Three-Dimensional Nonlinear Analysis of Components of Reinforced Concrete Framed Structures.

Srinivas Maddipudi
Louisiana State University and Agricultural & Mechanical College

Follow this and additional works at: https://repository.lsu.edu/gradschool_disstheses

Recommended Citation
https://repository.lsu.edu/gradschool_disstheses/5453

This Dissertation is brought to you for free and open access by the Graduate School at LSU Scholarly Repository. It has been accepted for inclusion in LSU Historical Dissertations and Theses by an authorized administrator of LSU Scholarly Repository. For more information, please contact gradetd@lsu.edu.
INFORMATION TO USERS

This manuscript has been reproduced from the microfilm master. UMI films the text directly from the original or copy submitted. Thus, some thesis and dissertation copies are in typewriter face, while others may be from any type of computer printer.

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleedthrough, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send UMI a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

Oversize materials (e.g., maps, drawings, charts) are reproduced by sectioning the original, beginning at the upper left-hand corner and continuing from left to right in equal sections with small overlaps. Each original is also photographed in one exposure and is included in reduced form at the back of the book.

Photographs included in the original manuscript have been reproduced xerographically in this copy. Higher quality 6" x 9" black and white photographic prints are available for any photographs or illustrations appearing in this copy for an additional charge. Contact UMI directly to order.

UMI
University Microfilms International
A Bell & Howell Information Company
300 North Zeeb Road, Ann Arbor, MI 48106-1346 USA
313/761-4700 800/521-0600
Three-dimensional nonlinear analysis of components of reinforced concrete framed structures

Maddipudi, Srinivas, Ph.D.
The Louisiana State University and Agricultural and Mechanical Col., 1992
THREE DIMENSIONAL NONLINEAR ANALYSIS OF COMPONENTS
OF REINFORCED CONCRETE FRAMED STRUCTURES

A Dissertation

Submitted to the Graduate Faculty of the
Louisiana State University and
Agricultural and Mechanical College
in partial fulfillment of the
requirements for the degree of
Doctor of Philosophy

in

The Department of Civil Engineering

by

Srinivas Maddipudi
B.E., Andhra University, India 1985
M.Tech., Indian Institute of Technology - Madras, India 1986
December, 1992
Acknowledgements

This study was conducted under the supervision of Dr. F. Barzegar, former Assistant Professor of Civil Engineering, LSU. The research work was completed under the supervision of Dr. V.K.A. Gopu, Professor of Civil Engineering, LSU. I gratefully acknowledge their guidance and encouragement throughout the course of this research work.

I would like to thank the members of my doctoral advisory committee Dr. L.A. de Bejar, Dr. G.W. Cochran, Professor B.J. Jones, Professor S.S. Iyengar and Professor G.Z. Voyiadjis for reviewing this report and offering helpful suggestions.

Financial support provided for this study by the Department of Civil Engineering and the National Science Foundation is gratefully acknowledged. Computer hardware, IBM 3090/MVS and FPS 500 mainframe systems, and funds provided by the university for conducting this study is gratefully acknowledged. Special thanks are to Dr. W.F. Beyer, Director of SNCC, and Dr. M. Foroozesh for their help in using the computational facilities.

I would like to thank Dr. C. Channakeshava, Research Associate at LSU, for his helpful suggestions during the course of this study. I wish to thank my colleagues - Mr. A. Ramaswamy, Mr. A. Puppala, Mr. R. Echle, Dr. K. Rebello, Mr. S. Sivakumar and Mr. A. Venson for their help during the course of my graduate program.

I am indebted to my parents for their encouragement and financial support throughout the course of my education. I wish to express my heartfelt thanks to my wife Krishna Kumari for her understanding, help and patience.
# Contents

Acknowledgements ............................................................... ii

Table of Contents ............................................................... iii

List of Tables ........................................................................ vi

List of Figures ....................................................................... vii

Abstract ................................................................................ x

1 Introduction ........................................................................ 1
   1.1 General .............................................................................. 1
   1.2 Experimental Investigations ............................................. 3
   1.3 Need for Analytical Studies .............................................. 7
   1.4 Objectives, Scope and Limitations ................................. 8
   1.5 Arrangement of This Report ........................................... 10

2 Method of Analysis ........................................................... 11
   2.1 Introduction ........................................................................ 11
   2.2 General Finite Element Formulation .............................. 11
   2.3 Convergence Criteria ...................................................... 13
   2.4 Choice of Finite Element ............................................... 14
   2.5 Computer Program "INARCS" ......................................... 14

3 Constitutive Modeling of Plain Concrete ............................... 17
   3.1 Introduction ................................................................. 17
   3.2 Simple Formulation of Triaxial Behavior of Concrete .... 18
      3.2.1 Model Description ...................................................... 19
      3.2.2 Stress-Strain Relations ........................................... 22
      3.2.3 Coupling Modulus H ............................................... 28
      3.2.4 Coupling Modulus Y ............................................... 28
      3.2.5 Constitutive Model Input Parameters ....................... 28
<table>
<thead>
<tr>
<th>Chapter</th>
<th>Title</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.3</td>
<td>Failure Criteria for Concrete</td>
<td>29</td>
</tr>
<tr>
<td>3.4</td>
<td>Constitutive Model Verification</td>
<td>33</td>
</tr>
<tr>
<td>3.4.1</td>
<td>Uniaxial Compression</td>
<td>33</td>
</tr>
<tr>
<td>3.4.2</td>
<td>Biaxial Compression</td>
<td>34</td>
</tr>
<tr>
<td>3.4.3</td>
<td>Biaxial Tension-Compression</td>
<td>36</td>
</tr>
<tr>
<td>3.4.4</td>
<td>Triaxial Compression</td>
<td>37</td>
</tr>
<tr>
<td>3.4.4.1</td>
<td>Proportional Loading</td>
<td>37</td>
</tr>
<tr>
<td>3.4.4.2</td>
<td>Non-proportional loading</td>
<td>38</td>
</tr>
<tr>
<td>4</td>
<td>Constitutive Modeling of Cracked Concrete</td>
<td>41</td>
</tr>
<tr>
<td>4.1</td>
<td>Introduction</td>
<td>41</td>
</tr>
<tr>
<td>4.2</td>
<td>Criterion for Cracking</td>
<td>41</td>
</tr>
<tr>
<td>4.3</td>
<td>Cracking Representations</td>
<td>42</td>
</tr>
<tr>
<td>4.4</td>
<td>Smeared Crack Concept</td>
<td>44</td>
</tr>
<tr>
<td>4.4.1</td>
<td>Strain Decomposition</td>
<td>44</td>
</tr>
<tr>
<td>4.4.2</td>
<td>Development of Constitutive Model</td>
<td>47</td>
</tr>
<tr>
<td>4.4.2.1</td>
<td>Crack Stiffness Parameters</td>
<td>50</td>
</tr>
<tr>
<td>4.5</td>
<td>Consistent Characteristic Length</td>
<td>55</td>
</tr>
<tr>
<td>4.6</td>
<td>Verification Studies</td>
<td>60</td>
</tr>
<tr>
<td>4.6.1</td>
<td>Constant Stress Field</td>
<td>60</td>
</tr>
<tr>
<td>4.6.2</td>
<td>Plain Concrete Notched Beam</td>
<td>61</td>
</tr>
<tr>
<td>5</td>
<td>Constitutive Modeling of Reinforcing Steel and Bond Slip</td>
<td>70</td>
</tr>
<tr>
<td>5.1</td>
<td>Introduction</td>
<td>70</td>
</tr>
<tr>
<td>5.2</td>
<td>Constitutive Relationship for Steel</td>
<td>70</td>
</tr>
<tr>
<td>5.3</td>
<td>Reinforcement Representations</td>
<td>71</td>
</tr>
<tr>
<td>5.4</td>
<td>Choice of Reinforcement Representation</td>
<td>74</td>
</tr>
<tr>
<td>5.5</td>
<td>Bond between Concrete and Reinforcing Steel</td>
<td>77</td>
</tr>
<tr>
<td>5.6</td>
<td>Finite Element Modeling of Embedded Reinforcement and Bond Slip</td>
<td>78</td>
</tr>
<tr>
<td>5.6.1</td>
<td>Geometric Formulation</td>
<td>79</td>
</tr>
<tr>
<td>5.6.2</td>
<td>Evaluation of Strain along the Rebar</td>
<td>84</td>
</tr>
<tr>
<td>5.6.3</td>
<td>Virtual Work Formulation</td>
<td>86</td>
</tr>
<tr>
<td>5.7</td>
<td>Determination of Normalized Coordinates of a Point on the Rebar</td>
<td>88</td>
</tr>
<tr>
<td>5.8</td>
<td>Verification Analyses</td>
<td>89</td>
</tr>
<tr>
<td>5.8.1</td>
<td>Constant Stress Field</td>
<td>90</td>
</tr>
<tr>
<td>5.8.2</td>
<td>Reinforced Concrete Beam</td>
<td>91</td>
</tr>
<tr>
<td>6</td>
<td>Reinforcement Mesh Mapping</td>
<td>100</td>
</tr>
<tr>
<td>6.1</td>
<td>Introduction</td>
<td>100</td>
</tr>
<tr>
<td>6.2</td>
<td>Mesh Configuration</td>
<td>101</td>
</tr>
<tr>
<td>6.3</td>
<td>Rebar Intersection Points with 3D Finite Element Faces</td>
<td>102</td>
</tr>
</tbody>
</table>
6.4 Procedure to Identify the Concrete Element Containing a Given Point on a Rebar ........................................ 104
6.5 Verification Analysis .................................................. 105

