NONLINEAR ANALYSIS OF PLANAR REINFORCED CONCRETE STRUCTURES

By
M. H. SALEM
B. MOHRAZ

A report on the study of
"Investigation of Multiple Opening Reinforced Concrete Conduits"
Sponsored by
Department of the Army
Corps of Engineers
Engineering Division
Civil Works
Contract No. DACW-73-72-C-0065

UNIVERSITY OF ILLINOIS
at URBANA-CHAMPAIGN
URBANA, ILLINOIS
JULY 1974
NONLINEAR ANALYSIS OF PLANAR
REINFORCED CONCRETE STRUCTURES

by
M. H. Salem
B. Mohraz

A Report on the Study of
"Investigation of Multiple Opening
Reinforced Concrete Conduits"

Sponsored by

Department of the Army
Corps of Engineers
Engineering Division
Civil Works
Contract No. DACW-73-72-C-0065

UNIVERSITY OF ILLINOIS
AT URBANA-CHAMPAIGN
URBANA, ILLINOIS

JULY 1974
ACKNOWLEDGEMENT

The study reported herein is based on a doctoral dissertation by Mr. Mohammed H. Salem under the direction of Dr. Bijan Mohraz, Assistant Professor of Civil Engineering. The work was carried out as part of an investigation of Multiple Opening Reinforced Concrete Conduits sponsored by the Department of the Army, Corps of Engineers, Engineering Division, Civil Works under contract No. DACW-73-72-C-0065.

The computer program was code-checked on the Burroughs 6700 computer facilities of the Civil Engineering Systems Laboratory of the Department of Civil Engineering and the numerical examples were obtained using the IBM 360/75 computer facilities of the Department of Computer Science at the University of Illinois.
TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>CHAPTER</th>
<th>INTRODUCTION</th>
<th>MATERIAl BEHAVIOR</th>
<th>IDEALIZATION OF STRUCTURES BY FINITE ELEMENTS</th>
<th>METHOD OF SOLUTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>General</td>
<td>General</td>
<td>General</td>
<td>General</td>
</tr>
<tr>
<td>1.1</td>
<td>1.2</td>
<td>2.1</td>
<td>3.1</td>
<td></td>
</tr>
<tr>
<td>1.3</td>
<td>Previous Work</td>
<td>Plain Concrete</td>
<td>The Displacement Model</td>
<td>The Triangular Element</td>
</tr>
<tr>
<td>1.4</td>
<td>Object and Scope</td>
<td>Reinforced Concrete</td>
<td>The Finite Elements Used in This Study</td>
<td>The Tie-Link Element</td>
</tr>
<tr>
<td>1.4.1</td>
<td>Notation</td>
<td>Failure Criteria for Concrete</td>
<td>The Linear Isoparametric Quadrilateral Element</td>
<td>The Bar Element</td>
</tr>
<tr>
<td>2.</td>
<td>Stress-Strain Relations for Concrete</td>
<td>Stress-Strain Relations for Concrete</td>
<td>Stress-Strain Relations for the Steel Reinforcement</td>
<td>Solution of Linear Equations</td>
</tr>
<tr>
<td>2.5.1</td>
<td>Elastic Properties</td>
<td>Total Concrete Cracked in One Direction</td>
<td>3.5</td>
<td>4.2 Solution of Linear Equations</td>
</tr>
<tr>
<td>2.5.2</td>
<td>Concrete Cracked in One Direction</td>
<td>Concrete Cracked in Two Directions</td>
<td>3.6</td>
<td></td>
</tr>
<tr>
<td>2.5.3</td>
<td>Concrete Cracked in Two Directions</td>
<td>Plastic Stiffness</td>
<td>3.7</td>
<td></td>
</tr>
<tr>
<td>2.5.4</td>
<td>Plastic Stiffness</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.1</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CHAPTER</td>
<td>Page</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>---------</td>
<td>------</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.3</td>
<td>Solution of Nonlinear Equations</td>
<td>48</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.3.1</td>
<td>Review of Available Methods</td>
<td>48</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.3.2</td>
<td>The Incremental-Iterative Procedure</td>
<td>50</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.3.3</td>
<td>Outline of the Computational Steps</td>
<td>53</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.4</td>
<td>Evaluation of Excessive Stresses</td>
<td>55</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.5</td>
<td>The Computer Program</td>
<td>58</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>NUMERICAL SOLUTIONS</td>
<td>59</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.1</td>
<td>General</td>
<td>59</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.2</td>
<td>The Thick Hollow Cylinder</td>
<td>60</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.3</td>
<td>Shallow Reinforced Concrete Beam</td>
<td>61</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.3.1</td>
<td>Experimental Beam Geometry and Behavior</td>
<td>61</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.3.2</td>
<td>Behavior Predicted From the Analysis</td>
<td>62</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.4</td>
<td>Deep Reinforced Concrete Beam</td>
<td>65</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.4.1</td>
<td>Experimental Beam Geometry and Behavior</td>
<td>65</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.4.2</td>
<td>Finite Element Solutions</td>
<td>66</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.4.3</td>
<td>Internal Stress Distribution in the Model With Tie-Link Elements</td>
<td>69</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.5</td>
<td>Reinforced Concrete Conduit 3 to 1 Loading</td>
<td>72</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.5.1</td>
<td>Experimental Model Geometry and Behavior</td>
<td>72</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.5.2</td>
<td>The Finite Element Solution for Model R4</td>
<td>74</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.6</td>
<td>Reinforced Concrete Conduit 1 to 1 Loading</td>
<td>77</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.6.1</td>
<td>Experimental Model Geometry and Behavior</td>
<td>77</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.6.2</td>
<td>The Finite Element Solution for Model R5</td>
<td>78</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>SUMMARY AND CONCLUSIONS</td>
<td>81</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>LIST OF REFERENCES</td>
<td>85</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

APPENDIX

<p>| A   | CONSTITUTIVE LAWS OF INCREMENTAL PLASTICITY          | 150 |
| B   | ELEMENT STIFFNESS MATRICES                          | 158 |
| B.1 | General                                            | 158 |
| B.2 | Stiffness Matrix for the Two-Dimensional Truss Element | 158 |
| B.3 | Stiffness Matrix for the 12 DOF Isoparametric Quadrilateral Element | 162 |
| B.4 | Stiffness Matrix for the Tie-Link Element           | 170 |</p>
<table>
<thead>
<tr>
<th>Appendix C</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>THE COMPUTER PROGRAM</td>
<td>174</td>
</tr>
<tr>
<td>C.1</td>
<td>General</td>
<td>174</td>
</tr>
<tr>
<td>C.2</td>
<td>Program Organization and Flow Chart</td>
<td>175</td>
</tr>
<tr>
<td>C.3</td>
<td>Input Data and Presentation of Results</td>
<td>176</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>-------------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>2.1</td>
<td>Actual and Assumed Uniaxial Stress-Strain Curve for Concrete</td>
<td>90</td>
</tr>
<tr>
<td>2.2</td>
<td>Empirical Stress-Strain Curves for Concrete</td>
<td>90</td>
</tr>
<tr>
<td>2.3</td>
<td>Stress-Strain Curves for Concrete Under Biaxial Compression (Ref. 18)</td>
<td>91</td>
</tr>
<tr>
<td>2.4</td>
<td>Effect of Stress State on the Behavior of Concrete in Tension (Ref. 20)</td>
<td>91</td>
</tr>
<tr>
<td>2.5</td>
<td>Tensile Strength as a Function of Compressive Strength (Ref. 20)</td>
<td>92</td>
</tr>
<tr>
<td>2.6</td>
<td>Flexural and Diagonal Tension Cracks in Reinforced Concrete</td>
<td>93</td>
</tr>
<tr>
<td>2.7</td>
<td>Biaxial Strength of Concrete</td>
<td>94</td>
</tr>
<tr>
<td>2.8</td>
<td>Cracked Concrete Element</td>
<td>95</td>
</tr>
<tr>
<td>2.9</td>
<td>Stress-Strain Curve for Steel</td>
<td>95</td>
</tr>
<tr>
<td>2.10</td>
<td>von Mises Yield Surface</td>
<td>95</td>
</tr>
<tr>
<td>3.1</td>
<td>Linear Isoparametric Quadrilateral Element</td>
<td>96</td>
</tr>
<tr>
<td>3.2</td>
<td>Source of Error in the Linear Isoparametric Element (Ref. 43)</td>
<td>96</td>
</tr>
<tr>
<td>3.3</td>
<td>Shape Functions for the Incompatible Model</td>
<td>97</td>
</tr>
<tr>
<td>3.4</td>
<td>Tie-Link Element</td>
<td>97</td>
</tr>
<tr>
<td>4.1</td>
<td>Element Side Pressure</td>
<td>98</td>
</tr>
<tr>
<td>4.2</td>
<td>Graphic Representation of Nonlinear Problem Solutions (Ref. 50)</td>
<td>98</td>
</tr>
<tr>
<td>4.3</td>
<td>Mixed Incremental-iterative Procedure</td>
<td>99</td>
</tr>
<tr>
<td>4.4</td>
<td>Graphical Representation of Concrete Becoming Plastic</td>
<td>99</td>
</tr>
<tr>
<td>5.1</td>
<td>Finite Element Idealization of the Thick Cylinder</td>
<td>100</td>
</tr>
<tr>
<td>5.2</td>
<td>Internal and External Displacements and Pressures as a Function of the Elastic-Plastic Boundary</td>
<td>102</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>5.2</td>
<td>Internal and External Displacements and Pressures as a Function of the Elastic-Plastic Boundary</td>
<td>102</td>
</tr>
<tr>
<td>5.3</td>
<td>The Distribution of Radial, Hoop, and Axial Stresses When the Radius of the Plastic Zone is 1.4a</td>
<td>103</td>
</tr>
<tr>
<td>5.4</td>
<td>Numerical Solution 2 -- Shallow Beam (Beam L-6)</td>
<td>104</td>
</tr>
<tr>
<td>5.5</td>
<td>Finite Element Idealization of Beam L-6 (Mesh 1)</td>
<td>105</td>
</tr>
<tr>
<td>5.6</td>
<td>Cracking Pattern at Yield Load, Beam L-6 (Mesh 1)</td>
<td>105</td>
</tr>
<tr>
<td>5.7</td>
<td>Stresses in the Steel Reinforcement, Beam L-6 (Mesh 1)</td>
<td>105</td>
</tr>
<tr>
<td>5.8</td>
<td>Load-Deflection Curves for Beam L-6 (Mesh 1)</td>
<td>106</td>
</tr>
<tr>
<td>5.9</td>
<td>Finite Element Idealization of Beam L-6 (Mesh 2)</td>
<td>107</td>
</tr>
<tr>
<td>5.10</td>
<td>Cracking Pattern at Yield Load, Beam L-6 (Mesh 2)</td>
<td>107</td>
</tr>
<tr>
<td>5.11</td>
<td>Load-Deflection Curves for Beam L-6 (Mesh 2)</td>
<td>108</td>
</tr>
<tr>
<td>5.12</td>
<td>Load-Deflection Curves for Beam L-6 (Mesh 2)</td>
<td>109</td>
</tr>
<tr>
<td>5.13</td>
<td>Numerical Solution 3 -- Deep Beam</td>
<td>110</td>
</tr>
<tr>
<td>5.14</td>
<td>Finite Element Layout</td>
<td>111</td>
</tr>
<tr>
<td>5.15</td>
<td>Load-Deflection Curves for the Deep Beam</td>
<td>113</td>
</tr>
<tr>
<td>5.16</td>
<td>Steel Stress Distribution</td>
<td>114</td>
</tr>
<tr>
<td>5.17</td>
<td>Stress Distribution in the Lower Layer of Tie-Link Elements</td>
<td>115</td>
</tr>
<tr>
<td>5.18</td>
<td>Experimental Cracking Pattern for the Deep Beam</td>
<td>115</td>
</tr>
<tr>
<td>5.19</td>
<td>Bond-Slip Curve After Nilson (Ref. 2)</td>
<td>116</td>
</tr>
<tr>
<td>5.20</td>
<td>Cracking Pattern at Different Load Levels for the Deep Beam</td>
<td>117</td>
</tr>
<tr>
<td>5.21</td>
<td>Direction and Relative Magnitudes of the Principal Stresses in the Deep Beam</td>
<td>118</td>
</tr>
<tr>
<td>5.22</td>
<td>Deformed Element Plots for the Deep Beam</td>
<td>119</td>
</tr>
<tr>
<td>5.23</td>
<td>Distribution of Longitudinal Strain Across the Depth of the Deep Beam</td>
<td>120</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>-------</td>
<td>------------------------------------------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>5.24</td>
<td>Distribution of the Longitudinal Stress Across the Depth of the Deep Beam</td>
<td>121</td>
</tr>
<tr>
<td>5.25</td>
<td>Distribution of the Shear Stresses Across the Depth of the Deep Beam</td>
<td>122</td>
</tr>
<tr>
<td>5.26</td>
<td>Distribution of Vertical Stresses Along the Deep Beam</td>
<td>123</td>
</tr>
<tr>
<td>5.27</td>
<td>Dimensions and Locations of Loading Jacks for Test Specimens R4 and R5</td>
<td>124</td>
</tr>
<tr>
<td>5.28</td>
<td>Loading Pattern and Material Idealization for Conduit R4</td>
<td>125</td>
</tr>
<tr>
<td>5.29</td>
<td>Photographs of Specimen R4 After Failure</td>
<td>126</td>
</tr>
<tr>
<td>5.30</td>
<td>The Finite Element Mesh for Specimens R4 and R5</td>
<td>127</td>
</tr>
<tr>
<td>5.31</td>
<td>Node Identification and Locations of Bar Elements for Specimens R4 and R5</td>
<td>128</td>
</tr>
<tr>
<td>5.32</td>
<td>Load-Deflection Curves for Specimen R4</td>
<td>129</td>
</tr>
<tr>
<td>5.33(a)</td>
<td>Load-Steel Strain Curves for Specimen R4 (Compression)</td>
<td>130</td>
</tr>
<tr>
<td>5.33(b)</td>
<td>Load-Steel Strain Curves for Specimen R4 (Tension)</td>
<td>131</td>
</tr>
<tr>
<td>5.34</td>
<td>Predicted Cracking Pattern for Specimen R4</td>
<td>132</td>
</tr>
<tr>
<td>5.35</td>
<td>Direction and Relative Magnitudes of the Principal Stresses in Specimen R4 at a Vertical Load of 90 KSF</td>
<td>133</td>
</tr>
<tr>
<td>5.36</td>
<td>Deformed Element Plot for Specimen R4 at a Vertical Load of 90 KSF</td>
<td>134</td>
</tr>
<tr>
<td>5.37</td>
<td>Distribution of Horizontal Strains in Specimen R4 at a Vertical Load of 90 KSF</td>
<td>135</td>
</tr>
<tr>
<td>5.38</td>
<td>Distribution of the Horizontal Stresses in Specimen R4 at a Vertical Load Level of 90 KSF</td>
<td>136</td>
</tr>
<tr>
<td>5.39</td>
<td>Distribution of Shear Stresses in Specimen R4 at a Vertical Load of 90 KSF</td>
<td>137</td>
</tr>
<tr>
<td>5.40</td>
<td>Loading Pattern and Material Idealization for Conduit R5</td>
<td>138</td>
</tr>
<tr>
<td>5.41</td>
<td>Photographs of Specimen R5 After Failure</td>
<td>139</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>5.42</td>
<td>Predicted Cracking Pattern for Specimen R5</td>
<td>140</td>
</tr>
<tr>
<td>5.43</td>
<td>Direction and Relative Magnitudes of the Principal Stresses in Specimen R5 at 67.5 KSF Load Level</td>
<td>141</td>
</tr>
<tr>
<td>5.44</td>
<td>Deformed Element Plot for Specimen R5 at 67.5 KSF Load Level</td>
<td>142</td>
</tr>
<tr>
<td>5.45</td>
<td>Load-Deflection Curves for Specimen R5</td>
<td>143</td>
</tr>
<tr>
<td>5.46(a)</td>
<td>Load-Steel Strain Curves for Specimen R5 (Compression)</td>
<td>144</td>
</tr>
<tr>
<td>5.46(b)</td>
<td>Load-Steel Strain Curves for Specimen R5 (Tension)</td>
<td>145</td>
</tr>
<tr>
<td>5.47</td>
<td>Distribution of Vertical Strains in Specimen R5 at 67.5 KSF Load Level</td>
<td>146</td>
</tr>
<tr>
<td>5.48</td>
<td>Distribution of Vertical Stresses in Specimen R5 at 67.5 KSF Load Level</td>
<td>147</td>
</tr>
<tr>
<td>5.49</td>
<td>Distribution of Shear Stresses in Specimen R5 at 67.5 KSF Load Level</td>
<td>148</td>
</tr>
<tr>
<td>5.50</td>
<td>Distribution of Horizontal Stresses in Specimen R5 at 67.5 KSF Load Level</td>
<td>149</td>
</tr>
<tr>
<td>A.1</td>
<td>Geometric Interpretation of the Normality Rule</td>
<td>157</td>
</tr>
<tr>
<td>B.1</td>
<td>Plane Truss Element</td>
<td>173</td>
</tr>
<tr>
<td>B.2</td>
<td>Tie-Link Element</td>
<td>173</td>
</tr>
<tr>
<td>C.1</td>
<td>Flow Chart for the Computer Program</td>
<td>179</td>
</tr>
<tr>
<td>C.2</td>
<td>Automatic Mesh Generation (Ref. 53)</td>
<td>180</td>
</tr>
<tr>
<td>C.3</td>
<td>Alphanumeric Finite Element Mesh Plot</td>
<td>181</td>
</tr>
</tbody>
</table>
CHAPTER 1

INTRODUCTION

1.1 General

Knowledge of the real behavior of reinforced concrete structures in the post-cracking stage is of fundamental importance to designers. The safety of most structures can be assessed correctly if estimates of their ultimate load carrying capacity can be obtained. Unfortunately, however, there is no precise reinforced concrete theory by which the distribution of internal forces at ultimate load can be determined. Due to our limited knowledge of the composite behavior of steel and concrete, many design formulas are empirical in nature and are based on the results of a large number of experiments performed specifically for studying the behavior of certain types of structures.

Present practices in the design of a reinforced concrete section subjected to pure bending, ignore the tensile strength of the cracked concrete altogether. Contrary to this, it is assumed that the cracked concrete can transmit part of the diagonal tensile stresses caused by the longitudinal shear stresses in a member subjected to both flexure and shear. This rather crude analytical model is also based on the hypothesis that plane sections remain plane and that full bond exists between steel and concrete. While this model has been used successfully for the design of conventional reinforced concrete structures, it was soon realized
that it is inadequate for explaining the behavior of complex structures such as box culverts under high embankments, shear walls, nuclear containment vessels, etc. For example, current practices for the design of box culverts consider each culvert as a closed frame subjected to uniform loads. Various members of the frame are then analyzed as beams subjected to a combination of axial force, shear, and bending. However, in the case of culverts under high embankments, members of the frames are thick and are subjected to large axial forces. Moreover, the state of stress in these members is biaxial rather than uniaxial.

Therefore, a more accurate method of analysis is imperative. The proposed method should be capable of handling important effects such as the nonlinear behavior of concrete and steel, time dependent properties of concrete (creep and shrinkage), bond-slip action between steel and concrete, aggregate interlock along the crack surface, and the dowel action of the longitudinal reinforcement. In view of such complexities in the model and because of the continuously changing topology of the structure as cracks propagate, any attempt to determine the principal stresses in a reinforced concrete structure by direct application of the classical theories of continuum mechanics is virtually impossible.

One method for obtaining information on the post-cracking behavior of a reinforced concrete structure and its ultimate load carrying capacity is to build a small scale model of the structure and test it to failure. However, for test results to be meaningful and reliable, very small scale reinforced concrete models should be avoided. But, prototypes and large
scale models are expensive to build. In addition, new models are needed when a certain influence is to be studied, making the procedure both time consuming and expensive.

Because of the versatility of the finite element method in obtaining solutions to structural and continuum mechanics problems, many attempts have been made to extend the method to the analysis of reinforced concrete beams, frames and shear walls. The finite element method can be thought of as a numerical procedure where the solution of a problem in continuum mechanics is approximated by the solution of a highly redundant articulated structure by the standard matrix method of structural analysis. At the present time the finite element library contains a number of refined elements that can provide solution to many elastic problems.

