>223.80 kip/in</td>
<td>232.53 kip/in</td>
<td>3.9%</td>
</tr>
</tbody>
</table>

Figure B9-1

Problem Sketch and Finite Element Model
**Chapter 4  Buckling Analysis**

**B10: Stability of a Rectangular Frame**

**TYPE:**
Buckling analysis, beam elements (BEAM2D).

**REFERENCE:**

**GIVEN:**
- \( L = b = 100 \text{ in} \)
- \( A = 1 \text{ in}^2 \)
- \( h = 1 \text{ in} \) (beam cross section height)
- \( I = 0.0833 \text{ in}^4 \)
- \( E = 1 \times 10^7 \text{ psi} \)
- \( P = 100 \text{ lb} \)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>( P_{cr} )</td>
<td>1372.4451 lb</td>
<td>1371.95 lb</td>
</tr>
</tbody>
</table>

**ANALYTICAL SOLUTION:**
\[
P_{cr} = 16.47EI/L^2 = 1372.4451 \text{ lb}
\]

Problem Sketch

Finite Element Model
**B11: Buckling of a Stepped Column**

**TYPE:**
Buckling analysis, beam element (BEAM2D).

**REFERENCE:**

**PROBLEM:**
Find the critical load and mode shape for the stepped column shown below.

**GIVEN:**

- L = 1000 mm
- A₁ = 10,954 mm²
- A₂ = 15,492 mm²
- E₁ = E₂ = 68,950 MPa
- ν₁ = ν₂ = 0.3
- I₁ = 1 x 10⁷ mm⁴
- I₂ = 2 x 10⁷ mm⁴
- P₁/P₂ = 0.5

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pᵦ</td>
<td>554.6 KN</td>
<td>554.51 KN</td>
</tr>
</tbody>
</table>

**ANALYTICAL SOLUTION:**

\[ P_\text{cr} = 0.326 \pi^2 E_1 I_1 / (2L)^2 = 554,600 \text{ N} \]

\[ = 554.6 \text{ KN} \]
B12: Buckling Analysis of a Simply Supported Composite Plate

**TYPE:**
Buckling analysis, composite shell element (SHELL4L).

**REFERENCE:**

**PROBLEM:**
Find the buckling load for \([45,-45,45,-45]\) antisymmetric angle-ply laminated plate under uniform axial compression \(p\).

**GIVEN:**
- \(a = b = 40\) in
- \(h = \sum h_i = 1\) in
- \(E_x = 400,000\) psi
- \(E_y = 10,000\) psi
- \(\nu_{xy} = 0.25\)
- \(G_{xy} = G_{yz} = G_{xz} = 5,000\) psi
- \(p = 1\) lb/in\(^2\)

**COMPARISON OF RESULTS:**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>(P_{cr})</td>
<td>334.0 lb/in</td>
<td>345.36 lb/in</td>
</tr>
</tbody>
</table>

**ANALYTICAL SOLUTION:**
Approximate solution is given by graph 5-16 in the reference.
**B13: Buckling of a Tapered Column**

**TYPE:**
Buckling analysis, beam element (BEAM2D).

**REFERENCE:**

**GIVEN:**
- \( b_1 = 1 \text{ in} \)
- \( b_2 = 4 \text{ in} \)
- \( \frac{b_2}{b_1} = \left(\frac{x}{a}\right)^2 \)
- \( I_1 = 1 \text{ in}^4 \)
- \( I_2 = 4 \text{ in}^4 \)
- \( \frac{I_1}{I_2} = 0.25 \)
- \( L = 100 \text{ in} \)
- \( a = 100 \text{ in} \)
- \( E = 1 \times 10^7 \text{ psi} \)
- \( h = 1 \text{ in} \)

**ANALYTICAL SOLUTION:**
\[ P_{cr} = \frac{1.678EI_2}{L^2} = 6712 \text{ lb} \]

**COMPARISON OF RESULTS:**
The Critical Load:

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>( P_{cr} )</td>
<td>6712 lb</td>
<td>6718 lb</td>
</tr>
</tbody>
</table>
Chapter 4  Buckling Analysis

---

**B14: Buckling of Clamped Cylindrical Shell Under External Pressure Using the Nonaxisymmetric Buckling Mode Option**

**TYPE:**
Linear buckling analysis using the nonaxisymmetric buckling mode option (SHELLAX).

**REFERENCE:**

**PROBLEM:**
Find the buckling pressure for the shown axisymmetric clamped-clamped shell.

**GIVEN:**
- \( R = 1 \text{ in} \)
- \( \nu = 0.3 \)
- \( L = 4 \text{ in} \)
- \( E = 10^7 \text{ psi} \)
- \( t = 0.01 \text{ in} \)