7 Analyses of RC Structural Elements ........................................ 111
7.1 Introduction ........................................................................ 111
7.2 Plain Concrete Beam Subjected to Torsion ............................ 112
  7.2.1 Introduction .................................................................. 112
  7.2.2 Finite Element Idealization ........................................ 112
  7.2.3 Load Application ....................................................... 115
  7.2.4 Results of Analysis and Discussion ............................... 115
7.3 Beam-Column ................................................................. 119
  7.3.1 Introduction .................................................................. 119
  7.3.2 Finite Element Idealization ........................................ 120
  7.3.3 Load Application ....................................................... 120
  7.3.4 Results of Analysis and Discussion ............................... 123
7.4 Reinforced Concrete Column ............................................... 127
  7.4.1 Finite Element Idealization ........................................ 128
  7.4.2 Load Application ....................................................... 128
  7.4.3 Modeling of Crushed Concrete .................................... 128
  7.4.4 Results of Analysis and Discussion ............................... 130
7.5 Beam-Column-Slab Connection ........................................... 131
  7.5.1 Introduction .................................................................. 131
  7.5.2 Finite Element Idealization ........................................ 132
  7.5.3 Load Application, Boundary Conditions and Material Properties .......................................................... 138
  7.5.4 Results of Analysis and Discussion ............................... 141

8 Summary and Conclusions .................................................... 162
8.1 Summary ........................................................................... 162
8.2 Conclusions ....................................................................... 163
8.3 Suggestions for Further Work ............................................ 165

References ............................................................................ 166

Vita ...................................................................................... 173
List of Tables

7.1 Material Properties of Concrete - Beam Column ......................... 123
7.2 Material Properties of Reinforcement - Beam Column .................. 124
7.3 Material Properties of Concrete - RC Column ........................... 127
7.4 Material Properties of Reinforcement - RC Column ..................... 128
7.5 Material Properties of Concrete - RC Connection ....................... 142
7.6 Material Properties of Reinforcement - RC Connection ................ 142
## List of Figures

1.1 Typical Shape of RC Framed Structure Subjected to Lateral Load ........................................ 2
1.2 Typical Deflected Shape of Structure in Plane of Column .................................................. 3
1.3 Exterior and Interior Connections ......................................................................................... 5
1.4 Test Structure ....................................................................................................................... 6
2.1 20 Noded Isoparametric Element ......................................................................................... 15
3.1 Interpretation of $\tau_{xy}$ in 3-D Stress Space (Gerstle 1981b) ............................................ 22
3.2 Relations for Concrete Secant Bulk Modulus (Cedolin 1977) .......................................... 24
3.3 Relations for Concrete Secant Shear Modulus (Cedolin, 1977) ......................................... 25
3.4 Comparison Between the Experimental and Analytical Results for Uniaxial Compression ................................................................................................................................. 27
3.5 Schematic Failure Surface of Concrete in 3D Stress Space ................................................. 30
3.6 Five Parameter Model Fitting of Triaxial Data ..................................................................... 30
3.7 Comparison Between the Experimental and Analytical Results for Uniaxial Compression ................................................................................................................................. 34
3.8 Comparison Between Experimental and Analytical Results for Biaxial Compression .................. 35
3.9 Comparison Between Experimental and Analytical Results for Biaxial Compression .................. 36
3.10 Comparison Between the Experimental and Analytical Results for Tension-Compression ................................................. 37
3.11 Comparison Between Experimental and Analytical Results for Triaxial Compression ................. 38
3.12 Comparison Between the Experimental and Analytical Results for Triaxial Compression ................. 39
3.13 Comparison Between the Experimental and Analytical Results for Triaxial Compression (Non Proportional) ................................................. 39
4.1 Discrete Crack Representation ............................................................................................... 43
4.2 Smeared Crack Representation ............................................................................................ 44
4.3 Resolution of Total Strain of a Fracture Zone into Concrete Strain and Crack Strain .................. 46
4.4 Local Coordinate System and Traction Across a Crack ...................................................... 47
4.5 Crack Interface Stiffnesses ................................................................................................... 51
4.6 Strain Softening Curve and Fracture Energy ........................................... 53
4.7 Idealized Strain Softening Diagram .......................................................... 53
4.8 Computation of $\phi$ Values ..................................................................... 58
4.9 Determination of Characteristic Length in a Square Element ............... 59
4.10 Axial Bar in Constant Stress Field ......................................................... 62
4.11 Force Displacement Response of Axial Bar ........................................... 63
4.12 Unreinforced Concrete Notched Beam ..................................................... 64
4.13 Notched Beam Material Properties ......................................................... 65
4.14 Crack Pattern in Notched Beam ............................................................... 66
4.15 Deformed Shape of the Notched Beam .................................................... 66
4.16 Load-Deformation Response for Coarse mesh ...................................... 68
4.17 Load-Deformation Response Comparison .............................................. 69
5.1 Stress-Strain Curves for Steel Reinforcing Bars (Nilson and Winter 1968) .... 72
5.2 Idealizations for Stress Strain Curves for Reinforcing Steel in Tension and Compression ................................................................. 73
5.3 Alternate Representations of Steel (ASCE 1982) ..................................... 76
5.4 Embedded Representation of Reinforcement ............................................ 80
5.5 Global and Local Coordinates Along a Rebar ........................................ 82
5.6 Reinforced Axial Bar in Constant Stress Field ....................................... 92
5.7 Load Displacement Response of Axial Bar ............................................. 93
5.8 Bond Slip along the Rebar ....................................................................... 93
5.9 Stress Distribution Along the Rebar .......................................................... 94
5.10 Reinforced Concrete Beam without Stirrups ........................................... 95
5.11 Finite Element Mesh for RC Beam .......................................................... 97
5.12 Crack Pattern and Stress Contour in RC Beam ...................................... 98
5.13 Load Deformation Response of Reinforced Concrete Beam ................. 99
6.1 Rebar Intersection with Concrete Finite Element Face .......................... 101
6.2 Column Details ....................................................................................... 106
6.3 Finite Element Model for Column and Reinforcement .......................... 107
6.4 Mapped Rebar Segments for Coarse Mesh ............................................ 109
6.5 Refined FE Mesh for Column and Mapped Segments .......................... 110
7.1 Torsion Beam Details .............................................................................. 113
7.2 Finite Element Idealization of Torsion Beam ......................................... 114
7.3 Deformed Shape of Torsion Beam .......................................................... 115
7.4 Crack Pattern in the Torsion Beam .......................................................... 116
7.5 Maximum Principal Strain Contour for Torsion Beam ......................... 117
7.6 Load-Deformation response of Torsion Beam ....................................... 119
7.7 Details of Beam-Column ....................................................................... 121
7.8 Finite Element Idealization of Beam-Column ....................................... 122
7.9 Reinforcement Embedded in Concrete FE model .................................. 123
7.10 Stress Contour for Beam-Column ......................................................... 125
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.11 Crack Pattern in Beam-Column</td>
<td>126</td>
</tr>
<tr>
<td>7.12 Load Deformation Response for Beam-Column</td>
<td>126</td>
</tr>
<tr>
<td>7.13 Load Deformation Response of RC Column</td>
<td>129</td>
</tr>
<tr>
<td>7.14 Comparison of Experimental and Analytical Stress Strain Relationship</td>
<td>130</td>
</tr>
<tr>
<td>7.15 Specimen Configuration</td>
<td>133</td>
</tr>
<tr>
<td>7.16 Main Beam and Column Reinforcement Details</td>
<td>134</td>
</tr>
<tr>
<td>7.17 Slab Reinforcement Details</td>
<td>135</td>
</tr>
<tr>
<td>7.18 Finite Element Idealization (Elevation and Plan)</td>
<td>136</td>
</tr>
<tr>
<td>7.19 Finite Element Model for the Connection (3D-View)</td>
<td>137</td>
</tr>
<tr>
<td>7.20 3D-Views of Reinforcement Mesh</td>
<td>139</td>
</tr>
<tr>
<td>7.21 3D-View of Reinforcement Embedded in Concrete FE mesh</td>
<td>140</td>
</tr>
<tr>
<td>7.22 Displacement Routine</td>
<td>141</td>
</tr>
<tr>
<td>7.23 Deformed Shape of Connection under Positive Loading</td>
<td>143</td>
</tr>
<tr>
<td>7.24 Minimum Principal Stress Contour for Positive Loading</td>
<td>144</td>
</tr>
<tr>
<td>7.25 Normal Stress Variation in Main Beam for Positive Loading</td>
<td>146</td>
</tr>
<tr>
<td>7.26 Shear Stress Contour for Positive Loading</td>
<td>147</td>
</tr>
<tr>
<td>7.27 Crack Pattern in Slab for Positive Loading</td>
<td>148</td>
</tr>
<tr>
<td>7.28 Crack Pattern in Main Beam and Column for Positive Loading</td>
<td>149</td>
</tr>
<tr>
<td>7.29 Deflected Shape of Connection under Negative Loading</td>
<td>150</td>
</tr>
<tr>
<td>7.30 Normal Stress Contour for Negative Loading</td>
<td>151</td>
</tr>
<tr>
<td>7.31 Shear Stress Contour for Negative Loading</td>
<td>153</td>
</tr>
<tr>
<td>7.32 Crack Pattern in Slab under Negative Loading</td>
<td>154</td>
</tr>
<tr>
<td>7.33 Crack Pattern in Main Beam and Column under Negative Loading</td>
<td>155</td>
</tr>
<tr>
<td>7.34 Crack Pattern in Transverse Beam under Negative Loading</td>
<td>156</td>
</tr>
<tr>
<td>7.35 Transverse Cracks in the Slab</td>
<td>157</td>
</tr>
<tr>
<td>7.36 Torsional Cracks in Transverse Beam</td>
<td>158</td>
</tr>
<tr>
<td>7.37 Failure Surface in the Connection</td>
<td>159</td>
</tr>
<tr>
<td>7.38 Stiffness Degradation of the Connection</td>
<td>160</td>
</tr>
<tr>
<td>7.39 Load Deformation Response of Connection</td>
<td>161</td>
</tr>
</tbody>
</table>
Abstract