The basic prerequisite for post-elastic analysis of reinforced concrete structures by the finite element method is a realistic idealization of the material behavior. Such an idealization should provide an adequate relationship between the stresses and strains, usually in incremental form, which reflects yielding of the reinforcing steel as well as cracking and crushing of the concrete. The accuracy of the finite element solution depends on how closely the constitutive relations of the materials are approximated. Therefore, comparisons of computational and experimental results are necessary for establishing the adequacy of the analytical model.
1.2 Previous Work

The first application of the finite element method to reinforced concrete was carried out by Ngo and Scordelis, who analyzed a cracked reinforced concrete beam assuming it to be linearly elastic. Nonlinear analysis of reinforced concrete beams has been carried out by Nilson. Both investigations used separate elements for the concrete and the reinforcing steel with special link elements to connect the two. Ngo, Franklin, and Scordelis used a similar approach to study the shear behavior of beams with diagonal tension cracks. The link elements used in these investigations have no dimension and are supposed to account for bond-slip, dowel action and aggregate interlock. In early nonlinear analysis the crack was propagated through the model by continuously changing the topology of the model as the crack extended from one element to the next. The computer solution had to be stopped each time the principal stresses in any element exceeded the cracking stress and then a new cracked structure had to be redefined before the solution was resumed. Such an approach has the disadvantage that either the crack direction is restricted to lines defining the edge of the element or a rezoning (a topological modification) is required when new cracks are formed.

Another method of incorporating cracks was introduced by Mohraz, Schnobrich and Echeverria Gomez, in which an incremental loading procedure was used to trace the response of a prestressed concrete reactor vessel to an increasing internal static pressure in one continuous computer analysis. This method calls for modifying the material property
of the region containing the crack. New orthotropic constitutive relations were obtained using an energy approach. Iterations were performed within each loading increment once a crack appeared and the resulting unbalanced stresses were distributed in the uncracked region.

Relevant studies carried out at the University of Illinois at Urbana-Champaign include the work done by Yuzugulu and Schnobrich, in which reinforced concrete shear wall-frame systems were analyzed using a composite plane stress quadrilateral element; the orthotropic reinforcement was incorporated in the material property of the element. The results of their analysis correlated fairly well with the experimental results. Storm analyzed a single-story one-bay reinforced concrete frame with brick and mortar infill composite. Hand et al. used a layered finite element to analyze reinforced concrete plates and shells. Finally, Suidan and Schnobrich used an isoparametric brick element to study the behavior of reinforced concrete beams.

Similar work has been carried out at other institutions. Franklin analyzed reinforced concrete frames with and without infilled shear panels using a layered frame-type element, quadrilateral plane stress elements, and link elements. Cervenka analyzed shear-wall panels and compared the analytical results with his experimental studies. Recently, Lin, using a similar approach to that of Hand, made an extensive study of the behavior of reinforced concrete slabs and shells in the nonlinear range.

A comprehensive list of references on this subject may be found in a recent state-of-the-art paper by Scordelis.
1.3 **Object and Scope**

The objective of this investigation is to develop a procedure for analyzing planar reinforced concrete structures in the nonlinear range. The proposed procedure can be used to predict the ultimate load carrying capacity and the behavior of a reinforced concrete structure throughout its loading history.

An attempt is made to formulate a material model for reinforced concrete that reflects the behavior of concrete and steel in actual structures. The general behavior of plain, as well as, reinforced concrete is briefly discussed, and mechanisms of shear transfer in cracked reinforced concrete members are reviewed in an attempt to incorporate the dominant modes of behavior in the proposed material model. The study is limited to short time behavior of reinforced concrete structures under monotonically increasing loads.

The investigation employs the linear isoparametric quadrilateral element with incompatible modes in obtaining solutions to reinforced concrete structures. In terms of economy this element provides a suitable alternative to the simple constant strain triangular element which has been used extensively in nonlinear finite element analysis.

Several parametric studies have been carried out to determine the sensitivity of the model to various material input parameters. The adequacy of the proposed procedure is illustrated by obtaining solutions to several reinforced concrete structures and comparing them with experimental results.
1.4  **Notation**

The symbols used in this work are defined where they first appear. For convenience they are summarized below:

- \( A \)  = bar cross-sectional area, or a constant
- \( a \)  = constant
- \( B \)  = constant
- \( [B],[B^*] \)  = transformation matrices relating strains and displacements
- \( b \)  = constant
- \( C \)  = compressive force in a reinforced concrete section
- \( C_1, C_2, C_3 \)  = constants
- \( c \)  = cosine of an angle
- \( [D],[D_c],[\overline{D}] \)  = material property matrices
- \( [D]_{ep}, [D]_p \)  = elasto-plastic and plastic material property matrices
- \( [D]_j \)  = material property matrix during iteration \( j \)
- \( d \)  = denotes the variation or derivative
- \( dA \)  = elementary area
- \( ds \)  = elementary line segment
- \( dv \)  = elementary volume
- \( d\lambda \)  = non-negative constant
- \( E, E_c, E_S \)  = moduli of elasticity
- \( E_{sc} \)  = secant modulus
\( F \) = yield surface
\( \{F\} \) = vector of generalized loads for the whole structure
\( \{f\} \) = vector of generalized loads for one element
\( f'_C \) = concrete cylinder compressive strength
\( f'_t \) = tensile strength of concrete
\( G \) = shear modulus or constant
g = dead weight
\( H_s, H_i, H_j \) = slope of the uniaxial stress-equivalent plastic strain, or weighting coefficient of numerical integration
I = moment of inertia
\( J_1, J_2, J_3 \) = stress invariants
\( [J] \) = Jacobian matrix
\( [K] \) = structural stiffness matrix
k = yield stress in pure shear
\( k_h, k_v \) = spring stiffnesses
\( [k] \) = element stiffness matrix
L = span length
\( \lambda \) = bar length
M = applied moment
\( N_i \) = shape function at node \( i \)
\( N_5, N_6 \) = incompatible shape functions
\( \{N\} \) = vector containing the components of the normal to the yield surface
\[
\begin{align*}
[N] & \quad = \text{matrix of shape functions and zeros} \\
n & \quad = \text{total number of unknowns} \\
\{n_1\},\{n_2\},\{n_3\} & \quad = \text{derivatives of the stress invariants with respect to the stresses at a point} \\
P & \quad = \text{arbitrary load} \\
p & \quad = \text{subscript indicating "plastic", or pressure intensity} \\
\{q\},\{q\} & \quad = \text{vectors of generalized displacements} \\
\{R\},\{r\} & \quad = \text{residual load vector for the whole structure and for one element} \\
r & \quad = \text{ratio} \\
S_x, S_y, S_z & \quad = \text{deviatoric stresses} \\
s & \quad = \text{one-dimensional coordinate or sine of an angle} \\
T & \quad = \text{tensile force in the longitudinal reinforcement} \\
[T_e] & \quad = \text{strain transformation matrix} \\
\{U\} & \quad = \text{vector of the generalized displacements for the whole structure} \\
U_0 & \quad = \text{strain energy} \\
u,v & \quad = \text{components of displacements in x and y directions} \\
\{u\} & \quad = \text{vector of u and v displacements at a point} \\
\bar{u}_i, \bar{v}_j & \quad = \text{u and v displacements at node i} \\
\{\bar{u}\} & \quad = \text{vector of nodal displacements} \\
V_a & \quad = \text{shear carried by aggregate interlock} \\
V_c & \quad = \text{shear carried by uncracked concrete}
\end{align*}
\]


\( V_d \) = shear carried by dowel action  \\
\( V_s \) = shear carried by shear reinforcement  \\
\( V_{\text{tot}} \) = vertical reaction  \\
\( W_p \) = potential of applied loads  \\
\( x, y, z \) = global coordinates  \\
\( x_i, y_i \) = global coordinates at node \( i \)  \\
\( \alpha \) = ratio of principal stresses, or experimental material constant  \\
\( [\alpha] \) = row of direction cosines, or column of non-nodal displacements  \\
\( [\alpha]_i \) = diagonal matrix of over-relaxation coefficients at load step \( i \)  \\
\( \beta \) = experimental material constant  \\
\( \beta_1, \beta_2 \) = constants  \\
\( \varepsilon \) = uniaxial strain  \\
\( \varepsilon_{\text{cr}} \) = cracking strain in concrete  \\
\( \varepsilon_0 \) = strain at peak stress  \\
\( \varepsilon_{\text{uc}} \) = concrete crushing strain  \\
\( \varepsilon_{\text{ut}} \) = concrete strain at zero tensile stress (after cracking)  \\
\( \varepsilon_p \) = equivalent uniaxial plastic strain  \\
\( \{\varepsilon\}, \{\varepsilon\} \) = strain vector at a point  \\
\( \{\varepsilon_e\}, \{\varepsilon_p\} \) = elastic and plastic strain vectors  \\
\( \Delta \) = denotes increment  \\
\( \theta \) = crack angle or inclination of the tie-link element
\( \kappa \) = hardening parameter

\( \mu \) = shear reduction factor

\( \nu \) = Poisson's ratio

\( \xi, \eta \) = local natural dimensionless coordinates

\( \bar{\xi}_i, \bar{\eta}_i \) = local natural coordinates at node \( i \)

\( \Pi \) = total potential energy

\( \sigma \) = uniaxial stress

\( \sigma_1, \sigma_2, \sigma_3 \) = principal stresses

\( \sigma_m \) = mean stress

\( \sigma_y \) = yield stress

\( \{ \sigma \}, \{ \bar{\sigma} \} \) = vector of stresses at a point

\( \{ \sigma_{\text{ex}} \} \) = excessive stresses

\( \tau_{\text{oct}} \) = octahedral shearing stress

\( \omega \) = unit weight of concrete in \( \text{lb/ft}^3 \)
2.1 General

The success of any finite element solution depends for the most part on selecting the realistic idealization of the material behavior as established from experimental results. Concrete, being composed of aggregates and mortar, is heterogeneous in nature; hence, it is very difficult to idealize its stress-strain relationship. Concrete exhibits orthotropic behavior after cracking. Due to the confinement effect, the orthotropic behavior occurs even under biaxial compression. Moreover, the material constants are difficult to establish because of the many uncertainties involved in determining the compressive and tensile strength of plain concrete. Both the standard cylinder test and the split cylinder test may exhibit wide variance for different batches of the same mix.

Amid all these uncertainties, the structural analyst is faced with making a number of decisions before any analysis can begin; specifically decisions regarding what material constants are to be used and which failure criteria are to be adopted. In this chapter a brief review of the behavior of plain concrete as well as that of reinforced concrete is presented and constitutive relations for use at various load levels are discussed. The discussion is limited to monotonically increasing loads.
2.2 Plain Concrete

A typical stress-strain curve for concrete is presented in Fig. 2.1. Under compression, concrete remains linear up to about 30 per cent of its ultimate strength. The curve reaches its peak at a strain of 0.002 to 0.003. The unloading portion in compression is indicative of the material disintegration that occurs before final crushing. Non-linear behavior of concrete is, in general, attributed to internal microcracks which initiate at the interface between larger aggregates and the surrounding mortar. At about 30 per cent of the ultimate load, these bond-type microcracks begin to increase in length, width, and number with increasing strain. At this load level the stress-strain curve begins to deviate from a straight line. X-ray photographs have shown that the bond cracks penetrate slowly through the mortar. The ultimate load carrying capacity of the specimen under uniaxial compression is reached when cracks form a continuous pattern causing the stress-strain curve to bend downwards. This hypothesis has been confirmed recently by Liu, Nilson, and Slate.

The shape of the uniaxial stress-strain curve in compression varies with the strain rate and the concrete strength. Many equations have been formulated to describe the standard cylinder test curves. These equations tend to plot below the test result curves for low strength concretes and above it for concretes with high strength; with the best results obtained from a curve fit. The European Concrete Committee suggests the following formula:
where $\varepsilon_0$ is strain at the maximum stress. For the special case when the secant modulus at ultimate is twice the initial tangent modulus $E$, Desayi, et al. suggest the following equation:

$$\frac{\sigma}{f_c} = \frac{\varepsilon}{\varepsilon_0} \left( 2 - \frac{\varepsilon}{\varepsilon_0} \right) \quad (2.1)$$

Both curves are shown in Fig. 2.2.

Under biaxial compression, the strength of concrete is expected to be higher since any compression in the other direction confines the concrete and slows the growth of microcracks. This increase in strength is reported by many investigators, but the results published deviate from each other considerably. Kupfer, Hilsdorf, and Rüsch pointed out that the discrepancy was due to the confinement exerted by the testing platen on the specimen's sides. To eliminate this undesirable effect, Kupfer, et al. used a steel brush-like platen that offers no resistance to lateral expansion or contraction of the specimen. Their tests showed that an increase in compressive strength of $0.27 f'_c$ occurred when the ratio of principal stresses was $\sigma_2/\sigma_1 = -0.5/-1.0$. For a ratio of $\sigma_2/\sigma_1 = -1.0/-1.0$ the increase was on the order of $0.16 f'_c$. Some of their experimental results are presented in Fig. 2.3. In a recent publication, Liu, et al. confirmed the Kupfer, Hilsdorf, and Rüsch findings and suggested the following formula for the stress-strain relationship of concrete under biaxial compression:
\[
\sigma = \frac{E\varepsilon}{(1 - \nu_{\alpha}) \left[ 1 + \left( 1 - \frac{1}{\nu_{\alpha}} \frac{E}{E_{sc}} - 2 \right) \left( \frac{\varepsilon}{\varepsilon_0} \right) + \left( \frac{\varepsilon}{\varepsilon_0} \right)^2 \right]}
\]

(2.3)

where \( \alpha \) is the ratio of principal stresses \( \sigma_2/\sigma_1 \) and \( E_{sc} \) is the secant modulus at ultimate load.

The tensile strength of concrete comprises approximately 10 percent of its uniaxial compressive strength. Until recently, there was no standard testing procedure for concrete in tension; probably because it is very difficult to perform a pulling test on a concrete specimen while maintaining the required uniform stress distribution in it. As shown in Fig. 2.1, under tension concrete behaves as a brittle material. Cracks form perpendicular to the load axis and very few deformations occur at the critical section before a cleavage failure takes place.

The split cylinder test is becoming more popular in spite of the fact that the stress distribution in this test is biaxial rather than uniform. Hilsdorf\(^{20, 21}\) was able to obtain uniform stress distribution while studying the tensile strength of 24 in. long prismatic concrete specimens. He used a special test set up in such a way that any strain eccentricities that may develop during the test could be corrected by an appropriate stress eccentricity generated through the loading frame. Figure 2.4 shows some of his findings. According to this diagram the stress-strain curve for concrete in tension may have an unloading portion similar to its behavior in compression. However, the strains plotted in Fig. 2.4 correspond to the average deformation of concrete at the extreme fiber.
over a gage length of 16 inches. Figure 2.4 confirms the fact that the modulus of rupture of concrete is at least 1.5 times its tensile strength. A specimen which is eccentrically loaded such that one extreme fiber strain is kept zero falls in between the uniform and the pure flexural stress distributions. Figure 2.4 is important for building a material model of concrete to be used in the finite element analysis. The required model should reflect the concrete behavior on a macroscopic level rather than on a microscopic one. This idealization is shown in Fig. 2.1 by an unloading portion that extends up to a strain of $\varepsilon_{ut}$. The value of $\varepsilon_{ut}$ in theory depends on the finite element mesh size; the coarser the mesh the higher the value of $\varepsilon_{ut}$. Values of 1 to 5 times the cracking strain could be used for $\varepsilon_{ut}$. This point will be discussed further in conjunction with cracking of reinforced concrete members.

It is of importance to relate the tensile strength of concrete to its compressive strength as obtained from the standard cylinder test. The following relation has been proposed:

$$f'_t = k\sqrt{f'_c}$$  \hspace{1cm} (2.4)

where $f'_c$ is the standard compressive strength in psi, $f'_t$ is the tensile strength in psi, and $k$ is a constant. For $k$ values of 4.0 to 5.0, the tensile strength as obtained from Eq. 2.4 correlates closely with that predicted by the split cylinder test (refer to Fig. 2.5).
2.3 Reinforced Concrete

Figure 2.6(a) shows a typical reinforced concrete test beam without web reinforcement. The part of the beam between the loads is subjected to a pure bending moment. If the beam is loaded gradually, the stress in the lower concrete fiber will exceed the tensile strength of the concrete. A vertical crack will form in the weakest spot in that zone of pure flexure. This crack, called a flexural crack, will propagate vertically towards the neutral axis of the beam. Upon formation of this crack, the stress in the concrete in the vicinity of the crack will drop to zero, whereas stress and hence strain in the reinforcing steel will increase so that equilibrium with the compressive force in the top portion will be preserved. The stress elsewhere in the reinforcement will maintain its value before cracking. Since the reinforcement is bonded to concrete, the stress in the concrete builds gradually from zero at the crack location to a value compatible with the strain at the load level. If the load is increased slightly, another flexural crack will form at a neighboring weak point. The stress distribution in concrete and steel between two adjacent cracks is shown schematically in Fig. 2.6(b).

The presence of cracks with finite widths at the level of the reinforcement makes slip between concrete and steel inevitable. This phenomenon is called bond-slip. Upon increasing the load further, a new crack may form somewhere between two existing cracks when the tensile strength is exceeded. This process will continue until the stress in the concrete near the reinforcement is negligible. This peculiar behavior
is represented by an unloading portion in the idealized stress-strain curve of concrete in tension shown in Fig. 2.1.

If failure does not occur prematurely somewhere else in the beam, then a flexural failure will occur either by the crushing of the concrete (brittle failure), or by the fracturing of the tensile reinforcement (ductile failure).

Between each support and the nearest concentrated load in the beam of Fig. 2.6(a) is a region of combined shear and bending moment. In this region, a 'diagonal tension' crack, inclined at 45° to the beam axis, may form near the mid-depth of the beam. Alternatively, an existing flexural crack will start curving slightly towards the applied load as the load on the beam increases. These inclined cracks may cause the beam to fail before its full flexural strength is reached. Moreover, they limit the ductility of the beam. This type of failure is termed 'shear failure' and may occur in one of the following three ways:

1 - The inclined crack may shear its way through the compression region causing the beam to separate into two pieces joined together by the tensile reinforcement. This type of failure is called 'diagonal tension failure.'

2 - If the concrete above the tip of the curved crack fails in compression, the failure is termed 'shear-compression'. Alternatively, if a secondary crack is initiated along the tensile reinforcement, the failure mode is called 'shear-tension.'
3 - In deep members, the diagonal tension crack causes a tied arch to be formed with the tensile reinforcement acting as a tie. This arch carries external loads by direct stresses to the supports with little or no shear at all. Failure occurs either by crushing of the concrete arch rib or by anchorage failure of the tensile reinforcement.

It is apparent from the study of failure modes in shear that the method of shear transfer in various parts of a member influences the type of failure; hence the behavior of the member. Our knowledge of the mechanism in which shear is transmitted from one plane to another is relatively recent, therefore, no quantitative evaluation of different shear modes is yet available. The basic modes of shear transfer are: 1) shear stress in the uncracked concrete; 2) aggregate interlock; 3) dowel action; 4) shear reinforcement. Figure 2.6(c) illustrates these modes in a beam with diagonal tension crack.

The most important type of shear transfer is aggregate interlock between the two faces of a diagonal tension crack. Because shear must be transmitted through the crack, relative movement between its two sides may occur and shear stresses may initiate over the rough surface provided that the crack width remains small. In their simulation of this type of shear transfer, Fenwick and Paulay found that about 60 percent of the shear stress is carried in the form of aggregate interlock. In another test, they observed that about 20 percent is carried by dowel action, $V_d$ in Fig. 2.6(c), of the longitudinal reinforcement when it crosses a crack. This mode of shear transfer is activated when the steel bar crossing the crack resists shear deformations that may occur, thus
producing tension in the surrounding concrete. If this tension exceeds the tensile strength of the concrete, then splitting will occur along the steel reinforcement causing this mode of shear transfer to be reduced considerably.

In summary, reinforced concrete beams may fail in flexure or in shear according to their span-to-depth ratios. Diagonal tension cracks may cause premature failure which is usually brittle and fatal. No quantitative description of the shear transfer modes is yet available but a thorough understanding of these modes is necessary for the formulation of a finite element material model.