**MODELING HINTS:**
The cylindrical shell is modeled with 20 uniform elements. The starting harmonic number for which the buckling load is calculated is set to 2. The minimum buckling load occurs at harmonic number 5 which corresponds to mode shape 4 since the program started from harmonic 2.
### COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOS/M</th>
</tr>
</thead>
<tbody>
<tr>
<td>Harmonic Number</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>Critical Load</td>
<td>33.5 psi</td>
<td>35.0 psi</td>
</tr>
</tbody>
</table>
**B15A, B15B: Buckling of Simply-Supported Cylindrical Shell Under Axial Load**

**TYPE:**
Linear buckling analysis using the nonaxisymmetric buckling mode option (SHELLAX).

**REFERENCE:**

**PROBLEM:**
Find the buckling load for the simply-supported cylindrical shell shown in the figure below.

**GIVEN:**
- \( R = 10 \text{ in} \)
- \( L = 16 \text{ in} \)
- \( \nu (\text{NUXY}) = 0.3 \)
- \( E = 10^7 \text{ psi} \)
- \( t = 0.1 \text{ in} \)

**MODELING HINTS:**
The cylindrical shell was modeled with 60 uniform elements. The starting harmonic number for which the buckling load is calculated was set to 1. The solution stopped at harmonic number 2 at which the minimum buckling load occurs. The number of maximum iterations for the eigenvalue calculations was set to 100.
### COMPARISON OF RESULTS:

<table>
<thead>
<tr>
<th>Harmonic No.</th>
<th>Critical Load</th>
<th>( B15A ) (SHELLAX)</th>
<th>( B15B ) (PLANE2D)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>2</td>
<td>( 6.05 \times 10^4 ) lb/rad</td>
<td>( 6.05 \times 10^4 ) lb/rad</td>
</tr>
<tr>
<td>COSMOS/M</td>
<td>2</td>
<td>( 6.07 \times 10^4 ) lb/rad</td>
<td>( 6.02 \times 10^4 ) lb/rad</td>
</tr>
</tbody>
</table>
Index

A
axisymmetric 2-7, 2-29, 2-30, 2-60, 2-62, 2-135, 2-141, 4-16
axisymmetric shell 2-29, 2-60, 4-16

B
beam 2-16, 2-18, 2-35, 2-36, 2-38, 2-43, 2-45, 2-53, 2-54, 2-63, 2-65, 2-70, 2-80, 2-81, 2-83, 2-85, 2-86, 2-111, 2-113, 3-5, 3-6, 3-14, 3-15, 4-7, 4-12
beam and truss 2-41
beam elements 2-14, 2-16, 2-18, 2-35, 2-36, 2-38, 2-43, 2-54, 2-70, 2-80, 2-81, 2-83, 2-85
BEAM2D 1-3, 2-16, 2-18, 2-38, 2-65, 2-71, 2-80, 2-81, 2-83, 2-85, 2-111, 2-113, 4-12, 4-13, 4-15
BEAM3D 1-3, 2-14, 2-35, 2-36, 2-41, 2-43, 2-45, 2-53, 2-54, 2-63, 2-67, 2-70, 2-86, 3-5, 3-6, 3-7, 3-14, 3-19, 4-2, 4-3, 4-4, 4-7, 4-11
BOUND 1-3
Buckling analysis 1-2, 4-1, 4-12, 4-13, 4-14, 4-15, 4-16, 4-18

C
centrifugal loading 2-62, 2-63
composite shell 2-33, 2-47
composite solid 2-56
constraint 2-68, 2-70, 2-71, 2-73
constraint elements 2-73
constraint equations 2-68
coupled degrees of freedom 2-129
coupled points 2-68
CPCNS 2-70
cross-ply laminated 2-33
cyclic symmetry 2-99, 2-100

E
elastic foundation 2-128
ELBOW 1-3, 2-25, 2-26, 2-130

F
fluid sloshing 3-23, 3-25, 3-27
Frequency (modal) analysis 1-2
frequency analysis 3-21, 3-23, 3-30, 3-31, 3-32, 3-34
fundamental frequency 3-4, 3-5, 3-17, 3-19, 3-20, 3-34, 3-35

G
GAP 1-3
gap elements 2-111, 2-113, 2-114
GENSTIF 1-3
gravity loading 2-130
Guyan reduction 3-32