This study deals with the three dimensional analysis of reinforced concrete structural members under multiaxial loading conditions. An incremental formulation in conjunction with the finite element method is used to simulate the behavior of reinforced concrete material. A hypoelastic model capable of simulating response of plain concrete under nonproportional loading conditions is employed. A five parameter strength envelope is used to predict the failure of concrete in multiaxial stress states.

A smeared crack model capable of handling multiple non-orthogonal cracks is used to represent the post cracking behavior of concrete. Objectivity in the results of analysis utilizing the smeared cracking approach is achieved by employing a consistent characteristic length in three dimensional applications.

The reinforcement in concrete is simulated using the embedded representation which allowed the bars to be modeled at their exact locations. Slip between concrete and steel is modeled by incorporating an additional degree of freedom, associated with the slip, at the intersection of the rebar with concrete finite element. Ability of the embedded steel segments to simulate the confinement effect on concrete is verified by analyzing an axially loaded reinforced concrete column.

A general mesh mapping procedure that significantly reduces the amount of work involved to prepare the data for finite element models, is proposed and implemented for three dimensional applications. This procedure eliminates the limitations on the choice of the grid for the concrete finite element mesh and simplifies the use of embedded representation in three dimensional applications.

The proposed models are implemented in a special purpose finite element program. The capabilities of the models are explored by simulating a number of experimental test specimens. These examples include, a notched beam, a beam under
torsion, a beam without stirrups, a beam-column, and a beam-column-slab connection. The results of analysis indicate that the effect of concrete cracking and yielding of steel on the behavior of concrete are simulated well. Also, the predicted cracking pattern and failure loads are found to be in good agreement with those obtained from experimental procedures.
Chapter 1

Introduction

1.1 General

Challenges in analysis and design of complex reinforced concrete (RC) structures have prompted the structural engineer to acquire a sound understanding of the behavior of reinforced concrete. In many cases current conventional design methods cannot be relied upon to provide realistic information on load displacement response, ultimate strength and failure mode of RC structures and structural elements to arrive at safe and cost effective designs. This is primarily due to the complexities associated with the development of rational analysis procedures which have necessitated that existing design methods be based on empirical procedures and their underlying experimental data. With the advent of scientific supercomputing and powerful numerical procedures such as the finite element method, it has become feasible to study the complete nonlinear response of RC structures.

The current American Concrete Institute specifications (ACI, 1983) for moment resisting RC framed structures are based on ultimate capacities of structural components. Under critical loadings, such as earthquake-induced lateral loading, the input energy to a RC structure is intended to be dissipated through the inelastic response of its sub-assemblies. Such inelastic design criteria are aimed at providing an economic structure by limiting the sizes of its components. The inelastic deformation in RC framed structures are mainly concentrated in certain critical regions. In multistorey RC frame buildings (figs. 1.1 & 1.2) the beam column connections are critical regions where the inelastic deformations are concentrated.
For this reason, understanding the behavior of these critical regions under different loading conditions through experiments and/or analytical methods is of utmost importance for designing safe and economic framed structures.

Figure 1.1: Typical Shape of RC Framed Structure Subjected to Lateral Load

The behavior of reinforced concrete material is highly nonlinear and the formulation of rational analytical procedures to describe this behavior is very involved. This is due to the difficulties introduced by nonlinearities such as: (1) the stress-strain behavior of plain concrete under multiaxial loadings; (2) stress and/or strain dependent failure criteria for concrete; (3) concrete crushing and post-crushing strain softening behavior; (4) concrete cracking and subsequent need to model the interaction between reinforcement and concrete; and (5) yielding of the reinforcement.
1.2 Experimental Investigations

In multistorey RC frame buildings the beam to column connections may be subjected to considerable inelastic deformations when the structures are subjected to critical loading conditions. Depending on their location in a building, these connections may be an assembly of two or three beams and two columns, as in an exterior connection, or they may be a typical interior connection with members framing in all four orthogonal directions (fig. 1.3).

In order to investigate various aspects of the behavior of RC beam to column connections several experimental studies have been carried out during the last two decades (Hanson and Conner 1967, 1972; Uzumeri and Seckin 1974; Jirsa et al. 1975; Meinheit and Jirsa 1977; Pauly et al. 1975; Viwathanatepa et al. 1979 and Otani et al. 1985). These experiments, however, did not include the slabs, which in a real building are normally cast monolithically with the floor beams and hence contribute to the connection response. Such slabs in an exterior connection,
for example, would increase the flexural capacity of the main beams while imposing torsional moments on the transverse edge beams and thereby influence the confinement of the joint.

A limited number of experiments on isolated beam-column connections including a slab have been recently reported (Durrani and Wight 1982; Ehsani and Wight 1982; Leon 1984; Joglekar et al. 1985; Durrani and Zerbe 1987; Durrani and Wight 1987; Kitayama et al. 1987 and Wolfgram 1989). These studies have demonstrated the effect of slabs on both the stiffness and the strength of the connections tested.

The behavior of beam-column connections with slabs within a full scale RC test building (fig. 1.4) was investigated for the first time in a recent US-Japan cooperative research project. As a part of this research effort a full scale seven-storey RC structure, designed based on a compromise between the US and the Japanese codes of practice, was tested in Japan (Okamoto et al. 1985). The post-test response analyses (Yoshimura and Kurose 1985) indicated that the floor slabs contributed significantly to the lateral load resisting mechanisms and ultimate capacities of the structure. This was mainly due to the fact that the measured effective flange width of beams, which were cast monolithically with the slabs, were much larger than those suggested in the Architectural Institute of Japan Reinforced Concrete Standards (1982) or the ACI code (1983). Such findings were also substantiated by the test results of a \( \frac{1}{6} \) scaled model of the same structure (Bertero et al. 1988) and furthermore, by the results of full scale tests of individual connections identical to those used in the second storey of the full scale test building (Joglekar et al. 1985).