2.4 Failure Criteria for Concrete

The maximum stress or strain failure criterion has been used extensively for the prediction of cracking and the ultimate strength of flexural members, where concrete fibers are stressed in one direction with little or no shear. According to this criterion, failure occurs if any of the principal stresses or strains exceeds a certain limiting value. In essence, this failure criterion ignores stresses in other directions, which is contrary to tests performed on concrete specimens subjected to different stress combinations. Another failure criterion suitable for brittle materials is the Mohr failure criterion, which may be expressed in the following simple form (for triaxial compression):

\[
\frac{1}{2} (\sigma_1 - \sigma_3) = f\left(\frac{\sigma_1 + \sigma_3}{2}\right) \quad \text{for} \quad \sigma_1 > \sigma_2 > \sigma_3.
\]
Richart, Brandzaeg, and Brown suggested the following form for Mohr's failure criterion (triaxial compression)

\[ \sigma_1 = f_c' + 4\sigma_3 \quad \text{if } \sigma_1 > \sigma_2 = \sigma_3 \]

The main criticism of this failure criterion is that it ignores the effect of the intermediate stress \( \sigma_2 \). However, generally it is considered reliable despite its simple form.

The strength of concrete under combined shear and direct stresses may be predicted closely by the octahedral shear stress failure criterion. This criterion relates the octahedral shear stress to the mean stress at failure

\[ \tau_{\text{oct}} = f(\sigma_m) \]

where

\[ \sigma_m = \frac{1}{3} (\sigma_x + \sigma_y + \sigma_z) \]

\[ \tau_{\text{oct}} = \frac{1}{3} \left[ (\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2) \right]^{1/2} \]

It can easily be seen that this failure criterion considers stresses in all directions and unlike the Mohr's criterion, it can be represented by a smooth surface in the stress space.

All problems considered in this investigation can be described as plane stress problems. Therefore, only failure criteria for concrete under biaxial stresses will be discussed. Kupfer, Hilsdorf, and Rüssch
and more recently Liu, Nilson, and Slate have obtained a failure envelope (Fig. 2.7) based on their extensive tests on plain concrete under different ratios of biaxial stresses. This failure envelope shows clearly the confining effect on the compressive strength of concrete as discussed in Section 2.2 above. It also shows that the tensile strength of concrete in one direction is not affected by the presence of tension in the other direction. However, the compressive strength in one direction is considerably reduced if a small tensile stress is encountered in the other direction, a case found in compression regions in the presence of shear stresses. Shown in Fig. 2.7 is a dashed line that represents the failure envelope based on Mohr's failure criterion. It is evident that this failure envelope is generally conservative, especially in the biaxial compression quadrant.

Mikkola and Schnobrich obtained close agreement with the experimental results of Kupfer, et al. by using the octahedral shear stress failure criterion. Two linear expressions of the form

\[ \tau_{\text{oct}} = a - b \sigma_m \]  

(2.5)

were used; where a and b are material constants. Equation 2.5 represents two expressions; one is valid for biaxial compression, while the other is valid for biaxial tension and tension-compression regions. If the constants a and b are evaluated in terms of concrete strength in tension and in compression \( f'_t \) and \( f'_c \), respectively, then Eq. 2.5 yields the following two expressions:
\[
\tau_{\text{oct}} + \sqrt{2} \left( \frac{1 - \alpha}{1 + \alpha} \right) \sigma_m - \frac{2\sqrt{2}}{3} \frac{\alpha}{1 + \alpha} f'_c = 0 \quad (\sigma_1 > 0) \quad (2.6)
\]

\[
\tau_{\text{oct}} + \sqrt{2} \left( \frac{\beta - \frac{1}{2}}{2\beta - \frac{1}{2}} \right) \sigma_m - \frac{\sqrt{2}}{3} \frac{\beta}{2\beta - \frac{1}{2}} f'_c = 0 \quad (\sigma_1 < 0) \quad (2.7)
\]

where \(\alpha\) and \(\beta\) values are given in Ref. 30 as

\[
\alpha = \frac{f'_t}{f'_c} \approx 0.10 \quad \beta = \frac{\sigma_1}{f'_c} = \frac{\sigma_2}{f'_c} \approx 1.16 \quad (2.8)
\]

Equations 2.6 and 2.7 have a discontinuity at points \((0, f'_c)\) and \((f'_c, 0)\) where either equations could be used. In this study, Eq. 2.6 will be used to indicate the boundary between cracked and uncracked concrete in biaxial tension and tension-compression regions. In the biaxial compression quadrant Eq. 2.7 will be used as a yield criterion which sets the boundary between the elastic and yielded concrete. Yielded concrete will crush if the equivalent plastic strain \(\bar{\varepsilon}_p\), as given in Eq. A.8 in Appendix A, exceeds the ultimate compressive strain \(\varepsilon_{uc}\).

2.5 Stress-Strain Relations for Concrete

2.5.1 Elastic Properties

Despite all the complications connected with concrete behavior in the nonlinear range, uncracked concrete could be considered as a linear isotropic homogeneous material. The elastic stress-strain relations under plane stress are:


\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix} = \frac{E}{1 - \nu^2} \begin{bmatrix}
1 & \nu & 0 \\
\nu & 1 & 0 \\
0 & 0 & \frac{1 - \nu}{2}
\end{bmatrix} \begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix}
\]

(2.9)

where \(\nu\) is Poisson's ratio with a value between 0.15 to 0.20 and \(E\) is the modulus of elasticity which is best determined from the experimental results. The ACI code recommends the following formula

\[E = 33 \, \omega^{1.5} \, f_c^{0.5}\]

(2.10)

where \(\omega\) is the unit weight of concrete in pounds per cubic foot, and \(f_c\) is the standard cylinder strength in pounds per square inch.

### 2.5.2 Concrete Cracked in One Direction

Concrete cracks according to the criterion established in Sec. 2.4 above. Usually, cracks occur perpendicular to the direction of maximum tensile stress, except in equal biaxial tension where no preferred direction exists. Upon cracking, reinforced concrete carries shear stresses according to one or more of the shear mechanisms discussed in Sec. 2.3. In the proposed material model, a cracked concrete element is considered to be made up of several concrete bars parallel to the crack direction, Fig. 2.8. These bars continue to carry tensile or compressive stresses along their axes. It is assumed that shear stresses are mobilized along the sides of these hypothetical bars when concrete cracks. However, it is reasonable to presume that their shear carrying capacity is reduced. This reflects the reduction in the aggregate interlock shear carrying
capacity with the opening of cracks. Elastic concrete transmits shear through elastic shear stresses. Its shear strength is measured by the shear modulus, \( G \).

\[
G = \frac{E}{2(1 + \nu)}
\]  

(2.11)

For a cracked concrete finite element, the reduction in its shear capacity is achieved by using a reduced shear modulus, \( \mu G \). The merits of this shear reduction factor, \( \mu \), will be investigated later.

Finally, the fact that the average tensile stress in a newly cracked element is not zero is implemented by assuming an unloading portion for concrete in tension, Fig. 2.1. However, the average tensile stress will diminish slowly as more cracks are introduced in the element. Theoretically, the length of this unloading portion will approach zero when the element size approaches zero. The effect of the length of the unloading portion will be studied later. Unfortunately, the proposed hypothetical unloading tail in tension implies that the material is unstable and has a negative modulus. A negative stiffness creates numerical difficulties when solving the resulting simultaneous equations in the finite element method. To bypass this difficulty, the tensile stress in a cracked element is treated as an additional stress that is released gradually during the iterative solution of the equations. How fast this stress should be released depends on how steep the unloading portion is. In summary, this artificial unloading portion will not be considered in the constitutive relations for cracked concrete, but will be accounted for during the iterative solution.
In mathematical form, the stress-strain relations for cracked concrete in the principal stress coordinate system \((x, y)\), rotated by an angle \(\theta\) from the original coordinate axes \((x, y)\), Fig. 2.8, can be written as

\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix} =
\begin{bmatrix}
0 & 0 & 0 \\
0 & E & 0 \\
0 & 0 & \mu G
\end{bmatrix}
\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix}
\] (2.12)

or in matrix form

\[
\{\sigma\} = [D] \{\varepsilon\}
\] (2.13)

The stress-strain relation for cracked concrete in the \(x-y\) coordinate system is

\[
\{\sigma\} = [D_c] \{\varepsilon\}
\] (2.14)

where \([D_c]\) can be obtained using an energy approach. By equating the energy stored in the element in the two coordinate systems, we obtain

\[
\{\varepsilon\}^T \{\sigma\} = \{\varepsilon\}^T \{\sigma\}
\] (2.15)

But the strains \(\{\varepsilon\}\) and \(\{\varepsilon\}\) are related through the transformation matrix, \([T_e]\), as

\[
\{\varepsilon\} = [T_e] \{\varepsilon\}
\] (2.16)

where \([T_e]\) is given by
\[
[T_\varepsilon] = \begin{bmatrix}
\cos^2\theta & \sin^2\theta & \cos\theta \sin\theta \\
\sin^2\theta & \cos^2\theta & -\cos\theta \sin\theta \\
-2\cos\theta \sin\theta & 2\cos\theta \sin\theta & \cos^2\theta - \sin^2\theta
\end{bmatrix}
\]  

Substitution of Eqs. 2.13, 2.14 and 2.16 in Eq. 2.15 yields
\[
\{\varepsilon\}^T [D_c] \{\varepsilon\} = \{\varepsilon\}^T [T_\varepsilon]^T [D] [T_\varepsilon] \{\varepsilon\}
\]

From the above expression, the required material matrix is given by
\[
[D_c] = [T_\varepsilon]^T [D] [T_\varepsilon]
\]  

2.5.3 Concrete Cracked in Two Directions

After concrete cracks in one direction, the direction of crack, \(\theta\), is fixed. If tensile stresses exist in the second direction, the maximum stress failure criterion is used to determine whether or not the element has cracked in the second direction. The material matrix for an element cracked in two directions in the \((x, y)\) coordinate axes can be written as
\[
\begin{bmatrix}
-\sigma_x \\
-\sigma_y \\
-\tau_{xy}
\end{bmatrix} =
\begin{bmatrix}
0 & 0 & 0 \\
0 & 0 & 0 \\
0 & 0 & \mu G
\end{bmatrix}
\begin{bmatrix}
-\varepsilon_x \\
-\varepsilon_y \\
-\gamma_{xy}
\end{bmatrix}
\]  

Using the transformation expressed in Eq. 2.18, the material properties in the original coordinate system can be obtained easily.
2.5.4 Plastic Stiffness

Concrete is idealized as an elastic-perfectly plastic material in biaxial compression, Fig. 2.1. The octahedral shearing stress yield criterion, Eq. 2.7, will be used to relate the behavior under biaxial stresses to that of uniaxial compression as expressed by the uniaxial stress-strain relations. Using the incremental plasticity theory, the stress-strain relations in the plastic range can be obtained as discussed in Appendix A. For a material with no strain hardening, the incremental stress-strain relations in the plastic range can be written as

\[ d\{\sigma\} = [D]_{ep} \, d\{\varepsilon\} \quad (2.20) \]

and

\[ [D]_{ep} = [D] - [D] \{N\} \{N\}^T [D] \{N\} \quad (2.21) \]

where \([D]\) is the elastic material property matrix and \({N}\) is a normal to the yield surface. The components of \({N}\), for the case of octahedral shear stress yield surface, are given in Eqs. A.13 and A.16 in Appendix A.

If an element cracked in one direction is subjected to compression in another direction, the element may yield in that direction. The incremental elasto-plastic material matrix in this case is,

\[ d\{\sigma\} = [0] \, d\{\varepsilon\} \quad (2.22) \]

where \([0]\) is a null matrix.
Finally, if the equivalent plastic strain in a plastic element exceeds the ultimate concrete strain in compression, then the element will crush. The stresses in the element drop to zero, and the element cannot carry any load.

2.6 Stress-Strain Relations for the Steel Reinforcement

The behavior of the reinforcing steel is idealized as an elastic-perfectly plastic material, Fig. 2.9. Two kinds of idealization are used in the finite element analysis. One is a bar element for which the stress-strain relations are obtained directly from the stress-strain diagram in Fig. 2.9. The second is a two-dimensional finite element. The elastic behavior of this element is the same as that given previously in Eq. 2.9 with $E$ referring to the modulus of elasticity of steel. In Fig. 2.9, $\bar{\sigma}_y$ and $\bar{\varepsilon}_y$ are the yield stress and yield strain of steel, respectively. In the plastic range, however, the elasto-plastic material property matrix is calculated from Eq. A.12 of Appendix A. The von Mises yield criterion, widely used for steel, will be adopted in this study (see Fig. 2.10). The components of the normal to this yield surface are given in Eq. A.13 and A.14 of Appendix A.
30

CHAPTER 3

IDEALIZATION OF STRUCTURES BY FINITE ELEMENTS

3.1 General

Because of its adaptability to computer programming and its versatility for handling various loadings and boundary conditions, the finite element method has been used extensively for the solution of engineering problems. Although the method was first developed for structural analysis, the general nature of the theory behind it has made its application to other branches of engineering possible. The finite element method is a numerical procedure in which the partial differential equations of the continuum are converted into a system of linear algebraic simultaneous equations, replacing integrations by finite summations. The continuum with an infinite number of degrees of freedom is thus simulated by a discrete model with a finite number of degrees of freedom.

Many authors have suggested that Courant was the first to use the 'spirit' of the finite element method in his investigation of the torsion problem in 1943. However, it was not until 1956 that the formal application of the finite element concept, as known today, came into being when the famous paper by Turner, Clough, Martin, and Topp was published. The first formulation of the method for use with elastic structural analysis was based on a direct stiffness approach. This approach had
the conceptual limitation of employing simple elements only. As the method developed, it was realized that it could be regarded as an application of the variational principles of structural mechanics, especially of the principle of minimum potential energy. Such a formulation enhanced the finite element method by offering a greater flexibility in the formulation of element stiffness matrices, and freed investigators from attaching a physical meaning to the generalized displacements. Furthermore, it placed the method on a sound theoretical foundation and broadened its scope to include non-structural problems.

With a variety of variational principles available to researchers, new finite element models were developed. In addition to the 'displacement model' based on the principle of minimum potential energy, there are the 'equilibrium model' first proposed by Fraeis de Veubeke and the so-called 'hybrid model' first developed by Pian. However, among these models the displacement method is the most widely used in structural mechanics and has been utilized in this study.

In the finite element method the continuum is viewed as a collection of a finite number of elements. Attention is then focused on one of these elements, whose displacement field is described uniquely in terms of the displacement values at selected nodal points. The principles of mechanics are then used to approximate the behavior of this typical piece of the body. A number of these elements are fitted together at appropriate nodes to make up a discretized model for the body, and its overall behavior is thus represented by a resulting set of linear
or nonlinear algebraic equations depending on the type of the continuum's behavior to be investigated. The choice of the element type should be made with care as there are many families of elements for each class of problems.

The shape functions, also known as interpolation functions, are used to approximate the displacement field in the element. If the same shape functions are used to represent both the displacement field and the element geometry, then the element is called 'isoparametric'. The formulation of displacement models and the computation of element stiffnesses have been both simplified and generalized by the concept of the isoparametric elements. This family of elements, first proposed by Zienkiewicz, Irons, and co-workers at the University of Wales, Swansea, gives, in general, quite accurate results, and can approximate very closely many complicated boundaries or surfaces.

In summary, the finite element method frees the structural analyst from the restrictions and complications of geometry and boundary conditions encountered in the solution of boundary value problems. Moreover, solutions for different loadings or different material properties can be obtained easily by changing only a few input parameters. However, there are a number of qualitative checks which have to be performed to insure convergence to the true solution as the number of elements is increased, namely, functional completeness, compatibility, and the 'patch test'. The first condition states that the displacement field should include rigid body modes as well as all pertinent constant strain states.
Full 'functional completeness' of the displacement field satisfies inter-element boundary compatibility. The 'patch test' provides a necessary condition for convergence in elements with incompatible modes (as discussed in Section 3.4), elements which are integrated approximately, or elements that have no clear physical basis.

3.2 The Displacement Model

The steps followed in the formulation of the finite element displacement analysis can be found in any standard text on finite elements. However, for the sake of completeness a description of the formulation procedure follows.

The basic step in any finite element analysis is to define the displacement field \( \{ u \} \) in each element in terms of several parameters \( \{ u \} \) associated generally with the displacements at the nodal points

\[
\{ u \} = [N] \{ \bar{u} \} \tag{3.1}
\]

where \([N]\) is a matrix of shape functions. With the displacement field defined within the element, the strain-displacement relations in the element are obtained from

\[
\{ \varepsilon \} = [B] \{ \bar{u} \} \tag{3.2}
\]

where the \([B]\) matrix is obtained by proper differentiation of the shape functions. For linear elasticity, stresses \( \{ \sigma \} \), are related to strains \( \{ \varepsilon \} \), through a constitutive law expressed in the \([D]\) matrix. Thus

\[
\{ \sigma \} = [D] \{ \varepsilon \} \tag{3.3}
\]
Assuming for the present discussion that the forces acting on an element are distributed forces only the potential of external forces is written as

\[ W_p = \int_s \{u\}^T \{p\} \, ds \]

The strain energy stored in the element is the integral of internal work (work done by internal stresses), or

\[ U_o = \int_v \frac{1}{2} \{\epsilon\}^T [\sigma] \, dv = \frac{1}{2} \int_v \{\epsilon\}^T [D] \{\epsilon\} \, dv \]

The total potential energy of the element, \( \Pi \), is the sum of its strain energy and the potential energy of the applied loads. Thus

\[ \Pi = \frac{1}{2} \int_v \{\epsilon\}^T [D] \{\epsilon\} \, dv - \int_s \{u\}^T \{p\} \, ds \quad (3.4) \]

Substitution of Eqs. 3.1 and 3.2 into Eq. 3.4 gives

\[ \Pi = \frac{1}{2} \int_v \{\overline{u}\}^T [B]^T [D] [B] \{\overline{u}\} - \int_s \{\overline{u}\}^T [N]^T \{p\} \, ds \]

For equilibrium to be ensured, the total potential energy must be stationary for variations of admissible nodal displacements. In other words, the first variation of the potential energy should be zero.

This gives

\[ \delta \{\overline{u}\}^T (\int_v [B]^T [D] [B] \, dv \{\overline{u}\} - \int_s [N]^T \{p\} \, ds) = 0 \]
Since the variation of the nodal displacements, $\delta \{u\}$, is arbitrary, the expression between the parentheses must vanish. This gives the desired equilibrium equation for the element

$$[k] \{\ddot{u}\} = \{f\}$$

where the stiffness matrix, $[k]$, and the nodal load vector, $\{f\}$, are defined as

$$[k] = \int_Y [B]^T [D] [B] \, dv$$

and

$$\{f\} = \int_S [N]^T \{p\} \, ds$$

The vector $\{f\}$ is sometimes referred to as the consistent load vector or the generalized load vector. The force-displacement relations for the overall structure are obtained by the proper summation of element stiffnesses. After accounting for the boundary conditions, one obtains

$$\{F\} = [K] \{U\}$$

where $\{F\}$ and $\{U\}$ are the generalized nodal forces and nodal displacements, respectively, and $[K]$ is an $n \times n$ matrix; $n$ being the total number of nodal parameters.

The matrix $[K]$ is the stiffness matrix of the structure. This matrix is characterized by being symmetric, positive definite, banded, and sparsely populated. Equation 3.8 represents a set of linear algebraic
simultaneous equations, the solution of which gives the generalized displace-
ments, \( \{U\} \). Efficient algorithms which make use of the above-
mentioned characteristics of the stiffness matrix, \([K]\), allow the solution of a large number of equations with the least number of computations. Once the generalized displacements are obtained, displacements, strains, and stresses may be computed from Eqs. 3.1, 3.2, and 3.3, respectively.

3.3 The Finite Elements Used in This Study

A major factor to be considered in any nonlinear analysis is economy. The cost of using higher order elements or a very fine mesh is prohibitive even with the fastest available computers. This is why the constant strain triangular element (CST) was used extensively in many nonlinear problems despite its unfavorable characteristics. In addition to its directionality, the CST is much too stiff in bending. Hence, a fairly fine mesh is needed to obtain a solution with reasonable accuracy. This results in preparation and checking of additional input data.

With regard to the cracking analysis of reinforced concrete beam-like structures, this element fails to reproduce steep stress gradients which exist in the compression zone at higher load levels, resulting in stiff behavior of the structure. Finally, CST elements retain a large amount of energy in the form of shear strain energy thereby causing a delay in yielding of the reinforcement.\(^{37}\)

The use of the linear isoparametric element provides a suitable compromise between the CST and higher order elements. Cracking or other
nonlinearities can be monitored at the element's four integration points, allowing partial cracking or yielding of the element according to the stress level at the integration points.