L
linear static analysis 1-2, 2-1, 2-29, 2-111, 2-113, 2-130
Linear static stress analysis 1-2
Linearized buckling analysis 1-2

M
MASS 1-3, 2-63, 3-3, 3-7
mass elements 2-63
Modal analysis 1-2
Mode shapes and frequencies 3-3, 3-4, 3-5, 3-6, 3-7, 3-8, 3-9, 3-10, 3-11, 3-12, 3-13, 3-14, 3-15, 3-16, 3-17, 3-19, 3-20, 3-21, 3-22, 3-23, 3-25, 3-27, 3-29

N
natural frequencies 3-3, 3-6, 3-8, 3-9, 3-10, 3-11, 3-12, 3-13, 3-15, 3-17, 3-21, 3-23, 3-25, 3-32
| O | orthotropic 2-33 |
| P | p-element 2-106, 2-117 |
|   | Piezoelectric 3-29 |
|   | piezoelectric 3-29 |
|   | PIPE 1-3, 3-8 |
|   | plane strain 2-12, 2-131, 2-133, 2-137 |
|   | plane strain solid 2-103 |
|   | plane stress 2-13, 2-106, 2-117, 2-131 |
| R | RBAR 1-3 |
|   | rigid bar 3-32, 3-33 |
|   | rigid body modes 3-14 |
| S | sandwich beam 2-109 |
|   | shell 2-29, 2-31, 2-39, 2-48, 2-50, 2-53, 2-66, 2-71, 2-89, 2-95, 4-10 |
|   | shell element 2-29, 2-31, 2-50, 2-67, 2-87, 2-89, 2-91, 2-95, 3-33 |
|   | SHELL3 1-3, 2-9, 2-15, 2-48, 3-9, 3-13, 4-6 |
|   | SHELL3L 1-3 |
|   | SHELL3T 1-3, 3-10, 3-17 |
|   | SHELL4 1-3, 2-31, 2-39, 2-53, 2-58, 2-66, 2-68, 2-121, 2-128, 3-9, 3-11, 3-12, 3-20, 3-32, 4-5, 4-9, 4-11 |
|   | SHELL4L 1-3, 2-33, 2-50, 2-67, 2-93, 2-109, 4-14 |
|   | SHELL4T 1-4, 2-79, 3-17, 3-22 |
|   | SHELL6 1-4, 2-13, 2-31, 2-66, 2-79, 3-13, 3-22, 4-6, 4-9 |
|   | SHELL9 1-4, 2-71, 2-87, 2-89, 2-91, 2-95 |
|   | SHELL9L 1-4, 2-33, 2-47, 2-93 |
|   | SHELLAX 1-4, 2-29, 2-60, 2-119, 2-120, 2-121, 3-21, 3-30, 3-31, 4-10, 4-16, 4-18 |
|   | SOLID 1-4, 2-19, 2-21, 2-47, 2-56, 2-73, 2-77, 2-79, 2-115, 3-15, 3-22, 3-23, 3-25 |
|   | SOLIDL 1-4, 2-47, 2-56, 2-93 |
|   | SOLIDPZ 1-4, 3-29 |
|   | SPRING 1-4 |
|   | stepped column 4-13 |
|   | stress intensity factor 2-97, 2-98, 2-131, 2-133, 2-135, 2-137 |
|   | stress stiffening 2-54, 3-34 |
|   | submodeling 2-127 |
|   | substructuring 2-51, 2-53, 2-58 |
|   | support reactions 2-80, 2-81 |
| T | TETRA10 1-4, 2-73, 3-22 |
|   | TETRA4 1-4, 2-110 |
|   | TETRA4R 1-4, 2-79, 2-91, 2-110, 3-22 |
|   | thermal stress analysis 2-10, 2-12, 2-21, 2-28, 2-35, 2-36, 2-38, 2-108 |
|   | Thick Shell 2-48 |
|   | thick shell 2-15 |
|   | thin plate 2-9 |
|   | thin shell 2-48 |
|   | TRIANG 1-4, 2-79, 2-100, 2-106, 2-107, 2-117, 2-127, 2-131, 2-132, 2-133, 2-135, 3-22 |
|   | truss 2-6, 2-10, 2-22, 2-23, 2-24, 2-35, 2-51, 2-53, 2-64, 2-99, 2-113, 3-16, 4-7 |
|   | TRUSS2D 1-4, 2-10, 2-51, 2-64, 2-99, 2-113, 3-16, 4-7 |
|   | TRUSS3D 1-4, 2-6, 2-22, 2-23, 2-24, 2-35, 2-41, 2-53, 3-3 |