The above studies have improved the level of insight into the response of RC connections. However, the majority of the reported tests were conducted under planar loading conditions and hence are unable to provide the response characteristics under general three-dimensional loading situations.
Figure 1.3: Exterior and Interior Connections
Figure 1.4: Test Structure
1.3 Need for Analytical Studies

Ideally analytical models to simulate a 3-D response of a complete RC buildings with all structural components affecting the response would be valuable. The need for developing such analytical models to predict RC structural behavior relates to their desired supplemental capabilities to predict the sequence of events and modes of failure observed in the test specimens. Analytical studies can provide additional insight into the behavior of tested specimens and may identify different internal mechanisms which contribute to their response. Successful analytical studies can be used in extrapolating the experimental results beyond the range considered. Appropriately formulated and tested models could, to some extent, eliminate the need for constructing and testing expensive test specimens in order to evaluate the prototype structural behavior.

Due to the complexity of the problem, only a few analytical investigations of the beam-column connections have been conducted. Will, Uzemeri and Sinha (1972) analyzed a RC beam-column exterior joint subjected to monotonic loading using the finite element method. In their analysis plane stress rectangular elements were used and the concrete was assumed to exhibit linear stress-strain behavior under compression. Soleimani et al. (1979) performed experimental and analytical investigation on beam-column connections. Their analytical model, ZAP model, idealized all rotations of fixed end at end plastic hinges with rotational springs. The stiffness of beam end zones was reduced to simulate the spread of plastic hinges.

Sheu and Hawkins (1980) developed a grid model for predicting the monotonic hysteretic behavior of slab-column connections transferring moments based on the matrix displacement approach. The isotropic slab was replaced by an equivalent grid. The width of the beams and spacing were arrived at by matching elastic solution with finite element solution. It was found that the width of beam for best match should be column dimension plus effective depth of the slab.

Noguchi (1981) analyzed beam-column connections using nonlinear finite element method. Plane stress analysis using linear strain triangular elements was employed. Discrete crack model with spring linkage elements for bond slip were used
in the analysis. Analytical model predicted a stiff response with high yield strength. However, cracking load and strain distribution in beam were simulated well.

Filippou, Popov and Bertero (1983) developed an analytical model for studying bond slip at RC interior joints under cyclic excitations. The equilibrium equations of bond problem were converted into integral formulations through a weighted residuals approach. A non linear stress strain law for steel which included Bauschinger's effect was used. Different relations were used for confined and unconfined concrete under compression. The concrete strain was assumed to be linearly distributed over the cross-section.

Most of the available analytical studies dealt with two dimensional cases and excluded cracking. The recent experimental investigations of the behavior of RC beam to column connections with slabs, however have not been supplemented with analytical studies. The number of analytical investigations on beam to column connections using the finite element method are limited (ASCE, 1985) and only a few studies dealt with the 3-D response of RC beam-column connections. The reported 3-D analyses have been basically on elastic behavior of connections to investigate the influence of parameters such as the shape of the joints, the width of the connecting beams, etc. on the overall connection response. These 3-D analytical studies have not been extended to investigate the nonlinear response of RC connections.

1.4 Objectives, Scope and Limitations

The difficulties involved in constructing an analytical model to predict the response of RC structures upto failure are multifold. In the material level, many nonlinear, and often interacting, behavior characteristics such as concrete cracking and crushing, aggregate interlock, bond slip, dowel action, time dependent effects of shrinkage and creep and yielding of reinforcement which contribute to structural response must be taken into account. With the recent advances in computer based finite element analysis techniques and their applications to RC structures (ASCE 1982,1985) it is now possible to incorporate several nonlinear behavior aspects in a given analysis.
The main objectives of the present study are:

- Selection of an appropriate three dimensional constitutive model for simulating the precracked response of concrete under multiaxial loading conditions. Formulation of a smeared crack model for simulating the three dimensional postcracking behavior of concrete. The postcracking model is capable of handling multi directional non-orthogonal cracks within the smeared crack concept.

- Investigate the effect of postcracking model on the objectivity of the finite element mesh.

- Formulation of a suitable analytical model to represent reinforcement in concrete finite elements for three dimensional analysis. Investigate the feasibility of considering the slip between concrete and steel in three dimensional finite element analysis.

- Develop a general mesh mapping procedure to aid the data preparation involved in three dimensional finite element models for RC structures.

- Develop a comprehensive computer software to simulate the response of experimentally tested RC framed building subassemblies using the constitutive models developed in this study.

- Evaluate the performance of the implemented analytical models by analyzing a range of test specimens: plain and reinforced concrete beam, column, beam-column and beam-column-slab connection.

The limitations of the present study are:

- The application of the analytical model is limited to a class of problems where only small strains and displacements need to be considered, in other words, geometric nonlinearities are not considered.

- Time dependent effects such as creep and shrinkage and thermal effects have not been considered.

- In modeling the concrete, the influence of cyclic loading has not been considered.
1.5 Arrangement of This Report

Chapter 2 of this report presents the method used to achieve the objectives of the present study, i.e. choice of the finite element and criteria for convergence are discussed.

In chapter 3, available constitutive models for uncracked concrete are reviewed and then the selected material model is described in detail. The implementation of the model for concrete is then verified by comparing the results with experimental observations.

Chapter 4 presents the review of constitutive models for cracked concrete followed by the implementation of the selected model. Two numerical examples, axial bar and notched beam are considered to verify the implemented material model for cracked concrete.

Chapter 5 describes the selection and implementation of the constitutive model for reinforcement and bond slip. A pull out specimen and a reinforced concrete beam are considered for verifying the concepts related to the reinforcement modeling.

Chapter 6 presents a general mesh mapping procedure to aid the data preparation for finite element models. A reinforced concrete column with two finite element meshes is considered to verify the applicability of the mesh mapping procedure.

The capabilities of the developed models in simulating the nonlinear response of reinforced concrete structures is evaluated by considering four additional examples in chapter 7. These examples include beam subjected to torsion, RC column, beam-column, beam-column-slab connection. The results of analysis are compared with the available experimental observations.

Conclusions of the present study are then presented in chapter 8. Possible extension of the present study are also identified.
Chapter 2
Method of Analysis

2.1 Introduction

The finite element method is by far the most powerful and popular numerical tool for the structural analyst. The reasons for this include its flexibility, simplicity, capability of modeling any geometry, loading and boundary conditions and local changes in material, adaptability to nonlinear problems, ease of programming and availability of commercial software. In the present work, finite element method of analysis was used to carry out the numerical simulations of RC structural elements.

2.2 General Finite Element Formulation

With the finite element displacement method a complex structure can be analyzed by treating the structural system as a set of elements interconnected at a finite number of discrete points called nodes (Zeinkiewicz 1979). Generally, compatibility of displacements across the element boundaries is maintained while the force equilibrium is satisfied approximately. Using the virtual work method, the element stiffness matrix and its equivalent load vector are constructed. Upon assembly of the element contributions to the structural stiffness matrix and load vector, the complete finite element equations are generated which then need to be solved for the unknown nodal displacements. Having each element's nodal displacements, the strains and stresses at the sampling points are then calculated.

Following is a brief summary of the procedure as it is followed in the developed computer program:
12

(1) The displacement vector $u^{(i)}$ at any point within the element is expressed in terms of the nodal point displacement vector $U_n^{(i)}$ by means of the assumed displacement interpolation matrix $N^{(i)}$.

$$u^{(i)} = N^{(i)} U_n^{(i)} \quad (2.1)$$

(2) The corresponding element strains are evaluated using the strain-displacement matrix $B^{(i)}$.

$$\varepsilon^{(i)} = B^{(i)} U_n^{(i)} \quad (2.2)$$

(3) Material stress-strain relations at the sampling points are obtained using the constitutive matrix $D$.

$$\sigma^{(i)} = D^{(i)} \varepsilon^{(i)} = D^{(i)} B^{(i)} U_n^{(i)} \quad (2.3)$$

(4) Invoking the principle of virtual work for each element, by imposing non-zero admissible nodal virtual displacements, and equating the external virtual work to the internal virtual work results in the customary expression

$$K^{(i)} U_n^{(i)} = F^{(i)} \quad (2.4)$$

where

$$K^{(i)} = \int_B B^{T(i)} D^{(i)} B^{(i)} \, dv \quad (2.5)$$

is the element stiffness matrix and

$$F^{(i)} = F_0^{(i)} + F_B^{(i)} + F_S^{(i)} + F_N^{(i)} \quad (2.6)$$

is the element nodal force vector. In equation 2.6, $F_0^{(i)}$, $F_B^{(i)}$, and $F_S^{(i)}$ represent the equivalent nodal point forces due to initial stresses, body forces, and surface tractions, respectively. $F_N^{(i)}$ is the vector of applied concentrated loads at the nodes.