In this investigation, three element types will be used: (1) a linear isoparametric quadrilateral element which idealizes concrete and/or steel, (2) a one-dimensional bar element which idealizes steel reinforcement, and (3) a special 'tie-link' element which simulates the bond-slip phenomenon in reinforced concrete. The use of a one- or a two-dimensional steel element depends on whether dowel action of the longitudinal reinforcement is to be considered. The formulation of stiffness matrices for the foregoing elements is presented in Sections 3.4 through 3.7, and their detailed formulations are given in Appendix B.

3.4 The Linear Isoparametric Quadrilateral Element

3.4.1 Basic Formulation

The derivation of the shape functions for this element is quite simple. Figure 3.1(a) shows such a quadrilateral element with its local or 'natural coordinates'. The local coordinate system allows the identification of a point within the element by a set of dimensionless numbers whose magnitude varies from 1 to -1. A transformation of coordinates between the local coordinates and the global cartesian set, Fig. 3.1(b), is obtained by using interpolation or shape functions, \( N_i(\xi, \eta) \). A shape function is defined such that its value is unity at one nodal point, and zero at all other nodal points. Such a coordinate transformation can be written in the following form:
\begin{align*}
\mathbf{x}(\xi, \eta) &= \sum_{i=1}^{4} N_i \bar{x}_i, \quad \mathbf{y}(\xi, \eta) = \sum_{i=1}^{4} N_i \bar{y}_i
\end{align*}

(3.9)

where \( \bar{x}_i, \bar{y}_i \) are the global coordinates at the nodal points. For the linear isoparametric element, the shape functions, \( N_i \), are easily obtained as

\begin{align*}
N_i = \frac{1}{4} \left( (1 + \xi \xi_i) (1 + \eta \eta_i) \right)
\end{align*}

(3.10)

where \( \xi_i, \eta_i \) are the natural coordinates at the nodal points.

As the word 'isoparametric' indicates, the same shape functions are used to describe the displacement field within the element, thus

\begin{align*}
\mathbf{u}(\xi, \eta) &= \sum_{i=1}^{4} N_i \bar{u}_i, \quad \mathbf{v}(\xi, \eta) = \sum_{i=1}^{4} N_i \bar{v}_i
\end{align*}

(3.11)

where \( \bar{u}_i, \bar{v}_i \) are nodal point displacements in the \( x \) and \( y \) directions, respectively.

It can be shown easily that the selected interpolation functions satisfy the requirements for convergence. Firstly, it is apparent that the displacements along a common side between two adjacent elements depend only on the nodal displacements at the two ends of this side. Hence, interelement compatibility is satisfied. Secondly, it is possible to select a combination of nodal displacements which causes all points on the element to experience the same displacement. In fact, this occurs when all nodal displacements in the element are the same. Therefore, rigid body modes are contained in the model.
The steps outlined in Section 3.2 may be followed to obtain the stiffness matrix of the isoparametric element. The details of the derivation are presented in Section 3 of Appendix B.

3.4.2 The Incompatible Modes

The linear isoparametric element is capable of reproducing displacements due to direct stresses. However, when it is used in problems in which the bending behavior is important, the convergence to the correct solution is obtained only with a very fine mesh. To illustrate this, consider an element subjected to pure bending, Fig. 3.2(a). Since the displacement in the y-direction in the element is linear, the distorted shape of the element, Fig. 3.2(b), cannot match the true deflected shape, Fig. 3.2(c). Consequently, the shear strains are not zero over most of the element which makes the element much too stiff for this type of loading. For a prismatic bar, subjected to end moments, M, Fig. 3.2(a), the exact solution can be written as:

\[ u = \frac{M}{EI} xy \]  \hspace{1cm} (3.12a)

and

\[ v = \frac{M}{2EI} (a^2 - x^2) + \frac{WM}{2EI} (b^2 - y^2) \]  \hspace{1cm} (3.12b)

where E is Young's modulus and I is the moment of inertia of the prismatic bar. The first displacement, u, is reproduced exactly by
the element, Fig. 3.2(c). However, as Eq. 3.12(b) indicates, the displacement $v$ changes quadratically with both $x$ and $y$, a configuration the linear element cannot reproduce. It is seen from Eqs. 3.12, that the exact solution satisfies the pure bending condition of zero shear strain everywhere, or

$$\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = 0$$

It can, therefore, be concluded that the error is in the same form of Eq. 3.12(b) and can be written as

$$v = \beta_1(1 - \xi^2) + \beta_2(1 - \eta^2) \quad (3.13)$$

Aware of this deficiency in the behavior of lower order isoparametric elements, Wilson, et al. added two quadratic shape functions to the basic shape functions in Eq. 3.10. The added functions, Fig. 3.3, have zero values at the corner nodes and vary quadratically over the element,

$$N_5 = (1 - \xi^2) \quad (3.14)$$

$$N_6 = (1 - \eta^2)$$

With this addition Eq. 3.11 becomes

$$u = \sum_{i=1}^{6} N_i u_i, \quad v = \sum_{i=1}^{6} N_i v_i \quad (3.15)$$
where \( u_5 \) and \( u_6 \), for example, need not represent displacements of any physical nodes. They can be viewed as mode amplitudes or simply as Lagrangian multipliers. Their magnitudes are selected by requiring that the total strain energy of the element be a minimum.

The only drawback to this procedure is that displacements along common edges of two adjacent elements are no longer compatible since the displacements along one common edge are not solely dependent on the displacement of the terminal nodes; hence the name 'incompatible modes'. With the loss of inter-element compatibility, convergence of finite element results to the true solution is not assured. However, several investigators reported satisfactory results with models that do not strictly meet the compatibility requirements. Irons and Razzaque have shown that for the case of elements with incompatible modes, the patch test provides a necessary condition for convergence. Such a test shows correct behavior for trapezoidal and rectangular patterns.

Using the new displacement field, Eq. 3.15, the stiffness matrix will have 12 rows instead of the basic 8 rows encountered in the compatible model. This enlarged matrix could be partitioned and, by condensing out the additional unknowns, the stiffness matrix will again have its usual size. Bakhrebah and Schnobrich, using two distinct theoretical tests, have demonstrated the superiority of the incompatible element over the compatible one. Wislon, et al. used a cantilever beam subjected to end loads as a numerical example to show the tremendous improvement that occurs with introducing incompatible modes at the cost of a few extra calculations.
3.5 The Triangular Element

Triangular elements are sometimes needed to idealize certain structured boundaries such as haunches and curved edges. Furthermore, these elements are used in the transition zone between a fine mesh of elements and a coarse one. While the stiffness matrix for such elements may be computed with the procedure outlined in Section 3.2, an alternate method was used in this study. The method consists of degenerating the linear quadrilateral element of Section 3.3 into a triangular element. This is accomplished by specifying that two corners of the quadrilateral element have identical coordinates. Computationally this is possible because the linear quadrilateral element used in this study is numerically integrated and the four integration points of the degenerate element are still distinct. No changes are necessary in the computer program.

While this approach saves the time needed to formulate a new element, it should be pointed out that there is no physical meaning associated with using incompatible modes in the case of a triangular element. Therefore, the use of incompatible modes is dropped for this case. Consequently, the behavior of a triangular element is inferior to the original quadrilateral one. The 'mixability' of this element (its behavior in a batch of quadrilateral elements) has been found to be adequate for simple tension and compression in a patch test performed in this study.
3.6 The Tie-Link Element

The tie-link element is used for simulating the bond-slip action between the reinforcing steel and the surrounding concrete as well as for tying two nodes together so that they have the same deformation. As mentioned previously, the steel reinforcement is idealized by either a bar element or a plane isoparametric quadrilateral element. Idealization with bar elements is suitable for shallow beams where dowel action is not noticeable. However, in deep beams where the longitudinal reinforcement contributes to the shear carrying capacity, a two-dimensional element is used for steel. The link element provides a linkage between steel and concrete in either case. It consists of two springs with spring constants $k_h$ and $k_v$, perpendicular to each other. The springs may be inclined by an angle $\theta$ with the global coordinates, Fig. 3.4. This element usually connects two joints which have the same coordinates, one belonging to the reinforcement and the other to the concrete. By properly selecting the stiffness of a spring in a tie-link element, slip in the direction of that spring may or may not be permitted. Generally it is assumed that concrete and steel have full bond in a direction perpendicular to the steel reinforcement; therefore, a very large stiffness is inputed to represent the link stiffness in that direction.

The detailed derivation of the stiffness matrix for this element is presented in Section 4 of Appendix B.
3.7 The Bar Element

The bar element is used to idealize both longitudinal and shear reinforcements in a reinforced concrete structure. The stiffness matrix for this element is given in Appendix B.
4.1 General

In the finite element displacement model the resultant equilibrium equations contain a vector of nodal loads. For other loading conditions, such as distributed loads or the self weight of the structure, the loads are converted into equivalent nodal loads or generalized loads before the solution process is carried out. The generalized loads are obtained by equating the work done by the two loads (the applied loads and their equivalent nodal loads) through a certain configuration of the element.

For a non-nodal loading distribution, the equivalent nodal loads may be evaluated from the following equation

\[ \{f\} = - \int_V \{N\}^T \{p\} \, dv \]  

For the case of a pressure on side I-J of a linear isoparametric element (Fig. 4.1), the integral in the above equation reduces to a line integral as follows:

\[ \{f\} = -t \int_{-1}^{+1} [N(\xi, \eta = -1)]^T \{\bar{p}\} \, ds \]  

where the line coordinate \( s \) is related to the local coordinate \( \xi \) as

\[ ds = \frac{\xi}{2} \, d\xi \]

with \( \lambda \) being the length of side I-J, and \( t \) is the thickness of the element.
For the case of the self weight of the isoparametric element, Eq. 4.1 becomes

\[
\{f\} = \text{g} \int_{-1}^{+1} \int_{-1}^{+1} [N(\xi, \eta)]^T \det[J] \, d\xi \, d\eta
\]  

(4.3)

where \( g \) is the unit weight of the element material.

Equations 4.2 and 4.3 are usually evaluated during the numerical integration in the computation of the element stiffness matrix. The nodal loads are then added to the vector of generalized loads.

In a displacement finite element model, only boundary displacement conditions can be prescribed. Stress or natural boundary conditions are not satisfied exactly. Usually, a very fine mesh or the use of higher order elements (elements with many degrees of freedom) is needed in order to get better results at locations of high stress concentration.

The generalized stiffness matrix for the whole structure is obtained by assembling the individual elements stiffnesses. This matrix is singular and cannot be inverted. The removal of the singularity requires the introduction of supports at a sufficient number of nodes. To account for the effect of support constraints, the rows and columns in the structural stiffness matrix corresponding to the supports as well as the corresponding rows in the generalized load vector are deleted.

4.2 Solution of Linear Equations

For elastic material, the resultant equilibrium equations are in the form
\[ [K] \{u\} = \{F\} \quad (4.4) \]

where \([K]\) is the non-singular structural stiffness matrix, \(\{U\}\) and \(\{F\}\) are vectors of the nodal displacements and loads, respectively. The size of the stiffness matrix, \([K]\), depends on the number of elements simulating the structure and may be as high as several thousands. Therefore, in solving Eq. 4.4, use of matrix sparsity, symmetry, multiple right hand sides, and banded matrix properties is made to save computational time.

Of all the available algorithms for solving the finite element equilibrium equations, the Gaussian elimination algorithm requires the least amount of arithmetic operations \(^{47}\) as compared to other solution procedures such as Choleski decomposition \(^{48}\) and Gauss-Seidel \(^{47}\) iterative scheme. In a banded Gaussian elimination procedure the manner of numbering the joints in a finite element mesh affects the bandwidth; hence, special care should be taken to keep the bandwidth to a minimum.

An alternative solution procedure is the frontal solution algorithm developed by Melosh and Bamford \(^{48}\), and Irons \(^{49}\). The frontal solution is based on a Gaussian elimination technique and derives its name from the creation of a front that advances through the nodal points. The method proceeds with the elimination, element by element, hence it is independent of nodal numbering. The frontal solution is particularly advantageous for elements with mid-side nodes and three-dimensional elements.

The Gaussian elimination solution for a banded matrix was used in this investigation. The solution algorithm used, as will be discussed
later, does not require the presence of all the equations in the computer memory core during the solution. Therefore, large number of equations can easily be solved.

4.3 Solution of Nonlinear Equations

4.3.1 Review of Available Methods

An important application of the finite element method is in the solution of nonlinear problems. There are two categories of nonlinearity—geometric nonlinearity and material nonlinearity. Geometric nonlinearity is encountered when a structure experiences large deformations. Deformations in reinforced concrete structures are generally small, thus, the geometric nonlinearity was neglected in this study. As discussed earlier, the material model used in this investigation reflects cracking as well as nonlinear behavior of concrete under biaxial state of stress. Therefore, the behavior of a cracked discontinuous medium is approximated by a continuous body having nonlinear material properties.

The finite element formulation with a nonlinear material model results in a set of nonlinear simultaneous equations. Since early finite element computer programs were written for analysis of elastic systems, it is logical that in early investigations nonlinear problems were solved as a series of linear elastic problems. In such a method, Fig. 4.2(a), the load is applied on the structure in increments. The structural stiffness matrix is updated at the beginning of each load step using the material properties at the end of the previous load increment. The accuracy of this method can be improved by using smaller
load steps or by employing a numerical procedure such as Runge-Kutta to obtain a better estimate of the incremental displacements in any load step. In the latter scheme the finite element equilibrium equations are reduced to a set of first order differential equations of the following form:

$$[K(U)] \, d\{U\} - d\{R\} = 0 \quad (4.5)$$

An alternative approach suitable for solving problems of the deformation theory of plasticity, is the Newton-Raphson method. In this method a series of iterations is carried out while the structure is fully loaded (see Fig. 4.2(b)). During each iteration, since the stiffness matrix used is approximate and equilibrium is not satisfied, an unbalanced force is computed and applied on the structure in the next iteration. In addition, a new tangent stiffness matrix is assembled and triangularized in each iteration. The iterations are repeated until equilibrium is satisfied. If the initial stiffness instead of the tangent stiffness is used throughout the iteration process, Fig. 4.2(c), the method is termed 'the modified Newton-Raphson'. The number of iterations in the latter procedure is larger than in the former. Nevertheless, the computational time needed for formulating and triangularizing a new stiffness matrix in each iteration is saved.
The advantages and applicability of the incremental and the iterative procedures for the solution of nonlinear problems are discussed in detail in the book by Desai and Abel. However, it is easily seen that a procedure which combines the two methods has the advantages of both and may be used to solve a wide spectrum of nonlinear problems. Such a mixed incremental-iterative procedure was used in this investigation.

4.3.2 The Incremental-Iterative Procedure

The incremental-iterative procedure for solving nonlinear problems is well suited for the finite element analysis of reinforced concrete structures. Firstly, the problem in this case is described as nonconservative, thus, relatively small load increments should be applied on the structure so that the real 'path' of the load-deflection curve is followed as closely as possible. Secondly, upon applying a new load step on the structure and analyzing it, equilibrium is not generally satisfied because the stiffness matrix used is approximate. Therefore, iterations must be carried out to restore the equilibrium of the structure.

The incremental method is shown graphically in Fig. 4.3. In matrix form, the method may be described as follows:

\[
[K]_i \{\Delta U\}_i^j = \{R\}_i^{j-1}
\]  

(4.6)
where \([K]_i\) is the incremental stiffness matrix for load step \(i\), \(\{\Delta U\}_i^j\) is the incremental displacement vector for load step \(i\) and iteration \(j\), and \(\{R\}_{j-1}\) is the residual load vector computed from the previous iteration, \(j-1\). This residual load vector is caused by the excessive stresses, \(\{\sigma_{ex}\}\), that the element can no longer sustain at the current strain level because of cracking, crushing, or yielding of concrete or steel.

Usually, the iterative procedure is terminated when it is believed that the solution is close to that of the equilibrium state; that is, when convergence is achieved. The criterion for convergence may be based on different quantities such as changes in displacements, residual forces, or changes in material properties in two consecutive iterations. In this study, the norm of the applied incremental load vector, \(\|\{\Delta P\}_i\|\), is compared to the norm of the residual load vector, \(\|\{R\}\|\), during the iterative solution. Convergence is attained if the ratio of the latter to the former is less than a prescribed tolerance.

The incremental stiffness matrix, \([K]_i\), is a tangent stiffness matrix which is assembled and triangularized at the beginning of each load step and it is used to analyze the structure during the iterations for that load step. The mixed iterative procedure is believed to be the most economical of all available procedures for solving nonlinear material problems. Nevertheless, the procedure has a few shortcomings. At higher load levels, several layers of the concrete elements may crack. If cracking is in a direction parallel to one of the coordinate
axes, then the tangent stiffness matrix becomes ill-conditioned. Additionally, in a test of a reinforced concrete beam, a crack of a certain length and width may form suddenly, whereas, in a finite element simulation of the same beam only a few integration points may crack in one iteration. This causes very slow convergence of the solution. To speed up the convergence, several over-relaxation methods have been proposed. According to these methods, the recurrence formula, Eq. 4.6, can be replaced by the following expression

\[
[a]_i [K]_i \{\Delta U\}_i^{j-1} = \{R\}_i^{j-1}
\]  

(4.7)

in which \([a]_i\) is a diagonal matrix of over-relaxation coefficients. A method proposed by Nayak and Zienkiewicz was implemented in the solution process for this study. According to this method, an 'accelerator' factor is computed for each unknown from its incremental values in two consecutive iterations. Even though the matrix \([a]_i\) was updated frequently (every three iterations), it was found that the method worked satisfactorily for plasticity problems only. For cracking problems, the over-relaxation method caused divergence of the solution in more than one instance. Hence, no 'accelerator' factors were used in the study. Instead, the structural stiffness matrix was updated at the beginning of each load step and whenever the number of iterations in one load step had exceeded a prescribed limit. It is believed that by doing so the number of iterations is reduced since using a new tangent stiffness improves the guess on the next incremental displacements.
4.3.3 Outline of the Computational Steps

The following summarizes the computational procedure for a typical load increment:

1. Apply a new load increment on the structure, \( \Delta P \). Store \( \Delta P \) in the residual load vector, \( \{ R \} \).

2. Update the stiffness matrix if needed. Using the Gaussian elimination procedure triangularize the updated stiffness matrix, analyze the structure using the load vector, \( \{ R \} \), and perform a back-substitution on the triangularized matrix to obtain the incremental displacements, \( \Delta U \). Update the displacements.

Do the following for each element and/or for each integration point in the element:

3. Using the incremental displacements and the strain displacement relations, calculate the incremental strains, \( \Delta \varepsilon \), and update the strains.

4. Using the incremental strains and the current material properties (those incorporated in the current stiffness matrix) determine the incremental stresses, \( \Delta \sigma_e \), and add them to the previous stresses to obtain the stress vector, \( \sigma' \).

5. Check this stress state against the applicable transition criteria (criteria for yielding, cracking, or crushing described in Chapter 2). If none is exceeded, proceed to step 10.
6. Based on the present strain and stress levels, calculate a new material property matrix, \( [D_j] \). Determine the stress vector, \( \{\sigma\} \), which the element can sustain at this strain level.

7. Subtract the stress, \( \{\sigma\} \), from \( \{\sigma'\} \) to obtain the excessive element stresses, \( \{\sigma_{ex}\} \).

8. If a new structural stiffness matrix is to be computed then update the material property matrix, \( [D] \).

9. Convert the excessive stresses, \( \{\sigma_{ex}\} \), into unbalanced nodal forces for the present integration point and add them to the unbalanced nodal forces of the element, \( \{r\} \).

10. If there are unprocessed integration points in this element repeat steps 3 through 10. If not, go to step 11.

11. If no transition criteria was exceeded for this element, go to step 12; otherwise add the unbalanced nodal forces, \( \{r\} \), to the global unbalanced loads, \( \{R\} \), and calculate a new stiffness matrix for the element.

12. If there are still more elements to be checked repeat steps 3 through 12. If not, proceed to step 13.

13. Use the convergence criterion mentioned earlier to determine if convergence is achieved. If it is not achieved, perform a new iteration starting from step 2. If convergence is attained, apply a new load step starting from step 1.
4.4 Evaluation of Excessive Stresses

During the iterative process the structure is analyzed using a stiffness matrix based on the material properties established during the previous iteration. The outcome of the analysis is the strain increment, \( \{\Delta \varepsilon\} \), which together with the previous material properties give the stress increment, \( \{\Delta \sigma_e\} \). This stress increment is added to the previous stresses to obtain \( \{\sigma_e\} \). However, because of the material nonlinearities these stresses are different from the true stresses, \( \{\sigma\} \). The difference between the two is the excessive stresses, \( \{\sigma_{ex}\} \). Thus

\[
\{\sigma_{ex}\} = \{\sigma'\} - \{\sigma\}
\]

As described in Chapter 2, the material properties at the beginning and at the end of each iteration can be calculated exactly from the current and the new values of strains, respectively. This calculation is straightforward in the case of cracking and crushing of concrete, but some difficulties are encountered in the case of concrete in the plastic range. One such difficulty is the case when concrete changes from an elastic to a plastic state in one load step. Figure 4.4 shows graphically this condition in a two-dimensional stress space. Based on the material properties available at the beginning of the load increment, the stress increases by \( \{\Delta \sigma_e\} \) and a new stress point, \( \{\sigma'\} \), is reached. The corresponding strain increment \( \{\Delta \varepsilon\} \), may be separated into two parts;
an elastic strain increment, \( r\{\Delta \varepsilon\} \), which corresponds to a stress point on the yield surface, and a plastic strain increment, \((1-r)\{\Delta \varepsilon\}\). The stress increment is separated into two parts in a similar manner. The factor \( r \) can be obtained by a linear interpolation involving the values of the yield function at the two stress points \( \{\sigma_0\} \) and \( \{\sigma'\} \). Linear interpolation is usually accurate if the load step is small. However, for large load steps a better estimate for the factor, given by Nayak and Zienkiewicz, can be used instead. The excessive stresses may be calculated from

\[
\{\sigma_{ex}\} = \int \frac{\{\Delta \varepsilon\}}{r\{\Delta \varepsilon\}} [D_p] d\{\varepsilon\} \quad (4.9)
\]

where \([D_p]\) is the plastic material property matrix which is given in Appendix A. The following relationship may be used as an approximation to Eq. 4.9

\[
\{\sigma_{ex}\} = (1 - r) [D_p] \{\Delta \varepsilon\} \quad (4.10)
\]

Using Eqs. 4.8 and 4.10, the correct incremental stress, \(\{\Delta \sigma\}\), can be calculated and the stress point \(\{\sigma'\}\) is obtained as shown in Fig. 4.4. However, because Eq. 4.10 is an approximation to the matrix differential equation, Eq. 4.9, this stress point may not, in general, lie on the yield surface. Such a departure from the yield surface is cumulative and should be avoided. One method of restoring the yield condition
is to scale down to the yield surface the stresses, \( \{\sigma_i\} \), along the normal to the surface. Another method of avoiding departures from the yield surface is to use a better approximation than that used in Eq. 4.9. Accordingly, a refined version of numerical procedures such as the Runge-Kutta or the predictor-corrector may be used to solve Eq. 4.9. No corrections are necessary in such a case.

Finally, the excessive stresses, \( \{\sigma_{\text{ex}}\} \), in an element are converted to unbalanced nodal forces using the following equation,

\[
\{r\} = \int_\Omega [B]^T \{\sigma_{\text{ex}}\} \, dv. \tag{4.11}
\]

In the case of the linear isoparametric element, this equation becomes

\[
\{r\} = t \int_{-1}^{+1} \int_{-1}^{+1} [B(\xi, \eta)]^T \{\sigma_{\text{ex}}(\xi, \eta)\} \det[J(\xi, \eta)] \, d\xi \, d\eta \tag{4.12}
\]

Equation 4.12 is integrated numerically as follows:

\[
\{r\} = t \sum_{i=1}^{2} \sum_{j=1}^{2} H_i H_j \det[J(\xi_i, \eta_j)][B(\xi_i, \eta_j)]^T \{\sigma_{\text{ex}}(\xi_i, \eta_j)\} \tag{4.13}
\]

For a truss element, Eq. 4.11 becomes

\[
\{r\} = A \cdot [B]^T \sigma_{\text{ex}} \tag{4.14}
\]
4.5 The Computer Program

The proposed method of investigation has been implemented in a computer program that is capable of performing nonlinear analysis of planar reinforced concrete structures subjected to fixed and monotonically increasing loads. The program has a restart capability and can handle problems with large number of degrees of freedom. It has an automatic mesh generator and an alphanumeric mesh plotter. The organization of the program and a brief description of the input and output features are presented in Appendix C.
CHAPTER 5

NUMERICAL SOLUTIONS

5.1 General

In this chapter the applicability of the proposed model to the solution of nonlinear reinforced concrete planar structures is demonstrated. Also, the effect of the parameters involved in the formulation of the material model is investigated.

Five numerical examples are selected. The analytical results are compared with the experimental results whenever possible. The first example is chosen to test the plasticity routine in the program. A thick circular cylindrical tube under internal pressure was solved and the results were compared with those from the finite difference solution obtained by Hodge. The other numerical examples ranged in complexity from a shallow reinforced concrete beam to a thick multiple-opening reinforced concrete conduit. The shallow beam, example 2, was studied to observe how well the model can predict the flexural failure pattern. The deep beam, example 3, served as an example for predicting diagonal tension failure in reinforced concrete members.

Examples 4 and 5 were solved to verify the applicability of the proposed method to more complex structures such as the thick reinforced concrete conduits.

All computations were carried out on the IBM 360/75 computing system operated by the Department of Computer Science of the University of Illinois.
5.2 The Thick Hollow Cylinder

An infinitely long thick circular cylinder was chosen for testing the plasticity routine of the computer program. Hodge has obtained a finite difference solution for this problem for the case of an elastic-perfectly plastic metal using the von Mises yield criterion. The cylinder has an inner radius $a$, an external radius $2a$, and it is subjected to a monotonically increasing internal pressure $p$, Fig. 5.1a. For this loading and since the cylinder is considered infinitely long, the axial strain in the cylinder is zero.

Two different idealizations of the problem were used in this study. In the first idealization, Fig. 5.1 (b), the solution was obtained using 10 axisymmetric quadrilateral elements. In the second idealization, Fig. 5.1 (c), 80 plane strain quadrilateral elements were used in one quarter of a slice of the cylinder. A fine mesh was necessary to approximate closely the circular edges. The mesh was generated using the automatic mesh generator described in Appendix C.

The results of the analysis are presented in non-dimensionalized form in Figs. 5.2 and 5.3 (G and $k$ in the figures are the modulus of shear and the yield stress in pure shear for the material used, respectively). Figure 5.2 shows the distribution of the internal and the external radial displacements for various positions of the elasto-plastic boundary radius, $c$. As can be seen from this figure, the axisymmetric finite element solution compares better with the finite difference solution than the plane strain finite element solution. This may be due to the
better idealization the circular edges of the cylinder by axisymmetric elements. Shown also in Fig. 5.2, is the distribution of the internal pressure.

Figure 5.3 shows the distribution of the radial, hoop, and axial stresses after the elastio-plastic boundary has propagated to 1.4a. For this case, only the axisymmetric finite element solution is compared with the finite difference results. This case corresponds to an internal pressure of 1.58 the pressure at the initiation of yielding.

5.3 Shallow Reinforced Concrete Beam

5.3.1 Experimental Beam Geometry and Behavior

A shallow reinforced concrete beam, tested at the University of Illinois, was analyzed to demonstrate the adequacy of the proposed model for predicting the flexural failure pattern, and to study the effect of the various parameters and assumptions made during the formulation of the model.

The geometry and the cross-sectional properties of the selected specimen, designated L-6 in Ref. 56, are shown in Fig. 5.4 (a). The average compressive strength of the concrete in the beam was 4470 psi. The modulus of rupture of the concrete was estimated to be 550 psi. Beam L-6 had no web reinforcement but contained 2 in$^2$ of tensile reinforcement of intermediate grade deformed bars (yield stress = 46 ksi, yield strain = 0.16%).
The beam was loaded to failure in 10 load increments. It failed in flexure at an ultimate load of 21.1 kips. Failure started when the reinforcement had yielded and was immediately followed by general crushing of concrete in compression in the region of pure flexure. Deep flexural cracks (approximately 8 in. long) were observed in the pure flexure region prior to failure. However, no diagonal tension cracks were seen in the shear span regions. The experimental load-deflection curve for this beam is shown as a solid continuous line in Fig. 5.8.

5.3.2 Behavior Predicted From the Analysis

Because of symmetry only one half of the beam was considered in the analysis, Fig. 5.5. The finite element mesh (mesh I in the figure) used for this preliminary investigation consisted of 110 quadrilateral concrete elements and 22 steel bar elements. The load was distributed over two elements since it was applied through a steel plate. Full bond was assumed to exist between steel and concrete. The material idealization is shown in Figs. 5.4 (b) and 5.4 (c).

Using the mesh in Fig. 5.5, three solutions were obtained to study the effect of the length of the tension unloading tail on the behavior of the beam. The resulting load-deflection curves are presented and compared with the experimental curve in Fig. 5.8. The solution with no tension unloading portion comes closest to the experimental load-deflection curve. The other two solutions (obtained using a
tension unloading tail of $5.0\varepsilon_{cr}$ and $10.0\varepsilon_{cr}$, respectively) showed stiffer behavior and slightly higher ultimate loads (see Fig. 5.8).

The cracking pattern immediately after the yielding of the reinforcement predicted by the first solution is shown in Fig. 5.6, where integration points that have reached yielding, crushing, or cracking and the direction of the crack are shown. The cracks were well distributed over the entire beam length. Almost all cracks penetrated the same length and no one crack was dominating over the rest of the cracks. This excludes any possibility for a shear failure of the beam. Few integration points have yielded in the compression zone of the pure flexure region.

The distribution of stresses in the reinforcement along the length of the beam at various load levels is shown in Fig. 5.7. This figure shows the effect of the redistribution of forces within the cracked beam on the changes in steel stress from one section to another as the load is increased.

The stiff behavior of the two solutions with tension unloading portion (Fig. 5.8) may be attributed to the following factors:

a) Since the analytical beam has exhibited stiffer behavior even in the elastic range, the material model used is probably stiffer than the material itself. No stress-strain curves for concrete were reported in Ref. 56 for comparison purposes. The modulus of elasticity of concrete was approximated by $3.5 \times 10^6$ psi using Eq. 2.10.

b) The concrete strength in uniaxial tension is lower than its modulus of rupture value which was used in the solution.
c) The use of a fairly fine mesh (110 elements and 138 nodes) may have alleviated the need to use a tension unloading tail for cracked concrete. It is worth mentioning here that the presence or absence of incompatible modes did not have any effect on the solution due to the fine mesh used. To study the effect of the fineness of the finite element mesh on the cracking solution, a relatively coarse mesh (mesh 2) consisting of 30 concrete elements and 10 steel bar elements, Fig. 5.9, was used to obtain an additional series of solutions. A tension-unloading tail of $1.0\varepsilon_{cr}$ long was used throughout.

The advantages accrued by including the incompatible modes in the finite element formulation are clearly shown in Fig. 5.11. The absence of incompatible modes caused the analytical beam model to be both stiffer and stronger than the experimental beam. On the other hand, using fewer elements has resulted in slightly higher load level at yielding of steel. The crack distribution for the last solution at the yield level is shown in Fig. 5.10.

All analytical results presented so far have been obtained assuming full shear carrying capacity of cracked concrete. To demonstrate the validity of this assumption, two additional solutions were obtained, Fig. 5.12. In the first solution it was assumed that concrete loses its shear carrying capacity upon cracking ($\mu = 0$ in Eq. 2.12). The results show that cracks have propagated along the reinforcement until the steel bars came loose from the surrounding concrete and caused the premature failure indicated in Fig. 5.12. Another
solution was obtained assuming that concrete retains 50 per cent of its shear carrying capacity once it cracks ($\mu = 0.5$ in Eq. 2.12). The load-deflection curve for this case, Fig. 5.12, was almost identical with that obtained assuming full shear capacity (Fig. 5.11).

The complete solution up to the yield loading took approximately 240 seconds using mesh 1 and 74 seconds using mesh 2 on the IBM 360/75 computer.

5.4 Deep Reinforced Concrete Beam

5.4.1 Experimental Beam Geometry and Behavior

A deep reinforced concrete beam was selected from a series of tests performed by Crist. The beam (designated as 2S3.6-1 in Ref. 57 and hereby referred to as deep beam) was chosen to study the applicability of the proposed model for the prediction of shear failures in reinforced concrete deep members. The beam had a span-to-depth ratio of 3.8. Its geometry is shown in Fig. 5.13(a).

The deep beam was simply supported and uniformly loaded. The longitudinal reinforcement consisted of two layers of 2 no. 11 bars each. The reinforcing bars were anchored at each end by welding them to cross pieces of reinforcing steel. However, the beam had no web reinforcement.

The average concrete compressive strength was 3698 psi and the split cylinder strength was 325 psi. The material properties used in the analytical investigation are presented in Fig. 5.13(b).
The beam was loaded to failure in five load increments. Inclined cracks were dominant throughout the loading sequence. Two symmetrical diagonal tension cracks had propagated all the way through the depth of the beam (Fig. 5.18) and had caused the beam to fail. Yielding of the longitudinal reinforcement occurred at a total load of 420 kips and the beam failed at a load of 459 kips. The failure mode was classified as diagonal tension bordering on a shear-compression failure. The load-deflection curve is presented in Fig. 5.15.

5.4.2 Finite Element Solutions

Several finite element solutions were carried out. In each solution, the concrete was idealized by 54 quadrilateral elements. The thickness of concrete elements alongside the tensile reinforcement was reduced to account for the presence of the steel bars. Two different idealizations were used for simulating the tensile reinforcement. In the first idealization the reinforcement was represented by two layers of steel bar elements whereas in the second idealization two-dimensional steel elements were used. It was thought that the latter representation would provide a better simulation to the dowel action of the longitudinal reinforcement in the presence of inclined cracks. The mesh geometry and the various element layouts are shown in Fig. 5.14. Dual nodes were used in the vicinity of the longitudinal reinforcement in anticipation of possible later use in connection with the tie-link elements.
To alleviate the problem of stress concentration that arises when point supports are used, the analytical model, like its experimental counterpart, was supported by 12x12 in. steel bearing plates which in turn rested on two roller supports. The reinforcement was anchored to the sides of the beam by a 3 in. thick plate, Fig. 5.14. The bearing and the anchor plates ran through the full thickness of the beam.

The first solution was obtained using concrete and steel bar elements with the assumption that the two types of elements are rigidly connected. The load-deflection curve obtained from this solution is presented in Fig. 5.15(a). It is noticed in this figure that the analytical model exhibits a stiffer behavior than the actual beam even in the elastic range. For this reason the modulus of elasticity of concrete was reduced in the next solution by about 15 per cent. Since the cracking and yielding criteria for concrete are based on stress rather than strain in this study, the reduction of the modulus of elasticity does not affect the stress distribution in the model appreciably. The two analytical load-deflection curves, Fig. 5.15(a), are not significantly different from each other. In both cases, the model predicted a failure load of 441 kips which is within 4 per cent of the actual failure load of the test beam.

In the third solution, quadrilateral steel elements instead of bar elements were used. Although a reduced modulus of elasticity for concrete was also used, the analytical load-deflection curve is still stiffer than the experimental curve as shown in Fig. 5.15(b). However,
both methods for idealization of steel predict similar failure loads. The stress distribution in the lower reinforcement layer for the last model at different load levels is shown in Fig. 5.16(a). The stress in the steel remained constant over approximately half the length of the beam indicating the presence of arch action at higher load levels.

In an attempt to further soften the analytical model, tie-link elements were used to connect the reinforcement to the concrete. The inclusion of such elements in the finite element mesh was easily achieved by dual nodes which are required for the tie-link element. The idealized bond-slip relationship chosen for this study is shown in Fig. 5.19. It is based on the results of the pulling tests performed by Nilson. The unloading portion of the curve was treated in a manner similar to the tension-unloading tail for cracked concrete. The bond stress was released gradually by iteration instead of using a negative spring stiffness for the tie-link element which has reached the unloading stage.

The resulting stress distribution in the lower layer of the reinforcing bars is shown in Fig. 5.16(b). The stress distribution in the steel obtained from this solution is smoother than that where no tie-link elements were used. Figure 5.17 shows the analytical bond stress distribution in the lower layer of the tie-link elements at different load levels. As expected, the bond stress in these elements seems to peak in the region of high shear stress (near the support). The presence of the tie-link elements caused a more flexible behavior of the model only after one of the tie-links, element No. 16 in Fig. 5.14(c), had yielded at a load level of 392 kips. This was followed by yielding a
few steel bar elements of the lower layer at a load level of 416 kips and also by yielding in link no. 8. From this load level on the present analytical model was considerably softer than its two predecessors. Failure occurred at a load level of 452 kips.

In another solution, tie-link elements were added to the quadrilateral steel element model. Unfortunately, the resulting equilibrium equations were ill-conditioned and the solution had to be stopped after applying the second load increment. This behavior could be due to using a very stiff spring to tie the dual nodes together in the vertical direction.

All the foregoing solutions were obtained assuming a tension-unloading curve for concrete of $1.0\varepsilon_{cr}$ long and a full shear carrying capacity for cracked concrete. Ironically, the use of a tension-unloading tail length of $5.0\varepsilon_{cr}$ predicted the failure of the beam after the applied load had reached approximately half of the ultimate load. This, as well as the influence of the shear capacity parameter of cracked concrete will be discussed in the next section.

5.4.3 Internal Stress Distribution in the Model With Tie-Link Elements

Unlike reinforced concrete beams failing in flexure, the cracking pattern for beams failing in diagonal tension is dominated by one or two distinct inclined cracks. These cracks are usually wide, hence, the cracking pattern obtained from an analytical model based on the
assumptions of continuous media is only approximate. Moreover, the inclined crack will be spread over several elements. Figure 5.20 shows the analytical cracking pattern at four different load levels. At a load level of 147 kips (low load level), Fig. 5.20(a), almost all cracks propagated to the same depth. At a load level of 294 kips some cracks propagated deeper than others indicating the presence of diagonal tension field in that region, Fig. 5.20(b). Similar cracking pattern existed at a load level of 343 kips, Fig. 5.20(c). Prior to failure some of the integration points in the upper central region of the beam experienced yielding in compression (Fig. 5.20(d)).

Figure 5.21(a) shows the direction and relative magnitudes of the principal stresses in the elastic range. Beam action is clearly dominating in the model at this load level. However, as the load is increased, the cracks carve out a well-defined arch as shown in Fig. 5.21(b). The deformed element plots corresponding to two load levels are shown in Fig. 5.22. Some of the vertical straight lines which define the finite element boundaries in Fig. 5.22 are no longer straight because of pronounced shear deformations which exist in the region of high shear stresses.

The distribution of the longitudinal strains and stresses across the depth of the beam at different load levels and at various sections is shown in Figs. 5.23 and 5.24, respectively. The discrepancy in the sign of the longitudinal strains and the corresponding longitudinal stresses in the vicinity of the support may be attributed to the fact
that the previously cracked concrete in that region constitutes a part of the inclined arch which has a horizontal component in compression. This is caused by the assumption that cracked concrete can carry an unlimited amount of shear stresses. This assumption is not realistic and has apparently caused the arch to be unduly flat and thick. It may also be the cause for the model to be slightly stiffer and more linear than the real cracked beam (see Fig. 5.15).

The distribution of the shear stresses across the depth of the beam at various sections and for different load levels is shown in Fig. 5.25. Shear stresses drop to relatively small values at the level of the longitudinal reinforcement, indicating the transfer of these stresses to the reinforcement in the form of bond stresses. The distribution of the vertical stresses along different horizontal sections of the beam is shown in Fig. 5.26.

From the many solutions carried out for this example, it appears that there is a critical value for the shear carrying capacity parameter of cracked concrete ($\mu$ in Eq. 2.12) for deep beams where failure is more likely to be in shear rather than flexure. Using a value of $\mu = 0.5$ did not improve the behavior of the analytical model and it caused the solution to diverge at a total load of 343 kips. On the other hand a $\mu$ value of 0.75 had no effect on the load-deflection curve.

It is interesting to note that using a tension-unloading tail length of $5.0e_{cr}$ for cracked concrete caused premature failure of the analytical beam. This early failure may be due to the fact that the
model forces tensile stresses to be reduced gradually to zero after cracking, thus preventing cracked concrete from resisting any compressive stresses in the cracked direction until all tensile stresses are dissipated. Therefore, the inclined arch is prevented and consequently early failure ensues.