(5) The global equilibrium equations obtained from the assembly of individual element contributions are

$$[K]\{U\} = \{F\} \quad (2.7)$$
Due to material nonlinearities, the resulting algebraic equations are highly nonlinear. An iterative scheme is used for solving these equations. The analysis is carried out by loading the structure in small increments. Satisfaction of the material stress-strain relationships (with a reasonable tolerance) at different sampling points within the structure constitutes a converged step. For this purpose each step of the analysis is completed using an iterative approach. In every iteration, equilibrium check is made by calculating the residual forces which result from the unbalanced loads. The residual forces, which are caused by the unsatisfied stress-strain relations at the sampling points, are calculated as the difference between the external applied loads, $F$, and the internal equivalent loads as

$$R = F - J_v B^T \sigma dv$$  \hspace{1cm} (2.8)

Convergence tests are applied to determine the level of residual (unbalanced) loads remaining. If the convergence tests are satisfied, the next load step is processed; otherwise the residual forces are reapplied to the structure and iterations are carried out until the convergence tests are satisfied. In the present study full Newton-Raphson method is employed for the solution scheme, i.e. in every iteration, the material constitutive matrix $[D]$ and the global stiffness matrix $[K]$ are updated.

### 2.3 Convergence Criteria

In any iterative procedure exact satisfaction of equation 2.7 and reduction of the residual forces to zero is almost impossible. As such acceptable limits on the degree of satisfaction of equilibrium equation 2.7 will have to be fixed. These limits refer to any of the following:

(i) Residual force vector

(ii) Incremental displacement vector or

(iii) Change in energy.

Since the first two quantities are vectors, either a limit on their absolute maximum value or a limit on some norm is used. The two convergence tests used in
this study to terminate the iterative solution process are:

\[
\| \{ R \} \| < 0.03 \| \{ \Delta P \} \| \quad (2.9)
\]

\[
\text{MAX} | \{ R \} | < 0.02 \| \{ \Delta P \} \| \quad (2.10)
\]

where \( \{ R \} \) is the residual load vector and \( \{ \Delta P \} \) is the applied incremental load vector. The first test, (eq. 2.9), compares Euclidean norm (square root of sum of squares) of the residual forces and applied incremental load vector and represents an average measure of equilibrium. The second test, (eq. 2.10), detects any highly localized residual loads that could be missed by a vector norm computations. Both tests must be satisfied for acceptance of a solution.

2.4 Choice of Finite Element

In the present study three dimensional 20 noded isoparametric elements (fig. 2.1) were used to represent the concrete in finite element models. The use of the isoparametric finite elements has been shown to be effective in most practical analyses (Ergataudis 1968). It has been observed that the 20-noded hexahedral element is excessively stiff if used with a 3x3x3 Gaussian numerical integration order for stiffness calculation. The order of numerical integration could be reduced to remedy this shortcoming. Therefore, a network of reduced 15 integration points (Irons 1973), symmetrically distributed within the element were used in this study. Such a scheme was not Gauss optimal but employed satisfactorily (Sarne 1975).

In the present research the reduction in the number of integration points for element stiffness calculations has reduced the computational time by approximately 40%.

2.5 Computer Program "INARCS"

To carry out the numerical simulations, the computer software for reinforced concrete material models was implemented into a special purpose finite element program INARCS (Incremental Nonlinear Analysis of Reinforced Concrete Structures).
The basic elastic finite element analysis modules employed in INARCS were selected from work of Faroozesh (1989).

Efficient implementation of the concepts, presented in this report, in the context of finite element software calls for the computer science concepts such as structured data. Most engineering computer programs have been and are being written in FORTRAN. All the concepts used in this study have been implemented in a finite element program (INARCS) written in FORTRAN. The data to be processed includes the element nodal coordinates, element connectivity, material parameters for constitutive model, topology of the reinforcement before and after mapping.

Use of the structured data may help in making a finite element program implementation easy and efficient. Because of its design, FORTRAN does not encourage the use of data structures other than the array. However, by borrowing the concepts used in other languages such as Pascal, FORTRAN code can be written to support data structures not supported by the language itself. In the context of finite element structural analysis, all the data related to a particular finite element could be
represented as a record consisting of a collection of fields. Each field may correspond to one of the data elements associated with the finite element under consideration. For example a record of an element data may consist of fields such as element identification number, number of sampling points, material identifier, number of reinforcing bars contained in the element etc. In turn each field of this record point to the corresponding arrays defining the data. For example the element identification number points to the index of the array containing the element connectivity data. However, since the FORTRAN does not support the data type record, it is emulated in the present work with integer arrays. Experience in using the emulated record type for handling the data has shown data structures to be extremely useful in developing the structural analysis code.
Chapter 3

Constitutive Modeling of Plain Concrete

3.1 Introduction

A rational analysis and, hence, design of complex RC structures through computer-based methods is often limited by the lack of adequate and simple material models for plain concrete. This is particularly true for situations where concrete is subjected to multi-dimensional loadings. A report on the finite element analyses of RC structures (ASCE 1982) shows that despite the general recognition of the nonlinear material behavior of concrete, most commercial programs use linear, elastic constitutive models for concrete. This may be attributed to the difficulties encountered in assessing various parameters involved with complex material models, and in their computer implementation.

Ideally a constitutive model for concrete should reflect definite strain hardening characteristics before failure, the failure itself, as well as some strain softening in the post-failure regime. The model should also perform satisfactorily under different states of stresses applied proportionally, as well as non-proportionally, be capable of handling unloading/reloading, and yet be simple, flexible and numerically feasible. Finally the material model must be easy to calibrate to a particular type of concrete.

A comprehensive list of constitutive models for simulating the pre-cracked response of concrete is given in the ASCE Task Committee Report (1982). These models may be classified as: (1) elasticity based models, (2) plasticity based models, (3) plastic-fracturing models, (4) endochronic models. Based on various hypotheses, these models express the stress-strain relationships for concrete in terms of material
and loading parameters calibrated from test results. Depending on the selected constitutive model for concrete, it would be necessary to complement the model with a suitable "failure" or "ultimate strength" envelope. A rather comprehensive description of such models is given by Chen (1982).

This chapter presents a detailed description of the constitutive model selected for modeling the concrete material, the numerical results obtained and a comparison of computed values with experimental observations.

3.2 Simple Formulation of Triaxial Behavior of Concrete

During the first phase of the present research effort related to finite element simulation of the three dimensional response of components RC frame buildings under monotonic loading, a comprehensive study of the available triaxial constitutive models for concrete was undertaken. An elasticity-based model proposed by Stankowski and Gerstle (1985) was selected and modified for finite element implementation. In a recent comparative evaluation of the performance of various constitutive models for triaxially loaded concrete, Eberhardsteiner (1987) has shown that the hypoelastic model proposed by Stankowski and Gerstle (1985) gives reasonably good results. The model uses an octahedral representation of the multiaxial stress-strain relations for plain concrete. In this hypoelastic model, based on incremental formulation, the concrete behavior is represented by variable tangent bulk and shear moduli. The formulation accounts for full coupling between hydrostatic and deviatoric effects, requires only a minimum of material information for its implementation.

The foregoing constitutive model was implemented in a specialized finite element program to analyze the responses of a number of test specimens. The following is an outline of the model formulation and its capabilities in predicting the behavior of various plain concrete specimens subjected to different stress conditions.
3.2.1 Model Description

In the model proposed by Stankowski and Gerstle (1985) an octahedral representation of the multiaxial stress-strain relations for concrete assuming isotropic, nonlinear behavior was used. This formulation relates octahedral stress and strain increments using variable tangent bulk and shear moduli.

The octahedral normal or hydrostatic strain increment $\Delta \varepsilon_0$ and the corresponding stress increment $\Delta \sigma_0$ are defined in terms of principal stress and strain increments $\Delta \sigma_i$ and $\Delta \varepsilon_i$ as

$$
\Delta \sigma_0 = \frac{(\Delta \sigma_1 + \Delta \sigma_2 + \Delta \sigma_3)}{3} \quad (3.1)
$$

$$
\Delta \varepsilon_0 = \frac{(\Delta \varepsilon_1 + \Delta \varepsilon_2 + \Delta \varepsilon_3)}{3} \quad (3.2)
$$

For proportional loading the octahedral shear stress increment $\Delta \tau_0$ and the corresponding strain increment $\Delta \gamma_0$ are defined in terms of principal stresses and strains as

$$
\Delta \tau_0 = \frac{[\Delta \sigma_1 - \Delta \sigma_2]^2 + (\Delta \sigma_2 - \Delta \sigma_3)^2 + (\Delta \sigma_3 - \Delta \sigma_1)^2]^{1/2}}{3} \quad (3.3)
$$

$$
\Delta \gamma_0 = \frac{[\Delta \varepsilon_1 - \Delta \varepsilon_2]^2 + (\Delta \varepsilon_2 - \Delta \varepsilon_3)^2 + (\Delta \varepsilon_3 - \Delta \varepsilon_1)^2]^{1/2}}{3} \quad (3.4)
$$

For non-proportional loading the incremental octahedral shear stress, expressed in terms of deviatoric stresses $S_i$ is defined as

$$
\Delta \tau_0 = \frac{(S_{11}\Delta S_{11} + S_{22}\Delta S_{22} + S_{33}\Delta S_{33})}{3\tau_0} \quad (3.5)
$$

which can also be expressed in terms of principal stresses as

$$
\Delta \tau_0 = \frac{1}{9\tau_0}[\sigma_1(2\Delta \sigma_1 - \Delta \sigma_2 - \Delta \sigma_3) + \sigma_2(2\Delta \sigma_2 - \Delta \sigma_1 - \Delta \sigma_3) + \sigma_3(2\Delta \sigma_3 - \Delta \sigma_1 - \Delta \sigma_2)] \quad (3.6)
$$

The expression for the incremental octahedral shear strains in terms of deviatoric strains $\varepsilon_i$ is
\[
\Delta \gamma_0 = \frac{[e_{11}\Delta \varepsilon_{11} + e_{22}\Delta \varepsilon_{22} + e_{33}\Delta \varepsilon_{33}]}{3\gamma_0}
\] (3.7)

or in terms of principal strains

\[
\Delta \gamma_0 = \frac{1}{9\gamma_0}[e_1(2\Delta \varepsilon_1 - \Delta \varepsilon_2 - \Delta \varepsilon_3) + e_2(2\Delta \varepsilon_2 - \Delta \varepsilon_3 - \Delta \varepsilon_1) + e_3(2\Delta \varepsilon_3 - \Delta \varepsilon_1 - \Delta \varepsilon_2)]
\] (3.8)

The octahedral shear strain increment depends on the current strains as well as on the increments of principal strains. For given principal strain increments it is required to evaluate the corresponding principal stress increments. Octahedral stresses and strains are related by the following constitutive relation (Gerstle 1981)

\[
\begin{pmatrix}
\Delta \varepsilon_0 \\
\Delta \gamma_0
\end{pmatrix} = 
\begin{pmatrix}
\frac{1}{3K} & \frac{1}{H} \\
\frac{1}{\mu} & \frac{1}{2G}
\end{pmatrix}
\begin{pmatrix}
\Delta \sigma_0 \\
\Delta \tau_0
\end{pmatrix}
\] (3.9)

The four moduli $K, G, H$ and $Y$ are tangent moduli which depend on the octahedral stresses and strains. They will be discussed in the next sections.

Since thus far only two stress and strain invariants have been considered, Eqs. 3.1 and 3.6 provide insufficient conditions for the determination of the three unknown principal stress increments. The additional condition needed is obtained based on the assumed coincidence of the deviatoric stress and strain increment vectors (Gerstle 1981) as follows

\[
\frac{\Delta \varepsilon_2 - \Delta \varepsilon_0}{\Delta \varepsilon_3 - \Delta \varepsilon_0} = B
\] (3.10)

or

\[
\frac{\Delta \varepsilon_2 - \Delta \varepsilon_0}{\Delta \varepsilon_1 - \Delta \varepsilon_0} = B
\] (3.11)

Since the strain increments are given, $B$ is a known quantity for each load step. Following this assumption, we solve Eqs. 3.1, 3.6 and 3.11 simultaneously and arrive at the desired principal stress increments

\[
\Delta \sigma_1 = \Delta \sigma_0 + e_1 \Delta \tau_0
\] (3.12)
\[ \Delta \sigma_2 = \Delta \sigma_0 + B.c_1 \Delta \tau_0 \]  
(3.13)

\[ \Delta \sigma_3 = \Delta \sigma_0 - (1 + B)c_1 \Delta \tau_0 \]  
(3.14)

in which

\[ c_1 = \sqrt{\frac{3}{2(1 + B + B^2)}} \]  
(3.15)

for proportional loading and

\[ c_1 = \frac{3\tau_0}{\sigma_1 + B\sigma_2 - (1 + B)\sigma_3} \]  
(3.16)

for non-proportional loading.

Eqs. 3.12, 3.13 and 3.14 may be written in a matrix form as

\[
\begin{pmatrix}
\Delta \sigma_1 \\
\Delta \sigma_2 \\
\Delta \sigma_3
\end{pmatrix} =
\begin{pmatrix}
1 \\
1 \\
1
\end{pmatrix}
\begin{pmatrix}
c_1 \\
Bc_1 \\
-(1 + B)c_1
\end{pmatrix}
\begin{pmatrix}
\Delta \sigma_0 \\
\Delta \sigma_0 \\
\Delta \sigma_0
\end{pmatrix}
\]  
(3.17)

In order to express the incremental octahedral stresses in terms of incremental octahedral strains, Eqn. 3.9 is inverted to give

\[
\begin{pmatrix}
\Delta \sigma_0 \\
\Delta \tau_0
\end{pmatrix} = \frac{1}{D} \begin{pmatrix}
\frac{1}{2G} & -\frac{1}{H} \\
-\frac{1}{Y} & \frac{1}{3K}
\end{pmatrix}
\begin{pmatrix}
\Delta \epsilon_0 \\
\Delta \gamma_0
\end{pmatrix}
\]  
(3.18)

where

\[ D = \left[ \frac{1}{3K} \ast \frac{1}{2G} \right] - \left[ \frac{1}{H} \ast \frac{1}{Y} \right] \]  
(3.19)

Also Eqs. 3.2 and 3.8, may be written in a matrix form as

\[
\begin{pmatrix}
\Delta \epsilon_0 \\
\Delta \gamma_0
\end{pmatrix} =
\begin{pmatrix}
\frac{1}{3} & \frac{1}{3} & \frac{1}{3} \\
\frac{1}{2G} & -\frac{1}{2G} & -\frac{1}{2G} \\
\frac{1}{2G} & -\frac{1}{2G} & -\frac{1}{2G}
\end{pmatrix}
\begin{pmatrix}
\Delta \epsilon_1 \\
\Delta \epsilon_2 \\
\Delta \epsilon_3
\end{pmatrix}
\]  
(3.20)
3.2.2 Stress-Strain Relations

As discussed above, in order to express the stress-strain relations for concrete, four moduli, $K$, $G$, $H$, and $Y$ need to be determined. In formulating the biaxial version of the present triaxial model, Gerstle (1981a) proposed the following expressions for the tangent shear modulus of concrete

$$G_T = G_0(1 - \frac{\tau_o}{\tau_{ou}})$$

(3.21)

in which $\tau_o$ is the current deviatoric stress, and $\tau_{ou}$ is the deviatoric strength which may be interpreted in two ways as (fig. 3.1).

![Figure 3.1: Interpretation of $\tau_{ou}$ in 3-D Stress Space (Gerstle 1981b)](image)

$$\tau_{ou} = \tau_{ou_1}$$

(3.22)

and

$$\tau_{ou} = \tau_{ou_2}$$

(3.23)
In the present study the use of the lower strength value (eqn. 3.22) has led to a value of $G_t$ which is very low, resulting in a very ductile behavior for all cases of loading. However, the interpretation of $\tau_{ou}$ according to eqn. 3.23 resulted in failure to achieve numerical convergence under both biaxial and triaxial loadings. These difficulties were also been reported by Stankowski (1985). While it may be possible to express $\tau_{ou}$ differently for various loading conditions (Gerstle 1981a, 1981b), it is desirable, from the computational standpoint, to have a common definition of $\tau_{ou}$ suitable for all stress histories.