In summary, the ultimate load of the test beam was predicted within 4 percent using the proposed model. The use of tie-link elements improves the results only slightly. The overall distribution of the internal strains and stresses can be predicted satisfactorily with the analytical model.

On the average, each solution took 350 seconds on the IBM 360/75 computer with five load steps.

5.5 Reinforced Concrete Conduit 3 to 1 Loading

5.5.1 Experimental Model Geometry and Behavior

This numerical example was chosen from a series of 8 tests conducted at the University of Illinois as a part of an investigation of Multiple Opening Reinforced Concrete Conduits. The investigation was initiated for the purpose of providing information needed for the rational design of conduits or 'box culverts' suitable for use under earth dams and other embankments with fill heights ranging up to about 250 ft. Under such high embankments the span-to-depth ratio of the horizontal members of the conduit becomes small and failure is more
likely to be in shear. Moreover, the stress state in the conduit members is complicated because each member is subjected to a combination of bending moment, shear, and high axial force. Solutions were obtained for two conduit models with identical geometry but subjected to different loading conditions. Solution of model R4 (referred to here as R4) is presented in this section, whereas model R5 will be discussed in the next section.

The dimensions of R4* are shown in Fig. 5.27. The nominal load ratio (vertical to horizontal) for this model was 3 to 1. The load was applied using closely spaced hydraulic jacks as shown in Fig. 5.27. The average concrete compressive strength was 5800 psi and the split cylinder strength was 470 psi. Model R4 had equal tensile and compressive reinforcement of 2 No. 4 deformed bars each (yield stress = 71 ksi, yield strain = 0.25%). No shear reinforcement was provided for any of the conduit models tested. A typical cross-section for the members of this conduit is shown in Fig. 5.28(b).

Model R4 was loaded to failure in 14 load steps and the strains and deflections were monitored throughout the loading sequence. Flexural cracks appeared at a vertical load level of 30 ksf (all load values refer to vertical loads on the horizontal members). No inclined cracks appeared anywhere in the model until a load level of 82.5 ksf was reached. The inclined cracks extended at 45 degrees from the inside

* Test results of models R4 and R5 are to be published in the near future.
face of the three horizontal end members towards their mid-depths. An explosive shear failure in one of the horizontal end spans occurred while the load was being increased beyond 112.5 ksf as shown in Fig. 5.29. The nominal span-to-depth ratio for the member in which failure occurred is 3.9 (based on a clear span length of 29.25 in. and an effective depth of 7.5 in.). However, if the distance between the points of zero moment is considered to be the effective span length, then this value becomes significantly lower, indicating an eminent shear type failure.

5.5.2 The Finite Element Solution for Model R4

Due to full symmetry of the structure only one quarter of it was considered in the analysis. The first finite element mesh used for this example contained 116 quadrilateral and 68 bar elements. It was found that the equilibrium of forces in the horizontal direction was off by about 14.0 percent in the elastic range. The number of elements was then increased to 213 quadrilateral and 98 bar elements as shown in Figs. 5.30 and 5.31. Because equilibrium in this mesh checked to within 1.0 percent, it was used in the cracking analysis of R4 as well as R5. No attempt was made to include tie-link elements since their use did not improve the solution of a deep member significantly.

The idealized material properties for R4 are shown in Fig. 5.28(c). A tension-unloading tail length of $1.0\varepsilon_{cr}^q$ was used. Full bond was assumed to exist between steel and concrete. It was assumed that concrete loses 10.0 percent of its shear carrying capacity upon cracking,
The complete analytical solution was achieved in seven load steps. Convergence was in general very slow (an average of 40 iterations for each load step during which the generalized stiffness matrix was updated twice) in spite of the fact that the tolerance for convergence was increased from 1.0 to 5.0 percent. The solution predicted failure after a vertical load of 105 ksf had been reached. The solution was then terminated.

The load-deflection curves at different locations in the conduit are presented and compared with the experimental results in Fig. 5.32. It is noticed that at higher load levels the analytical model is stiffer than the experimental model especially in the horizontal end spans where shear cracking was dominant. This stiff behavior may again be attributed to the assumptions of full bond between steel and concrete and the nearly full shear carrying capacity of cracked concrete.

In general the strains in steel obtained from the analysis correlated very well with the experimental results almost everywhere in the conduit. However, the correlation was better for compressive strains than for tensile strains, particularly in the regions of excessive cracking as shown in Fig. 5.33. This could also be attributed to the reasons mentioned previously (full bond and shear carrying capacity). The analytical solution predicted the yielding of the steel in the horizontal end members at a vertical load of 105 ksf. The yielding apparently caused the solution to diverge. The reinforcement in the experimental model yielded in the same general location but at a slightly higher load.
Positive and negative moment cracks (positive if formed on the interior face of the member) appeared in the horizontal members when the load reached 30 ksf. However, at 90 ksf load level a band of inclined cracks appeared near the interior supports of the horizontal end spans, as well as the horizontal middle span as shown in Fig. 5.34. Also, at this load level a few negative moment cracks appeared in the vertical side members and the concrete in the interior haunches yielded. The presence of the inclined cracks caused a well-defined arch to be formed in the horizontal end spans as shown in Fig. 5.35, where the direction and the relative magnitudes of the principal stress are plotted.

The deformed elements at a load level of 90 ksf are shown in Fig. 5.36. Of interest in this plot is the outward movement of the vertical side members. A similar phenomenon was also observed during the test where the oil volume in the hydraulic jacks on the vertical side members had to be reduced in order to maintain the 3 to 1 loading ratio. Also, no curvature can be seen in the interior vertical members which function primarily as columns supporting the horizontal members.

The distribution of the shear stresses and the horizontal strains and stresses at a vertical load of 90 ksf and at various sections is shown in Figs. 5.37, 5.38, and 5.39. The strain distribution at some locations deviates from a straight line indicating the presence of shear deformations. The shear stress in particular is not zero at few locations on the specimen's surface. This error is mainly due to
the approximate nature of the finite element method and the fact that
the stress at any joint is the average stress value in the surrounding
integration points.

The complete solution (up to a load of 105 ksf) took 17 minutes
on the IBM 360/75 computer.

5.6 Reinforced Concrete Conduit 1 to 1 Loading

5.6.1 Experimental Model Geometry and Behavior

This conduit model is identical in geometry with model R4, Fig.
5.27. However, the loading ratio (vertical to horizontal) for this
model is 1 to 1. The average concrete compressive strength was 5300
psi and the split cylinder strength was 360 psi. Both the positive
and negative reinforcements consisted of 2 No. 4 deformed steel bars.
The material idealization for this numerical example is shown in Fig.
5.40.

Model R5 was loaded to failure in nine load steps. During the
application of load No. 9, which was supposed to reach 75 ksf, there
was a shear failure in the vertical side member as shown in Fig. 5.41.
No inclined cracks were observed at the previous load in this region.
However, the test logs indicated that the concrete in the compression
region had crushed. The failure mode can be classified as diagonal
tension bordering a shear-compression failure.
5.6.2 The Finite Element Solution for Model R5

As mentioned in the previous section, the finite element mesh used for model R4 was also used to analyze this conduit. Full bond between steel and concrete and a full shear carrying capacity for cracked concrete were assumed in the analysis. The analysis was carried out in five load steps. Convergence was achieved much faster than in model R4 (20 iterations on the average for each load step). The solution was stopped at a load of 67.5 ksf when convergence was not achieved in 40 iterations. At this load convergence was very slow indicating that the ultimate load capacity for this model is slightly higher than 67.5 ksf.

The stresses predicted by the analysis indicate that the structure remained elastic up to a load level of approximately 17.0 ksf. The highest stresses were observed in the two vertical side members which indicate that failure is likely to occur in these members. The span-to-depth ratio for the vertical sides is 5.4 (based on a clear span length of 40.5 in. and an effective member depth of 7.5 in.), hence, the members are deep and may fail in shear. Positive and negative moment cracks were observed in the vertical side members as the applied load reached 45 ksf, Fig. 5.42. Negative moment cracks were also noticed in the horizontal end spans at this load level. Unlike the experimental model, inclined cracks appeared during the finite element solution in the vertical side members at 60 ksf load level. At an applied load of
67.5 ksf the inclined cracks were spread over several elements and they almost reached the adjacent concrete haunch. Also, a shallow arch was formed in the vertical sides due to the inclined cracks as shown in Fig. 5.43, where the direction and the magnitude of principal stresses are plotted.

The deformed element plot at 67.5 ksf load level is shown in Fig. 5.44. The deflection predicted by the analysis correlated with the experimental results better in model R5, Fig. 5.45, than in model R4. However, the load-deflection curve predicted at the center of the vertical sides shows softer behavior at low load levels and slightly stiffer behavior at high load levels. The strains in the reinforcing steel at different locations are plotted and compared with the experimental results in Fig. 5.46. As in model R4, the compressive steel strains compared very well with the actual test results but the tensile strains were always lower, particularly near the center of the interior face of the vertical side members. This discrepancy is partly due to the fact that the finite element mesh in that particular location is relatively coarse and the strain in the steel at that location had to be averaged over a length of 4.0 in. The distribution of the vertical strains and stresses and the shear stresses at different cross-sections is shown in Figs. 5.47, 5.48, and 5.49. The distribution of the horizontal stresses at various sections is shown in Fig. 5.50.

In conclusion, the ultimate strength for model R5 was predicted almost exactly using the proposed model. The predicted deflections and
strains correlated fairly well with the test results. The discrepancy between the predicted and the experimental tensile steel strains in the vertical side members is partially due to the coarse finite element mesh used in these members. However, this stiff behavior was also predicted for deep members that were extensively cracked. It was pointed out then that the excessive stiff behavior was caused by the assumptions of full bond between steel and concrete and the nearly full shear carrying capacity of cracked concrete.

The complete solution took approximately 8.5 minutes on the IBM 360/75 computer.
A model for the nonlinear analysis of planar reinforced concrete structures, based on the finite element method, has been presented. An attempt was made to incorporate the main characteristics affecting the behavior of reinforced concrete structures in the material model. Concrete was assumed to be an isotropic homogeneous material in the elastic range. At high stresses concrete may crack or yield according to the octahedral shearing stress failure criterion. This failure criterion seems to predict best the strength of plain concrete under biaxial state of stress.

Experimental results on reinforced concrete members show that cracked concrete retains part of its tensile stress between the cracks due to bond stresses. This phenomenon was incorporated in the material model by introducing to the uniaxial stress-strain curve for concrete in tension an artificial unloading portion. The ability of cracked concrete to transmit shear between its cracked surfaces through different mechanisms was modeled by allowing concrete to retain a portion of its elastic shear modulus after cracking. Concrete was assumed to be an ideally plastic material in compression. Crushing of concrete takes place if the equivalent uniaxial plastic strain exceeds a prescribed value. When concrete cracks the crack direction is fixed at that
location throughout the solution. Steel was treated as an elastic-perfectly plastic material.

Three types of finite elements were used in this study. The linear isoparametric quadrilateral element with incompatible modes was used to idealize concrete. The steel reinforcement was idealized by one-dimensional bar elements or by quadrilateral elements. Tie-link elements were used to connect the steel reinforcement to concrete in such a way that slip between the two materials is permitted according to a nonlinear relationship. The structural stiffness matrix was modified periodically to account for material nonlinearities. The method of solution followed an incremental-iterative scheme in which the structure was loaded with several load increments. In each load increment the unbalanced forces due to material nonlinearities were computed and iterations were performed until equilibrium was attained.

Several numerical examples were solved to demonstrate the validity of the proposed model. The structures analyzed ranged in complexity from a shallow reinforced concrete beam to a multiple-opening reinforced concrete box culvert. In addition, a number of solutions were obtained to study the effects of several parameters on the analytical solutions. The results obtained from the analytical solutions compared fairly well with the experimental findings.

The analytical model gave excellent results for reinforced concrete members failing in flexure. For deep reinforced concrete
members, the analytical model seemed to predict stiffer behavior than that predicted by experiment. The stiff behavior can be attributed to the relatively coarse finite element mesh, and to the crude assumptions for bond-slip relationship and the constant shear carrying capacity of cracked concrete.

The parametric study showed that the size of the load step used affected very little the behavior of the model. The length of the assumed tension-unloading tail for cracked concrete did not affect the ultimate load carrying capacity of the structure. A value of 1.0 to 3.0 \(e_{cr}\) for the tail length can be used. The shear carrying capacity factor was found not to affect the ultimate load of beams failing in flexure as long as it was not zero. However, for deep beams failing in shear there seems to be a critical value for this parameter below which premature failure is predicted. Time and cost limitations did not allow further investigation of this parameter.

In conclusion, reinforced concrete structures are difficult to model analytically mainly because of the meager quantitative information available for bond-slip action and shear transfer in cracked reinforced concrete members. Nevertheless, within the scatter of results that two identical reinforced concrete experiments may exhibit, the proposed model was capable of predicting the ultimate load carrying capacity and the distribution of the internal stresses and strains in steel and concrete for planar reinforced concrete structures.
Finally, it is believed that with our present knowledge of the performance of reinforced concrete structures, using a more sophisticated material model may lead to extra cost without much improvement in results. Therefore, further experiments to determine a quantitative relationship for the mechanisms of shear transfer in cracked reinforced concrete structures is imperative.
LIST OF REFERENCES


Fig. 2.1 Actual and Assumed Uniaxial Stress-Strain Curve for Concrete

Fig. 2.2 Empirical Stress-Strain Curves for Concrete

\[ \sigma = \frac{\frac{E \varepsilon}{\varepsilon_0}}{1 + \left(\frac{\varepsilon}{\varepsilon_0}\right)^2} \]

\[ \sigma = f_c' \frac{\varepsilon}{\varepsilon_0} \left(2 - \frac{\varepsilon}{\varepsilon_0}\right) \]
Fig. 2.3 Stress-Strain Curves for Concrete Under Biaxial Compression (Ref. 18)

Fig. 2.4 Effect of Stress State on the Behavior of Concrete in Tension (Ref. 20)
Fig. 2.5 Tensile Strength as a Function of Compressive Strength (Ref. 20)
(a) Typical Test Beam

(b) Stress Distribution in Cracked Concrete

(c) Shear Transfer in a Beam With Diagonal Tension Crack

Fig. 2.6 Flexural and Diagonal Tension Cracks in Reinforced Concrete Beams
Kupfer, Hilsdorf, Rüschen (18) (experimental)

Fig. 2.7 Biaxial Strength of Concrete
Fig. 2.8 Cracked Concrete Element

Fig. 2.9 Stress-Strain Curve for Steel

Fig. 2.10 von Mises Yield Surface
Fig. 3.1 Linear Isoparametric Quadrilateral Element

(a) Local System  
(b) Cartesian Coordinates

Fig. 3.2 Source of Error in the Linear Isoparametric Element (Ref. 43)

(a) Pure Bending  
(b) Exact Displacements  
(c) Linear Isoparametric Element Displacements
conforming element
8 external DOF

non-conforming element
8 external DOF
4 internal DOF

\[ N_3 = \frac{1}{4} (1+\xi)(1+\eta) \]

Basic shape function

\[ N_5 = (1 - \xi^2) \]

Additional shape function

Fig. 3.3 Shape Functions for the Incompatible Model

Fig. 3.4 Tie-Link Element
Fig. 4.1 Element Side Pressure

Fig. 4.2 Graphic Representation of Nonlinear Problem Solutions (Ref. 50)
Fig. 4.3 Mixed Incremental-iterative Procedure

Fig. 4.4 Graphical Representation of Concrete Becoming Plastic
Fig. 5.1 Finite Element Idealization of the Thick Cylinder
(c) Plane Strain Finite Element Mesh

Fig. 5.1 —— Continued
Fig. 5.2 Internal and External Displacements and Pressures as a Function of the Elastic-Plastic Boundary
Fig. 5.3 The Distribution of Radial, Hoop, and Axial Stresses When the Radius of the Plastic Zone is 1.4a
(a) Beam Geometry and Reinforcement

\[ a(psi) \]

\[ \frac{p}{2} \quad \frac{p}{2} \]

\[ 70'' \quad 36'' \quad 70'' \]

\[ 176'' \]

\[ 12'' \]

\[ 9.84'' \]

\[ 6'' \]

\[ A_s = 2 \]

\[ (b) \] Concrete Idealization

\[ \sigma(\text{psi}) \]

\[ -4400 \]

\[ 0 \quad -0.00126 \quad -0.004 \]

\[ \varepsilon \]

Yielding

Crushing

\[ \sigma(\text{psi}) \]

\[ 550 \]

\[ \varepsilon_{cr} \]

\[ 5\varepsilon_{cr} \]

\[ 10\varepsilon_{cr} \]

\[ \varepsilon \]

\[ E_c = 3.5 \times 10^6 \text{ psi} \]

\[ \varepsilon_{cr} = 0.0016 \]

\[ (c) \] Steel Idealization

\[ \sigma(\text{psi}) \]

\[ 46000 \]

\[ -0.0016 \]

\[ 0.0016 \]

\[ -46000 \]

\[ E_s = 29 \times 10^6 \text{ psi} \]

\[ \varepsilon \]

Fig. 5.4 Numerical Solution 2 -- Shallow Beam (Beam L-6)
Fig. 5.5 Finite Element Idealization of Beam L-6 (Mesh 1)

Legend: \ Location and direction of cracked integration points
+ Location of yielded or crushed integration points

Fig. 5.6 Cracking Pattern at Yield Load, Beam L-6 (Mesh 1)

Fig. 5.7 Stresses in the Steel Reinforcement, Beam L-6 (Mesh 1)
Fig. 5.8 Load-Deflection Curves for Beam L-6 (Mesh 1)
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 | 19 | 20 | 21 | 22 | 23 | 24 | 25 | 26 | 27 | 28 | 29 | 30 |
| P/2 |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |

**Fig. 5.9 Finite Element Idealization of Beam L-6 (Mesh 2)**

Legend:
- \( \backslash \) Location and direction of cracked integration points
- + Location of yielded or crushed integration points

**Fig. 5.10 Cracking Pattern at Yield Load, Beam L-6 (Mesh 2)**
Experimental
Without Incompatible Modes
Using Incompatible Modes
(Tension-Unloading Tail Length of \(1\varepsilon_{cr}\))

Fig. 5.11 Load-Deflection Curves For Beam L-6 (Mesh 2)
Fig. 5.12 Load-Deflection Curves for Beam L-6 (Mesh 2)
(a) Deep Beam Geometry and Reinforcement

Concrete in Tension

Concrete in Compression

Steel in Tension and Compression

(b) Material Idealization

Fig. 5.13 Numerical Solution 3 -- Deep Beam
Fig. 5.14 Finite Element Layout
(a) Mesh Geometry and Concrete Elements
Fig. 5.14 --- Continued
Fig. 5.15 Load-Deflection Curves for the Deep Beam
Fig. 5.16 Steel Stress Distribution

(a) Stress Distribution in the Lower Integration Points of the Quadrilateral Steel Elements

(b) Stress Distribution in the Lower Layer of the Steel Bar Elements
Fig. 5.17 Stress Distribution in the Lower Layer of Tie-Link Elements.