In an attempt to remedy this shortcoming, alternatives were investigated in the present study. General expressions describing behavior of concrete under triaxial loading are considered as possible alternative to represent the variation in the stiffness of the concrete material. Cedolin (1977) has proposed the following equations for the secant bulk and shear moduli for a general loading condition (figs. 3.2 and 3.3)

\[
\frac{K_T}{K_0} = 0.85\left[1 - \frac{(ln(2.5))\varepsilon_0}{0.0014}\right](2.5)^{0.54\gamma} + 0.15
\]

\[
\frac{G_T}{G_0} = 0.81\left[1 - \frac{(ln(2.0))\gamma_0}{0.002}\right](2.0)^{0.54\gamma} - 4\gamma_0 + 0.19
\]

in which $K_T$ and $G_T$ are the tangent moduli and $K_0$ and $G_0$ denote initial moduli.

Kotsovos and Newman (1979) have observed that the predicted stresses using the Cedolin's expressions are satisfactory only up to about 70% of the ultimate load. Based on the triaxial test data, they proposed the following expressions for $K_s$ and $G_s$ (secant moduli), which are independent of the deviatoric strength

\[
\frac{K_s}{K_0} = \frac{1}{1 + 0.52(\frac{\tau_{ou}}{K_0})^{1.99}}
\]

\[
\frac{G_s}{G_0} = \frac{1}{1 + 3.57(\frac{\tau_{ou}}{G_0})^{1.7}}
\]

where $K_0$ and $G_0$ are the initial bulk and shear moduli, respectively.

By differentiating the above equations the tangent bulk and shear moduli are expressed as
Figure 3.2: Relations for Concrete Secant Bulk Modulus (Cedolin 1977)
Figure 3.3: Relations for Concrete Secant Shear Modulus (Cedolin, 1977)
\[
\frac{K_T}{K_o} = \frac{1}{(1 + 2^{b-1} A)} \ for \ \frac{\sigma_o}{f'_c} \geq 2
\]  
(3.29)

in which

\[A = 0.516 \ for \ f'_c \leq 32 \ MPa\]  
(3.30)

\[A = \frac{1}{(1 + 0.0027(f'_c - 32)^{2.397})} \ for \ f'_c > 32 \ MPa\]  
(3.31)

\[b = 2.0 + 1.81 \times 10^{-8} f''_{c,461}\]  
(3.32)

\[\frac{G_T}{G_o} = \frac{1}{[1 + cd(f'_c)^{d-1}]}\]  
(3.33)

in which

\[c = 3.573 \ for \ f'_c \leq 32 \ MPa\]  
(3.34)

\[c = 3.573\left[\frac{1}{1 + 0.0134(f'_c - 32)^{1.414}}\right] \ for \ f'_c > 32 \ MPa\]  
(3.35)

\[d = 2.12 + 0.0183 f'_c \ for \ f'_c < 32 \ MPa\]  
(3.36)

\[d = 2.7 \ for \ f'_c \geq 32 \ MPa\]  
(3.37)

Comparison of the deformational behavior for the uniaxial compression test as predicted using the expressions given by Cedolin (Eqs. 3.24 and 3.25) and Kotsovos (Eqs. 3.29- 3.37) for \(K_T\) and \(G_T\) is shown in figure 3.4. As could be seen, better correlation was obtained using the expressions given by Kotsovos. In the present study Eqs. 3.29- 3.37 were used consistently to formulate the stress-strain relationships for concrete under multiaxial loading. In section 3.4 it will be shown that the use of these expressions resulted in satisfactory predictions of the test results for a variety of analyzed concrete specimens subjected to general loading conditions.
Figure 3.4: Comparison Between the Experimental and Analytical Results for Uniaxial Compression
3.2.3 Coupling Modulus H

Gerstle (1981a) observed that volume contraction of concrete occurred under pure deviatoric stress conditions and that this coupling between deviatoric stress and volumetric strain increased with hydrostatic stresses. This coupling is accounted for by using the modulus H in equation 3.9, which is based on the experimental data and is given as

\[ H = [10 + \frac{300}{\sigma_0 - 10}] \times 10^3 \, MPa \text{ for } \sigma_0 > 10 \, MPa \]  

(3.38)

3.2.4 Coupling Modulus Y

It was shown by Scavuzzo et al. (1983) that for a pure hydrostatic stress increment, at a distance \( r_o \) from the hydrostatic axis (fig. 3.1), deviatoric strain increments occurred in the material. This phenomenon is accounted for by using the coupling modulus Y in eqn. 3.9. This modulus, which is based on the preliminary experimental data (Scavuzzo 1983), is given as

\[ Y = \frac{3.93 \times 10^6}{\tau_0^2} \, MPa \]  

(3.39)

which implies that coupling would vanish for stress increments along the hydrostatic axis.

3.2.5 Constitutive Model Input Parameters

The following concrete properties were needed as input data for the employed constitutive model:

(a) Uniaxial compressive strength \( f'_c \)
(b) Modulus of elasticity \( E \)
(c) Poisson’s ratio \( \nu \)

With the above parameters, the initial bulk modulus \( K_0 \) and the initial shear modulus \( G_0 \) could be determined as follows (Gerstle 1981a)

\[ K_0 = \frac{E}{3(1 - 2\nu)} \]  

(3.40)
In conjunction with the hypoelastic constitutive model, 'ultimate strength' or 'failure' criteria are needed for a complete characterization of concrete material behavior. One method of representing the general functional form of the failure surface (Fig. 3.1) is to use the principal stresses as

\[ F(\sigma_1, \sigma_2, \sigma_3) = 0 \]  

As shown in fig. 3.1, the diagonal which has equal distances from the three principal axes is called the hydrostatic axis. The plane perpendicular to this axis is called the deviatoric plane. To simplify the description of the failure surface, an angular coordinate \( \theta \), called the angle of similarity, which lies in the deviatoric plane may be used (fig. 3.5). The meridians of the failure surface are the intersection curves between the failure surface and the meridian planes, which contain the hydrostatic axis and are oriented at different angles \( \theta \) with respect to the deviatoric plane. The two meridian planes corresponding to \( \theta = 0^\circ \) and \( \theta = 60^\circ \) are called the tensile meridian and the compressive meridian, respectively (fig. 3.6).

In equation 3.42 the three principal stresses can also be expressed in terms of the three principal-stress invariants \( I_1, J_2, \) and \( J_3 \). Alternatively, Eqn. 3.42 may be written as

\[ F(I_1, J_2, J_3) = 0 \]  

In describing the failure surface, the third stress invariant \( J_3 \) can be related to the angle of similarity \( \theta \), through

\[ \cos 3\theta = \frac{3\sqrt{3}}{2} \frac{J_3}{J_2^{3/2}} \]  

\[ G_0 = \frac{E}{2(1 + \nu)} \]  

(3.41)
Figure 3.5: Schematic Failure Surface of Concrete in 3D Stress Space

(a) Hydrostatic Section (Launay and Gachon, 1972).
(b) Deviatoric Sections (William and Warnke, 1975)

Figure 3.6: Five Parameter Model Fitting of Triaxial Data
In the present study the five parameter failure surface proposed by Willam and Warnke (1975) was used for predicting the ultimate strength of concrete. This failure model is complex and requires considerable computational effort, but yields results which compare extremely well with the experimental data obtained for a wide range of stress combinations and intensities. This model contains the effect of all the three stress invariants and possesses the observed features of the failure surface such as smoothness, symmetry, convexity, and curved meridians. The model establishes a failure surface with curved meridians in which the generators are approximated by a second order parabola along $\theta = 0^\circ$ (tensile meridian) and $\theta = 60^\circ$ (compressive meridian) with a common apex at the hydrostatic axis (Fig. 3.6a). The intersections of this failure surface and the deviatoric planes, between the two tensile and compressive meridians ($0 < \theta < 60$), are represented as parts of elliptic curves as shown in Fig. 3.6b.