Fig. 5.18 Experimental Cracking Pattern for the Deep Beam.
Fig. 5.19 Bond-Slip Curve After Nilson (Ref. 2)
Fig. 5.20 Cracking Pattern at Different Load Levels for the Deep Beam
Fig. 5.21 Direction and Relative Magnitudes of the Principal Stresses in the Deep Beam
Fig. 5.22 Deformed Element Plots for the Deep Beam

(a) Elastic Deformations

(b) Deformations at Yielding of Steel
Fig. 5.23 Distribution of Longitudinal Strain Across the Depth of the Deep Beam
Fig. 5.24 Distribution of the Longitudinal Stress Across the Depth of the Deep Beam
Fig. 5.25 Distribution of the Shear Stresses Across the Depth of the Deep Beam
Fig. 5.26 Distribution of Vertical Stresses Along the Deep Beam

Load Levels: (1) 49 kips (3) 294 kips
(2) 147 kips (4) 416 kips
Fig. 5.27 Dimensions and Locations of Loading Jacks for Test Specimens R4 and R5
Fig. 5.28 Loading Pattern and Material Idealization for Conduit R4
Fig. 5.29 Photographs of Specimen R4 After Failure
Fig. 5.30 The Finite Element Mesh for Specimens R4 and R5
Fig. 5.31 Node Identification and Locations of Bar Elements for Specimens R4 and R5
Fig. 5.32 Load-Deflection Curves for Specimen R4
Fig. 5.33(a) Load-Steel Strain Curves for Specimen R4 (Compression)
Fig. 5.33(b) Load-Steel Strain Curves for Specimen R4 (Tension)
Fig. 5.34 Predicted Cracking Pattern for Specimen R4
Fig. 5.35 Direction and Relative Magnitudes of the Principal Stresses in Specimen R4 at a Vertical Load of 90 KSF
Fig. 5.36 Deformed Element Plot for Specimen R4 at a Vertical Load of 90 KSF
Fig. 5.37 Distribution of Horizontal Strains in Specimen R4 at a Vertical Load of 90 KSF
Fig. 5.38 Distribution of the Horizontal Stresses in Specimen R4 at a Vertical Load Level of 90 KSF
Fig. 5.39 Distribution of Shear Stresses in Specimen R4 at a Vertical Load of 90 KSF
(a) Loading Pattern for R4

\[ E_c = 3.0 \times 10^6 \text{ psi} \]

Concrete in Tension

\[ \varepsilon_{cr} = 2\varepsilon_{cr} \]

(b) Typical Section

4#4 bars

11.5" clearance

10"

(c) Material Idealization

\[ E_s = 28.4 \times 10^6 \text{ psi} \]

Steel in Tension and Compression

\[ .0025 \]

Fig. 5.40 Loading Pattern and Material Idealization for Conduit R5
Fig. 5.41 Photographs Specimen R5 After Failure
Fig. 5.42 Predicted Cracking Pattern for Specimen R5
Fig. 5.43 Direction and Relative Magnitudes of the Principal Stresses in Specimen R5 at 67.5 KSF Load Level
DEFORMED ELEMENT PLOT  AMPLIFICATION FACTOR= 60

Fig. 5.44 Deformed Element Plot for Specimen R5 at 67.5 KSF Load Level
Fig. 5.45 Load-Deflection Curves For Specimen R5
Fig. 5.46(a) Load-Steel Strain Curves for Specimen R5 (Compression)
Fig. 5.46(b) Load-Steel Strain Curves for Specimen R5 (Tension)
Fig. 5.47 Distribution of Vertical Strains in Specimen R5 at 67.5 KSF Load Level
Fig. 5.48 Distribution of Vertical Stresses in Specimen R5 at 67.5 KSF Load Level
Fig. 5.49 Distribution of Shear Stresses in Specimen R5 at 67.5 KSF Load Level
Fig. 5.50 Distribution of Horizontal Stresses in Specimen R5 at 67.5 KSF Load Level
APPENDIX A

CONSTITUTIVE LAWS OF INCREMENTAL PLASTICITY

In this appendix the incremental elasto-plastic stress-strain relation are described in a general form. The incremental plasticity theory is used to obtain the explicit stress-strain relations.

The basic ingredients necessary for describing the plastic behavior of a material are a yield criterion, a flow rule, and a hardening rule. The yield criterion describes an initial elastic region. Mathematically, it is expressed by a surface in the stress space. The shape of this surface depends on the initial elastic boundary, on the plastic flow history, and on the hardening parameter $K$ which describes the modifications in the yield surface during the plastic flow. Therefore, the general equation of the yield surface is of the form

$$F(\{\sigma\}, \{\varepsilon_p\}, K) = 0$$  \hspace{1em} \text{(A.1)}

where $\{\sigma\}$ and $\{\varepsilon_p\}$ contain the relevant components of stress and total plastic strain, respectively. In this equation $F = 0$ denotes a plastic state; $F < 0$ indicates an elastic state; however, there is no meaning associated with $F > 0$. It is assumed that the yield surface, Eq. A.1, is convex containing the origin and regular.

A basic assumption in the theory of plasticity is that the incremental strains are separable into elastic and plastic portions. Thus,
The incremental elastic strains, \( \text{d} \{ \varepsilon_e \} \), are related to the incremental stresses through the matrix of elastic constants, \([D]\), as

\[
\text{d} \{ \varepsilon_e \} = [D]^{-1} \text{d} \{ \sigma \}
\]

(A.3)

The flow rule relates the plastic strain increments \( \text{d} \{ \varepsilon_p \} \) to stresses and their increments. The normality rule states that the plastic strain increments are derived from a plastic potential. If the plastic potential is identical with the yield surface, then the flow rule is referred to as the associated flow rule. Therefore, the normality rule is expressed as

\[
\text{d} \{ \varepsilon_p \} = d\lambda \left\{ \frac{\partial F}{\partial \{ \sigma \}} \right\}
\]

(A.4)

where \( d\lambda \) is a non-negative constant to be determined. If we denote the normal to the yield surface as \( \{ N \} \), then equation (A.4) becomes

\[
\text{d} \{ \varepsilon_p \} = d\lambda \{ N \}
\]

(A.5)

The total differential of the yield function is

\[
\text{d} F = \left\{ \frac{\partial F}{\partial \{ \sigma \}} \right\} \text{d} \{ \sigma \} + \left\{ \frac{\partial F}{\partial \{ \varepsilon_p \}} \right\} \text{d} \{ \varepsilon_p \} + \frac{\partial F}{\partial \kappa} \kappa = 0
\]

(A.6)

The following remarks could be made about Eq. A.6. Firstly, since the case where, \( \text{d} F < 0 \), can only lead to unloading from plastic to elastic state, then \( \kappa \) must vanish for an elastic state. Therefore, the condition for unloading is

\[
\kappa = 0
\]
Secondly, when a change from one plastic state to another is not accompanied by a change in the plastic strain state, the process is termed neutral loading. This condition is satisfied when

\[ \{ \frac{\partial F}{\partial \{ \sigma \}} \} \ d\{ \sigma \} = 0 \]

For an ideal plastic material this condition should always be satisfied. Otherwise the stress point loses contact with the yield surface. A geometric interpretation to this condition is that the stress increment vector should always be tangent to the loading surface, (Refer to Fig. A.1). Finally, if a change in the plastic state is accompanied by plastic deformations, then loading occurs. The condition for loading is

\[ \{ \frac{\partial F}{\partial \{ \sigma \}} \} \ d\{ \sigma \} > 0 \]

The total differential expressed in Eq. A.6 may be written as

\[ \{ N \}^T \ d\{ \sigma \} - H d\lambda = 0 \]  \hspace{1cm} (A.7)

where

\[ H = - \frac{1}{d\lambda} (\frac{\partial F}{\partial \kappa} \ d\kappa + \{ \frac{\partial F}{\partial \{ \varepsilon_p \}} \}^T \ d\{ \varepsilon_p \}) \]

The hardening rule is associated with the manner of constructing consecutive yield surfaces. The isotropic hardening rule assumes a
uniform expansion of this initial yield surface. The hardening parameter \( \kappa \) describes this evolution. Two definitions have been proposed \(^{26}\) for this parameter. One definition states that \( \kappa \) is a function of the plastic work only, hence, it is independent of the strain path, i.e.

\[
d\kappa = \{\sigma\}^T d\{\varepsilon_p\}
\]

The other definition is based on the assumption that the work hardening parameter, \( \kappa \), is solely a function of the so-called equivalent plastic strain

\[
d\kappa = d\varepsilon_p
\]

where \( \varepsilon_p \) is given by

\[
d\varepsilon_p = \left[ \frac{1}{3} (2d\varepsilon_x^p + 2d\varepsilon_y^p + 2d\varepsilon_z^p + d\gamma_{xy}^p + d\gamma_{yz}^p + d\gamma_{zx}^p) \right]^\frac{1}{2} \quad (A.8)
\]

The two definitions are identical \(^{26}\) for the von Mises yield conditions. However, the first definition is more general and will be used here. Nayak and Zienkiewicz \(^{27}\) have shown that for isotropic hardening, the parameter \( H \) in Eq. A.7 is the slope of the uniaxial stress-equivalent plastic strain curve. Substituting Eqs. A.3 and A.5 into Eq. A.2 one obtains

\[
d\{\varepsilon\} = [D]^{-1} d\{\sigma\} + d\lambda \{N\} \quad (A.9)
\]

which after some numerical manipulation may be written as \(^{27}\)

\[
d\lambda = \frac{\{N\}^T [D] d\{\varepsilon\}}{H + \{N\}^T [D] \{N\}} \quad (A.10)
\]

Substituting Eq. A.10 into Eq. A.9 and rearranging the terms one obtains

\[
d\{\sigma\} = [D]_{ep} d\{\varepsilon\} \quad (A.11)
\]
where \([D]_{ep}\) is the required elasto-plastic material property matrix and is given by

\[ [D]_{ep} = [D] - [D_p] \]  

(A.12)

in which

\[ [D_p] = [D] \{N\} \{N\}^T [D] \]

\[ H + \{N\}^T [D] \{N\} \]

Equation A.11 gives the stress increment uniquely with the corresponding strain increment. It is convenient to use Eq. A.11 in connection with the finite element method.

Lastly, with many yield surfaces proposed for different kinds of materials, it is advantageous to make the plasticity routine in the finite element computer program independent of the failure criterion used. To this end Nayak and Zienkiewicz\(^{27}\) have suggested that failure criteria be expressed in terms of three invariant quantities \((\sigma_m, \bar{\sigma}, \phi)\) given as

\[ \sigma_m = \frac{J_1}{3} = \frac{1}{3} (\sigma_x + \sigma_y + \sigma_z) \]

\[ \bar{\sigma} = J_2^{\frac{1}{2}} = \left[ \frac{1}{2} (S_x^2 + S_y^2 + S_z^2) + \tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2 \right]^{\frac{1}{2}} \]

\[ \phi = \frac{1}{3} \sin^{-1} \left[ -\frac{3}{2} \frac{J_3}{\sigma^3} \right] \quad -\frac{\pi}{6} < \phi < \frac{\pi}{6} \]

where
$J_1$, $J_2$, $J_3$ are the stress invariants, and $s_x$, $s_y$, $s_z$ are the deviatoric stresses obtained from

$$s_x = \sigma_x - \sigma_m; \quad s_y = \sigma_y - \sigma_m; \quad s_z = \sigma_z - \sigma_m$$

The stress invariant, $J_3$, is given as

$$J_3 = s_x s_y s_z + 2\tau_{xy} \tau_{yz} \tau_{zx} - s_x \tau_{yz}^2 - s_y \tau_{zx}^2 - s_z \tau_{xy}^2$$

To obtain the normal to the yield surface, $\{N\}$, the chain rule of differentiation is applied to Eq. A.1 in the following form

$$\{N\} = \frac{\partial F}{\partial \{\sigma\}} = \frac{\partial F}{\partial \sigma_m} \frac{\partial \sigma_m}{\partial \{\sigma\}} + \frac{\partial F}{\partial \sigma} \frac{\partial \sigma}{\partial \{\sigma\}} + \frac{\partial F}{\partial \phi} \frac{\partial \phi}{\partial \{\sigma\}}$$

or

$$N = C_1\{n_1\} + C_2\{n_2\} + C_3\{n_3\}$$

where

$$\{n_1\} = \frac{\partial \sigma_m}{\partial \{\sigma\}}, \quad \{n_2\} = \frac{\partial \sigma}{\partial \{\sigma\}}, \quad \{n_3\} = \frac{\partial \phi}{\partial \{\sigma\}}$$

For the case where the relevant stresses are $\sigma_x$, $\sigma_y$, $\sigma_z$, and $\tau_{xy}$ the derivatives of the stress invariants are

$$\{n_1\}^T = \left\{\frac{\partial \sigma_m}{\partial \{\sigma\}}\right\}^T = \frac{1}{3} \{1, 1, 1, 0\}$$

$$\{n_2\}^T = \left\{\frac{\partial \sigma}{\partial \{\sigma\}}\right\}^T = \frac{1}{2\sigma} \{s_x, s_y, s_z, 2\tau_{xy}\}$$

$$\{n_3\} = \frac{\partial \sigma}{\partial \{\sigma\}} = -\frac{\sqrt{3}}{2\cos 3\phi} \left[ \frac{1}{\sigma} \frac{\partial J_3}{\partial \{\sigma\}} - \frac{3J_3}{4} \frac{\partial \sigma}{\partial \{\sigma\}} \right]$$

(A.13)
The values of the constants $C_1$, $C_2$, and $C_3$ depend on the failure criteria used. For von Mises yield criterion, these constants are

$$C_1 = 0, \quad C_2 = \sqrt{3}, \quad C_3 = 0$$  \hspace{1cm} (A.14)

For the octahedral shear stress yield criterion, as given in Eq. 2.6 of the text, the constants are

$$C_1 = \frac{1 - \alpha}{1 + \alpha}, \quad C_2 = \frac{1}{3}, \quad C_3 = 0$$  \hspace{1cm} (A.15)

With regards to Eq. 2.7, these constants become

$$C_1 = \frac{\beta - 1}{2\beta - 1}, \quad C_2 = \frac{1}{3}, \quad C_3 = 0$$  \hspace{1cm} (A.16)

As can be seen from Eqs. A.14, A.15, and A.16, adding a new failure criterion to the computer program requires only changing the values of the constants, $C_1$, $C_2$, and $C_3$. This formulation is adopted for this study.
Fig. A.1 Geometric Interpretation of the Normality Rule

(a) Elastic Unloading

(b) Neutral Loading

(c) Loading
APPENDIX B

ELEMENT STIFFNESS MATRICES

B.1 General

The stiffness matrices for the three elements (the bar element, the linear isoparametric quadrilateral element, and the tie-link element) used in this study are presented herein. The stiffness matrices are formulated in a form suitable for implementation in the computer program. Because of the disadvantage of using multi-subscripted arrays from the computational efficiency standpoint, tensor notation is avoided and replaced by column vectors whenever possible.

In the following sections, a general stiffness formulation based on the use of interpolation functions is followed in obtaining the stiffness matrices for the first two elements. The third element is treated as a special case of the truss element. The formulation of the stiffness matrix for the 12 degrees of freedom (DOF) isoparametric quadrilateral element presented in Section B.3 is based on the work of Wilson.

B.2 Stiffness Matrix for the Two-Dimensional Truss Element

Figure B.1(a) shows representation of a bar element in a simple natural coordinate system, whereas, Fig. B.1(b) shows the same element in the global coordinate system, x, y. Since the bar element is
connected to two joints i and j the shape functions, Fig. B.1(c) are linear and may be expressed as

\[ N_1 = \frac{1}{2} (1 - \xi) \]

\[ N_2 = \frac{1}{2} (1 + \xi) \]  \hspace{1cm} (B.1)

For one-dimensional representation, the spatial coordinate, s, is related to the natural coordinate as follows:

\[ s = \frac{1}{2} (1 - \xi) s_1 + \frac{1}{2} (1 + \xi) s_2 \]  \hspace{1cm} (B.2)

The inversion of the above equation gives

\[ \xi = \frac{s - s_3}{s/2} \]  \hspace{1cm} (B.3)

where

\[ s_3 = \frac{1}{2} (s_1 + s_2) \]

and \( s \) is the length of the bar element. In the global coordinate system, x, y the displacement field can be expressed in terms of the same shape functions as

\[
\begin{pmatrix}
-u_1 \\
u_2
\end{pmatrix} =
\begin{bmatrix}
N_1 & N_2 & 0 & 0 \\
0 & 0 & N_1 & N_2
\end{bmatrix}
\begin{pmatrix}
\bar{u}_1 \\
\bar{u}_2 \\
\bar{u}_3 \\
\bar{u}_4
\end{pmatrix} \hspace{1cm} (B.4)
\]
or in compact form

\[ \{ u \} = [N] \{ \bar{u} \} \]  

(B.5)

where \( \{ \bar{u} \} \) is a vector of nodal displacements as shown in Fig. B.1(b). The axial deformation, \( u_s \), is related to the two-dimensional displacements through the direction cosines as

\[ u_s = [\alpha_x \alpha_y] \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} \]  

(B.6)

where

\[ \alpha_x = \frac{x_2 - x_1}{s}, \quad \alpha_y = \frac{y_2 - y_1}{s} \]

Substituting Eq. B.5 into Eq. B.6, one obtains

\[ u_s = [\alpha] [N] \{ \bar{u} \} \]  

(B.7)

where \([\alpha]\) is a row vector of direction cosines.

The strain, assumed to be constant throughout the bar, is written as

\[ \varepsilon = \frac{du_s}{ds} = [\alpha] \frac{d[N]}{ds} \{ \bar{u} \} \]  

(B.8)

The derivatives of the shape functions are obtained using the chain rule of differentiation and making use of Eq. B.3. Therefore,
Using Eqs. B.9 and B.10, the uniaxial strain is expressed as

\[
\epsilon = \frac{1}{\ell} \begin{bmatrix}
-\alpha_x & \alpha_x & -\alpha_y & \alpha_y
\end{bmatrix}
\begin{bmatrix}
\bar{u}_1 \\
\bar{u}_2 \\
\bar{u}_3 \\
\bar{u}_4
\end{bmatrix}
\]  

(B.11)

Or in matrix form

\[
\epsilon = [B] \{\bar{u}\}
\]  

(B.12)

The stiffness matrix is given as

\[
[k] = \int_V [B]^T E [B] \, dv
\]

Since [B] is constant, no integration is required; hence

\[
[k] = EA\ell \, [B]^T \, [B]
\]
Upon carrying out the matrix multiplication $[B]^T [B]$, the stiffness matrix may be expressed as

$$[k] = \frac{EA}{\lambda} \begin{bmatrix} \alpha_x^2 & -\alpha_x & \alpha_x \alpha_y & -\alpha_x \alpha_y \\ -\alpha_x & \alpha_x^2 & -\alpha_x \alpha_y & \alpha_x \alpha_y \\ \alpha_x \alpha_y & -\alpha_x \alpha_y & \alpha_y^2 & -\alpha_y^2 \\ -\alpha_x \alpha_y & \alpha_x \alpha_y & -\alpha_y^2 & \alpha_y^2 \end{bmatrix}$$

(B.13)

Finally, the stress in the element is calculated as follows:

$$\sigma = \frac{E}{\lambda} \begin{bmatrix} \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_4 \end{bmatrix} \begin{bmatrix} -\alpha_x & \alpha_x & -\alpha_y & \alpha_y \end{bmatrix}$$

(B.14)

B.3 Stiffness Matrix for the 12 DOF Isoparametric Quadrilateral Element

As discussed in Section 3.3, eight of the twelve degrees of freedom in this element are associated with the nodal points located at the four corners as shown in Fig. 3.1. The additional four degrees of freedom are internal which are eliminated so that the resultant stiffness matrix is the usual 8 x 8 square matrix.
The basic (compatible) shape functions are presented in Eq. 3.10 which can be expanded in the following long form

\[
\begin{align*}
N_1 &= \frac{1}{4} (1 - \xi) (1 - \eta) \\
N_2 &= \frac{1}{4} (1 + \xi) (1 - \eta) \\
N_3 &= \frac{1}{4} (1 + \xi) (1 + \eta) \\
N_4 &= \frac{1}{4} (1 - \xi) (1 + \eta)
\end{align*}
\]

Two parabolic 'incompatible' modes are added to the above linear shape functions to approximate pure bending. As shown in Fig. 3.3 the shape functions take a value of zero at the element corners. The two incompatible shape functions (see also Eq. 3.14) are

\[
\begin{align*}
N_5 &= (1 - \xi^2) \\
N_6 &= (1 - \eta^2)
\end{align*}
\]

Any point within the element may be described in a set of local \(\xi, \eta\) coordinates which are related to the global \(x, y\) coordinates through the basic shape functions as

\[
\begin{align*}
x(\xi, \eta) &= \sum_{i=1}^{4} N_i x_i \\
y(\xi, \eta) &= \sum_{i=1}^{4} N_i y_i
\end{align*}
\]

(B.17)
where $\bar{x}_i$, $\bar{y}_i$ are the global coordinates of the nodal points. The displacement field within the element is approximated by both the basic and the additional shape functions (Eqs. B.15 and B.16)

$$u(\xi, \eta) = [N] \{\bar{u}\} \quad (B.18)$$

$$v(\xi, \eta) = [N] \{\bar{v}\}$$

where $[N]$ is a row vector of interpolation functions given as

$$[N] = [N_1, N_2, N_3, N_4, N_5, N_6]$$

and $\{\bar{u}\}$ and $\{\bar{v}\}$ are column vectors of the external and internal nodal displacements. They are given as

$$\{\bar{u}\}^T = \{\bar{u}_1, \bar{u}_2, \bar{u}_3, \bar{u}_4, \alpha_1, \alpha_2\}$$

$$\{\bar{v}\}^T = \{\bar{v}_1, \bar{v}_2, \bar{v}_3, \bar{v}_4, \beta_1, \beta_2\}$$

in which, $\bar{u}_i$ and $\bar{v}_i$ are the nodal displacements in the $x$ and $y$ directions, respectively. The remaining terms $\alpha_1$, $\alpha_2$, $\beta_1$, and $\beta_2$ are amplitudes of the incompatible modes.

For plane stress problems, the strain-displacements relations are

$$\varepsilon_x = \frac{\partial u}{\partial x} = [N_x] \{\bar{u}\}$$

$$\varepsilon_y = \frac{\partial v}{\partial y} = [N_y] \{\bar{v}\} \quad (B.19)$$

$$\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = [N_y] \{\bar{u}\} + [N_x] \{\bar{v}\}$$
where \([N_x]\), \([N_y]\) are matrices of shape function derivatives with respect to \(x\) and \(y\), respectively. In matrix form, Eq. B.19 is

\[
\begin{pmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{pmatrix}
=\begin{pmatrix}
[N_x] & [0] \\
[0] & [N_y]
\end{pmatrix}
\begin{pmatrix}
[u] \\
[v]
\end{pmatrix}
\]

in which \([0]\) is a row of six zeros. In compact form Eq. B.20 may be written as

\[
\{\varepsilon\} = [B] \{q\}
\]

(B.21)

where \(\{\varepsilon\}\) is a column vector of element strains, \([B]\) is a 3x12 matrix of shape function derivatives and zeros, and \(\{q\}\) is a column vector of the twelve generalized displacements. The evaluation of the elements of matrix \([B]\) requires the transformation of the derivatives to \(\xi, \eta\) variables, which can be accomplished as follows

\[
\begin{pmatrix}
\frac{\partial N_i}{\partial \xi} \\
\frac{\partial N_i}{\partial \eta}
\end{pmatrix}
= \begin{pmatrix}
\frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\
\frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta}
\end{pmatrix}
\begin{pmatrix}
\frac{\partial N_i}{\partial x} \\
\frac{\partial N_i}{\partial y}
\end{pmatrix}
= [J] \begin{pmatrix}
\frac{\partial N_i}{\partial x} \\
\frac{\partial N_i}{\partial y}
\end{pmatrix}
\]

(B.22)

where \([J]\) is the well-known Jacobian matrix. The surface element \(dxdy\) is replaced in all calculations by

\[
dxdy = \text{det} [J] \, d\xi d\eta
\]

(B.23)
where

\[ \det[J] = x, \xi \ y, \eta - x, \eta \ y, \xi \]  
(B.24)

Upon inverting the Jacobian matrix, the required inverse relations are obtained in the following form

\[
\begin{pmatrix}
\frac{\partial N_i}{\partial x} \\
\frac{\partial N_i}{\partial y}
\end{pmatrix}
= \frac{1}{\det[J]}
\begin{pmatrix}
\frac{\partial y}{\partial \eta} & -\frac{\partial y}{\partial \xi} \\
-\frac{\partial x}{\partial \eta} & \frac{\partial x}{\partial \xi}
\end{pmatrix}
\begin{pmatrix}
\frac{\partial N_i}{\partial \xi} \\
\frac{\partial N_i}{\partial \eta}
\end{pmatrix}
\]  
(B.25)

The derivatives in Eq. B.25 may be readily calculated from Eq. B.17. Thus,

\[ x, \xi = \sum_{i=1}^{4} N_i, \xi \bar{x}_i \]
\[ x, \eta = \sum_{i=1}^{4} N_i, \eta \bar{x}_i \]  
(B.26)
\[ y, \xi = \sum_{i=1}^{4} N_i, \xi \bar{y}_i \]
\[ y, \eta = \sum_{i=1}^{4} N_i, \eta \bar{y}_i \]

For plane stress problems the stress-strain relations are expressed through the material property matrix, [D], as
\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix} = \frac{E}{1 - \nu^2} \begin{bmatrix}
1 & \nu & 0 \\
\nu & 1 & 0 \\
0 & 0 & \frac{1-\nu}{2}
\end{bmatrix} \begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix}
\] (B.27)

in which \(E, \nu\) are material constants. In compact form Eq. B.27 is

\[
\{\sigma\} = [D] \{\varepsilon\}
\] (B.28)

Using the principle of minimum potential energy, the equilibrium equations are obtained as shown in Section 3.2. Accordingly,

\[
\{f\} = [k] \{q\}
\] (B.29)

where \(\{f\}\) is a column vector of 12 generalized loads and \([k]\) is the required element stiffness matrix which for a unit thickness of the material can be evaluated from the following expression (Eq. 3.6):

\[
[k] = \int_A [B]^T \{D\} [B] \, dA
\]

or in natural coordinates

\[
[k] = \int_{-1}^{+1} \int_{-1}^{+1} [B]^T \{D\} [B] \det [J] \, d\xi d\eta
\] (B.30)

The difficulties with the closed form surface integration of the above equation are avoided by the use of a 2x2 numerical integration based on Gauss quadrature rule. Equation B.30 yields

\[
[k] = \sum_{i=1}^{2} \sum_{j=1}^{2} H_i H_j \det[J(\xi_i, \eta_j)] \{B(\xi_i, \eta_j)\} \{D\} \{B(\xi_i, \eta_j)\}
\] (B.31)
in which $H_i$ and $H_j$ are weight coefficients which take the value of unity for the case of two point integration (Table 3 in Ref. 35). The variables $\xi_i$ and $\eta_j$ are the coordinates of the four integration points, which lie at a distance of $\pm 1/\sqrt{3}$ on either side of the local coordinate axes. Within the computer program the elements of the $[B]$ matrix and the product $\det[J] [B]^T [D] [B]$ are computed for each integration point and then they are combined.

The outcome of the numerical integration is a 12 x 12 matrix, which if used in its present form, will result in extra computational effort that can be averted by condensing out the non-nodal degrees of freedom. This process, known as static condensation, is achieved by rearranging the rows and columns of the matrix equation, Eq. B.29, and by partitioning the resultant matrices in the following form:

$$
\begin{align*}
\{f\}_{1-8} & = 
\begin{bmatrix}
[k_{aa}] & [k_{ab}] \\
[k_{ba}] & [k_{bb}] 
\end{bmatrix}
\begin{bmatrix}
\{\bar{q}_{1-8}\} \\
\{\bar{q}\}
\end{bmatrix}
\end{align*}
$$

where $\{\bar{q}^T\}$ is a vector of corner nodal displacements which is given as

$$
\{\bar{q}\}^T = \{\bar{u}_1, \bar{u}_2, \bar{u}_3, \bar{u}_4, \bar{v}_1, \bar{v}_2, \bar{v}_3, \bar{v}_4\}
$$

and $\{\bar{q}\}$ is a vector of non-nodal displacements

$$
\{\bar{q}\}^T = \{\alpha_1, \alpha_2, \beta_1, \beta_2\}$$
Since there are no forces associated with the vector, \( \{ \overline{\alpha} \} \), it may be expressed in terms of the corner displacements as

\[
\{ \overline{\alpha} \} = -[k_{bb}]^{-1}[k_{ba}] \{ q \}
\]  \hspace{1cm} (B.33)

From Eqs. B.32 and B.33 the nodal forces can be expressed in terms of the nodal displacements as

\[
\{ f \}_{8 \times 1} = [[k_{aa}] - [k_{ab}] [k_{bb}]^{-1} [k_{ba}]]_{8 \times 8} \{ q \}_{8 \times 1}
\]  \hspace{1cm} (B.34)

The resultant 8 x 8 stiffness matrix gives excellent results for problems with profound bending modes. However, the incompatible modes have very small effect if the stress gradient is flat.

Finally, having obtained the displacements of the structure, the strains and stresses in the element have to be calculated. The strains may be computed from Eq. B.21, but this requires obtaining the values of the generalized displacements \( \{ \overline{\alpha} \} \). This extra calculation may be avoided by partitioning the matrix \( [B] \) as follows:

\[
\{ \varepsilon \} = \begin{bmatrix} [B_a] \\ [B_b] \end{bmatrix} \begin{bmatrix} \{ q_{1-8} \} \\ \{ \overline{\alpha} \} \end{bmatrix}
\]

Upon substituting Eq. B.33 in the above equation, we obtain

\[
\{ \varepsilon \} = ([B_a] - [B_b] [k_{bb}]^{-1} [k_{ba}]) \{ q \}_{8 \times 1}
\]

or in compact form
The elements of the matrix \( [B^*] \) are computed during the static condensation process for each point for which the stress calculation is required and then stored on a magnetic tape. Later, after evaluating the nodal displacements, the matrix \( [B^*] \) is retrieved from the magnetic tape and the stresses are computed according to the following equation.

\[
\{\sigma\} = [D] [B^*] \{\bar{q}\}
\]

**B.4 Stiffness Matrix for the Tie-Link Element**

Figure B.2 shows a representation of the tie-link element in the x, y plane. For the purpose of computing the stiffness matrix, one may assume this element as being composed of two bar elements which have the following stiffness

\[
\begin{bmatrix}
    k_h & 0 \\
    0 & k_v
\end{bmatrix}
\]

where \( k_h \) and \( k_v \) are the spring stiffnesses. The total deformation of the first spring in terms of the nodal displacements can be obtained from Eq. B.12 as

\[
U_{s1} = \varepsilon \ [B] \ \{\bar{u}\}
\]

or
\[ U_{s1} = \begin{bmatrix} -c & c & -s & s \\ \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} \]

where \( c, s \) are cosine and sine \( \theta \), respectively; \( \theta \) being the inclination of the element to the x-axis. Similarly, the total deformation of the second spring, which is perpendicular to the first one, is

\[ U_{s2} = \begin{bmatrix} s & -s & -c & c \\ \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} \]

Combining the two deformations in one equation we obtain

\[
\begin{bmatrix} U_{s1} \\ U_{s2} \end{bmatrix} = \begin{bmatrix} -c & c & -s & s \\ s & -s & -c & c \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix}
\]

or simply

\[
\{U_s\} = [B] \{u\}
\]

The stiffness matrix can be evaluated from the following equation
\[ [k] = [B]^T [D] [B] \]

upon carrying out the matrix multiplication in the foregoing equation, one obtains

\[
[k] = \begin{bmatrix}
A & -A & B & -B \\
-A & A & -B & B \\
B & -B & G & -G \\
-B & B & -G & G \\
\end{bmatrix}
\]

where

\[ A = k_h c^2 + k_y s^2 \]
\[ B = (k_h - k_y) c s \]
\[ G = k_h s^2 + k_y c^2 \]

The forces in the springs may be evaluated from the product of the spring stiffnesses and deformations, Eqs. B.37 and B.38,

\[
\begin{bmatrix}
\vec{f}_1 \\
\vec{f}_2 \\
\end{bmatrix} = \begin{bmatrix}
k_h & 0 \\
0 & k_y \\
\end{bmatrix} \begin{bmatrix}
-c & c \\
-s & s \\
\end{bmatrix} \begin{bmatrix}
\vec{u}_1 \\
\vec{u}_2 \\
\vec{u}_3 \\
\vec{u}_4 \\
\end{bmatrix}
\]

\[(B.41)\]
where $c$, $s$ are cosine and sine $\theta$, respectively; $\theta$ being the inclination of the element to the $x$-axis. Similarly, the total deformation of the second spring, which is perpendicular to the first one, is

$$U_{s2} = \begin{bmatrix} s & -s & -c & c \end{bmatrix} \begin{bmatrix} \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_4 \end{bmatrix}$$

Combining the two deformations in one equation we obtain

$$\begin{bmatrix} U_{s1} \\ U_{s2} \end{bmatrix} = \begin{bmatrix} -c & c & -s & s \\ s & -s & -c & c \end{bmatrix} \begin{bmatrix} \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_4 \end{bmatrix} \quad (B.38)$$

or simply

$$\{U_s\} = [B] \{\bar{u}\} \quad (B.39)$$

The stiffness matrix can be evaluated from the following equation
\[ [k] = [B]^T [D] [B] \]

upon carrying out the matrix multiplication in the foregoing equation, one obtains

\[
[k] = \begin{bmatrix}
A & -A & B & -B \\
-A & A & -B & B \\
B & -B & G & -G \\
-B & B & -G & G \\
\end{bmatrix} \tag{B.40}
\]

where

\[
A = k_h c^2 + k_v s^2 \\
B = (k_h - k_v) cs \\
G = k_h s^2 + k_v c^2
\]

The forces in the springs may be evaluated from the product of the spring stiffnesses and deformations, Eqs. B.37 and B.38,

\[
\begin{align*}
\{f_1\} &= \begin{bmatrix} k_h & 0 \\ 0 & k_v \end{bmatrix} \begin{bmatrix} -c & c & -s & s \\ s & -s & -c & c \end{bmatrix} \begin{bmatrix} \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_4 \end{bmatrix} \\
\{f_2\} &= \begin{bmatrix} 0 & k_v \\ 0 & 0 \end{bmatrix} \begin{bmatrix} -c & c & -s & s \\ s & -s & -c & c \end{bmatrix} \begin{bmatrix} \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_4 \end{bmatrix}
\end{align*} \tag{B.41}
where $c$, $s$ are cosine and sine $\theta$, respectively; $\theta$ being the inclination of the element to the $x$-axis. Similarly, the total deformation of the second spring, which is perpendicular to the first one, is

\[
U_{s2} = \begin{bmatrix} s & -s & -c & c \end{bmatrix}
\begin{bmatrix}
\bar{u}_1 \\
\bar{u}_2 \\
\bar{u}_3 \\
\bar{u}_4 
\end{bmatrix}
\]

Combining the two deformations in one equation we obtain

\[
\begin{bmatrix}
U_{s1} \\
U_{s2}
\end{bmatrix} = \begin{bmatrix} -c & c & -s & s \end{bmatrix}
\begin{bmatrix}
\bar{u}_1 \\
\bar{u}_2 \\
\bar{u}_3 \\
\bar{u}_4 
\end{bmatrix}
\]

or simply

\[
\{U_s\} = [B] \{\bar{u}\}
\]

The stiffness matrix can be evaluated from the following equation
\[ [k] = [B]^T [D] [B] \]

upon carrying out the matrix multiplication in the foregoing equation, one obtains

\[
[k] = \begin{bmatrix}
A & -A & B & -B \\
-A & A & -B & B \\
B & -B & G & -G \\
-B & B & -G & G \\
\end{bmatrix}
\]

(B.40)

where

\[
A = k_h c^2 + k_v s^2
\]

\[
B = (k_h - k_v)c_s
\]

\[
G = k_h s^2 + k_v c^2
\]

The forces in the springs may be evaluated from the product of the spring stiffnesses and deformations, Eqs. B.37 and B.38,

\[
\begin{align*}
\{f_1\} &= \begin{bmatrix} k_h & 0 \\ 0 & k_v \end{bmatrix} \begin{bmatrix} -c & c & -s & s \\ s & -s & -c & c \end{bmatrix} \begin{Bmatrix} \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_4 \end{Bmatrix} \\
\{f_2\} &= \begin{bmatrix} k_h & 0 \\ 0 & k_v \end{bmatrix} \begin{bmatrix} -c & c & -s & s \\ s & -s & -c & c \end{bmatrix} \begin{Bmatrix} \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_4 \end{Bmatrix}
\]

(B.41)
Fig. B.1 Plane Truss Element

Fig. B.2 Tie-Link Element
APPENDIX C

THE COMPUTER PROGRAM

C.1 General

In writing a computer program for nonlinear analysis of structures, one is confronted with the large amount of data that has to be manipulated and stored during the various phases of the incremental analysis. The space required for the data storage becomes even larger if round-off errors in the specific machine used are large so that double precision arithmetic operations are necessary. This will roughly double the amount of core storage. Therefore, it is impossible for all the data to reside in the computer memory during the execution of the program. This difficulty may be bypassed by storing the data on auxiliary storage devices such as disks and magnetic tapes. In this way, relevant data is transferred from the auxiliary storage to the computer core, operated on, and stored back on the auxiliary storage. However, such process becomes costly unless an efficient data retrieval system is employed.

Usually, data is stored on files with headings for identification. Some of these files are temporary (scratch files); others are permanent (global files). Permanent files are especially important for providing the restart capability during the incremental solution. The restart feature enables the analyst, to stop the computer solution, from time to
time, examine the results, or even transfer some of the results to a plotting device and obtain a graphical representation of the displacements, stresses, and strains.

C.2 Program Organization and Flow Chart

The computer program in this study was coded using the FORTRAN IV procedural language. It was code-checked on the Burroughs 6700 computing facilities of the Department of Civil Engineering Systems Laboratory and the numerical examples were obtained on the IBM 360/75 computing facilities of the University of Illinois. The program performs all arithmatic computations in double precision. It has a restart capability and a simple data retrieval system. The structure of the program follows the general outline of the SAP program.

The main feature of the computer program is the MAIN routine which controls the sequence of operations in the problem solution, manages the data retrieval system, and maintains full restart capability. The Program capacity is limited by the total number of nodal points in the problem. However, increasing the program capacity can be achieved easily by changing the size of the COMMON statement in MAIN. The program allocates an area of the computer core as a large working space. In this area, stiffness matrices for different elements are calculated, the structural stiffness matrix is assembled and triangularized, and the stresses are computed. Thus, the main memory of the computer is used in a dynamic manner.
The specification of a small working space will, however, increase the Input-Output time. For the problems solved in this investigation, a working space size of 15000 computer words were found to produce a favorable ratio of Processor's time to Input-Output time. If the length of the working space is exceeded during any phase of the calculation, the program will terminate itself after printing a diagnostic message giving the total length required.

For each element type there are two major subroutines in the program. One subroutine allocates the space, reads element incidences and material properties, and generates the stiffness matrix. The other subroutine computes the strain and stress and performs the computations for nonlinear analysis. New elements can easily be added to the program.

A flow chart for the order of calculations in the program is presented in Fig. C.1.

C.3 Input Data and Presentation of Results

The preparations of the input data for a finite element problem is tedious and time consuming. Input data has to be checked carefully before it is read into the computer. In this study, an attempt was made to minimize the efforts needed for generating and checking the finite element mesh by implementing an automatic mesh generator and a mesh plotter on the line printer.
The finite element automatic mesh generator was adapted from Ref. 53. It is based on mapping a region in the x-y plane into a region in the \( \xi-\eta \) plane (\( \xi \) and \( \eta \) are right-handed coordinates) by satisfying Laplace's equations over the mapped grid, Fig. C.2. In generating the grid, the perimeter of the region is given in the two coordinate systems. Intermediate points on the perimeter are generated by a linear interpolation. The interior points of the finite element grid are then found by satisfying Laplace's equation in the finite difference form for each of the coordinates, x and y over the transformed grid, Fig. C.2(b). This procedure results in finite elements that are similar in size and shape to adjacent elements, Fig. C.2(a).

When a problem is being solved for the first time, the computer program will stop short of generating the element stiffness matrices. It allows a final check on the input data. In addition to printing out the joint coordinates, incidences and joint loads, the program gives a graphical representation of the finite element mesh. This is achieved by implementing in the computer program an alphanumeric mesh plotting routine. The routine generates an alphanumeric image of an arbitrary finite element mesh on the line printer. The available printer characters are used to draw the element sides and node positions together with their identification numbers. It should be pointed out that the plot is approximate because the printer's resolution is only about one percent. However, the printed picture is generally sufficient for visual
inspection. In addition to printing the entire mesh, the analyst may select a certain sequence of elements to be plotted. This option allows the enlargement of portions of the mesh (zooming). A plot for one of the problems considered in this investigation is shown in Fig. C.3.

Complete or partial printout of displacements, strains, and stresses can be obtained at each load increment or at any specified number of load increments. The analyst can monitor the displacements at certain nodes during the iterative procedure. The stresses and strains are given at integration points for the linear quadrilateral element. However, average stresses at the joints can also be obtained by averaging the stresses in four neighboring integration points.

Finally, the results can also be obtained in graphic form on the CALCOMP plotter. Such off-line plots include the finite element mesh for the undeformed and deformed structure, stresses and strains at various sections, and stress contour plots. Typical plots are presented in Chapter 5 in presentation and discussion of the results.
Flow Chart for the Computer Program
(a) Finite Element Grid Generated Using Laplace's Equation

(b) The Right-handed Coordinates Grid

Fig. C.2 Automatic Mesh Generation (Ref. 53)
Fig. C.3 Alphanumeric Finite Element Mesh Plot