With this failure surface the ultimate strength of the material can be predicted if the applied stresses satisfy the following condition

$$f(\sigma_m, \tau_m, \theta) = \frac{1}{r(\sigma_m, \theta)} \frac{\tau_m}{f'_c} - 1 = 0$$  \hspace{1cm} (3.45)

where

$$\tau_m^2 = \frac{3}{5} \tau_{\text{oct}}^2 = \frac{2}{5} J_2$$  \hspace{1cm} (3.46)

$$\sigma_m = \frac{I_1}{3}$$  \hspace{1cm} (3.47)

$$r(\sigma_m, \theta) = \frac{1}{\sqrt{5} f'_c} r(\sigma_m, \theta)$$  \hspace{1cm} (3.48)

in which

$$r(\sigma_m, \theta) = \frac{P + Q}{(4r_c^2 - r_t^2) \cos^2 \theta + (r_c - 2r_t^2)}$$  \hspace{1cm} (3.49)

where

$$P = 2r_c(r_c^2 - r_t^2) \cos \theta$$  \hspace{1cm} (3.50)
\[ Q = \tau_c(2\tau_t - \tau_c) \times [4(\tau_c^2 - \tau_t^2)\cos^2\theta + 5\tau_t^2 - 4\tau_t\tau_c]^{1/2} \quad (3.51) \]

\[ \cos\theta = \frac{2\sigma_1 - \sigma_2 - \sigma_3}{\sqrt{2[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]^{1/2}}} \quad (3.52) \]

The parabolic meridians \( \tau_t \) and \( \tau_c \) are given as follows:

\[ \frac{\tau_t}{\sqrt{5}f_c'} = a_0 + a_1 \frac{\sigma_m}{f_c'} + a_2 (\frac{\sigma_m}{f_c'})^2 \cos\theta = 0^\circ \quad (3.53) \]

\[ \frac{\tau_c}{\sqrt{5}f_c'} = b_0 + b_1 \frac{\sigma_m}{f_c'} + b_2 (\frac{\sigma_m}{f_c'})^2 \cos\theta = 60^\circ \quad (3.54) \]

where \( a_0, a_1, a_2, b_0, b_1 \) and \( b_2 \) depend on the five parameters of the model which are obtained from the test data.

The failure surface in the deviatoric plane resembles a tetrahedron in the low compression regime with increasing bulge at higher hydrostatic compression, approximating a circular cone asymptotically (fig. 3.6b). This failure model shows close agreement for both low and high pressure regimes along both tensile and compressive meridians, and for deviatoric sections.

The following parameters are needed as model input for establishing the failure surface of a specific concrete material:

1. The uniaxial compressive strength \( (f_c') \),
2. The uniaxial tensile strength \( (f_t') \),
3. The equal biaxial compressive strength \( (f_m') \),
4. The high-compressive-stress point on the tensile meridian, and
5. The high-compressive-stress point on the compressive meridian.

In general, all of the above parameters are not readily available. They also vary for different types of concrete. In the present work, the analytical failure surfaces for different concrete materials were calibrated using a set of data based on the experimental work by Launay and Gachon (1972). When the experimental data of Balmer (1949) were used instead, the predicted values of ultimate strength did not change significantly. For more accurate modeling of the triaxial behavior of a particular concrete material it is imperative that the five required parameters specific to the type of the concrete under investigation be obtained.
3.4 Constitutive Model Verification

A series of verification studies of the implemented concrete constitutive model were carried out and the results of the analyses are presented in this section. In the process of this verification, an automatic check on the numerical procedures, possible sources of error, and the program flow was also obtained. In each analysis a single 20-noded finite element with 15-point quadrature rule (Irons 1971) was employed. Experimental loads were simulated as being uniformly distributed and applied on the appropriate faces of the element.

In order to fully scrutinize the implemented model, test results from different sources were selected for analysis.

3.4.1 Uniaxial Compression

The stress-strain response predicted by the employed model is compared with the experimental data of Kupfer (1969) in fig. 3.4. This also shows the comparison between the predicted responses using the expressions for $G_T$ and $K_T$ given by Cedolin (1977), (eqs. 3.24 and 3.25), and those by Kotsovos and Newman (1979), (eqs. 3.29-3.3). As discussed earlier, the use of Cedolin’s expressions for tangent shear and bulk moduli, yielded a stiffer response when compressive stresses exceeded about 70% of the uniaxial compressive strength $f'_c$. On the other hand, using Kotsovos’ expressions a good correlation was obtained between the analytical and experimental results in both the major and minor principal directions (fig. 3.4). Nevertheless, at the ultimate, a slightly stiffer response was predicted by Kotsovos’ expressions which may be attributed to the fact that the value of $G_T$ does not vanish at ultimate strength and a small finite value was retained for $G_T$. This, however, did not have a major influence on the deformations for stress levels below 95% of the ultimate.

Fig. 3.7 shows the comparison between the deformational behavior predicted by the analytical model (using $K_T$ and $G_T$ as given by Kotsovos) and the experimental data of Stankowski (1985) for a lower strength concrete with $f'_c = 3.5$
ksi (24.1 MPa). A very good correlation was obtained for stress levels up to 90% of the ultimate strength. Again, a slightly stiffer behavior was simulated near the ultimate. This may be insignificant considering the scatter in the experimental data as reported by Stankowski (1985).

![Graph](image)

Figure 3.7: Comparison Between the Experimental and Analytical Results for Uniaxial Compression

### 3.4.2 Biaxial Compression

The analytical and experimental (Kupfer, 1969) results for equal biaxial compression are compared in fig. 3.8. As could be seen a slightly stiffer response was obtained for stress levels reaching the ultimate. Also in the minor principal direction (direction 2), a slightly softer behavior was predicted for stress levels between 40 and 80% of the ultimate.
Figure 3.8: Comparison Between Experimental and Analytical Results for Biaxial Compression
For further comparison, the experimental results from the test series carried out by Schickert and Winkler (1977) at the Federal Material Testing Laboratory at Berlin (BAM) were simulated. Fig. 3.9 shows the comparison between the analytical and experimental results for a stress ratio of -2/-3. The predicted results were within the experimental scatter (shaded areas in fig. 3.9). However, the predicted behavior was slightly on the stiffer side of the scatter.

\[ \sigma_1(\text{ksi}) \]

Fig. 3.10 shows the comparison between the analytical and experimental (Kupfer 1969) results for a stress ratio of -1/0.052. The predicted ultimate strength was approximately 8% lower than the experimental value. An excellent correlation is obtained between the analytical and experimental results in the major principal direction (direction 1).

### 3.4.3 Biaxial Tension-Compression

Fig. 3.10 shows the comparison between the analytical and experimental (Kupfer 1969) results for a stress ratio of -1/0.052. The predicted ultimate strength was approximately 8% lower than the experimental value. An excellent correlation is obtained between the analytical and experimental results in the major principal direction (direction 1).
3.4.4 Triaxial Compression

The results of triaxial tests carried out by Linse and Aschl (1976) at the Technical University of Munich (TUM) in Germany were selected for analysis. Both proportional and non-proportional loadings were included.

3.4.4.1 Proportional Loading

Fig. 3.11 shows the comparison between the analytical and experimental results for the proportionally applied loading with stress ratios of -1:-0.2:-0.2. A very good correlation was obtained in the major principal direction (direction 1) up to a stress level of 85% of the ultimate, after which a stiff behavior was observed. In the minor principal directions (directions 2 and 3) the correlation is very good.
for the full stress range reaching the ultimate. The reported experimental scatter should also be given due consideration in the above comparison. Fig. 3.12 shows the comparison between the analytical and experimental results for a stress ratio of -1:-1:-0.15. For this test, the reported scatter in data was considerable, but still an observation could be made that the correlation was good for low levels of stress (up to 80% of ultimate) and a slightly softer response was predicted at higher stress levels.

\[ \sigma_1 (\text{ksi}) \]

![Graph showing comparison between analytical and experimental results for triaxial compression.](image)

Figure 3.11: Comparison Between Experimental and Analytical Results for Triaxial Compression

### 3.4.4.2 Non-proportional loading

To illustrate the capabilities of the present analytical model in predicting the general triaxial behavior of concrete, the response of a concrete test specimen subjected to non-proportional triaxial loading was analyzed. During testing the specimen was loaded hydrostatically to \( \sigma_0 = 3.9 \text{ ksi} \) followed by an increase of
Figure 3.12: Comparison Between the Experimental and Analytical Results for Triaxial Compression

Figure 3.13: Comparison Between the Experimental and Analytical Results for Triaxial Compression (Non Proportional)
\( \sigma_1 \), i.e., along the compression meridian to failure, while \( \sigma_2 = \sigma_3 \) were held at 3.9 ksi (fig. 3.13). This non-proportional loading was simulated by hydrostatic preloading, followed by the increments of uniaxial compression \( \sigma_1 \) to failure, which causes changes in both hydrostatic and deviatoric stress components, and results in coupling as a major effect induced by the high hydrostatic stresses.

Fig. 3.13 shows the comparison between the analytical and experimental results. The analytical prediction of the initial hydrostatic response was slightly stiffer than the actual response. Beyond \( \sigma_1 = 3.9 \) ksi the correlation was only fair up to a stress level of 60% of the ultimate. But after this level till the ultimate, the correlation was good. The same behavior was observed in other principal directions. In each minor principal direction (directions 2 and 3) after pre-hydrostatic loading of 3.9 ksi, a further increase in the strain was predicted by the model, but this did not exceed the largest experimentally observed value (corresponding to \( \sigma_1 = 3.9 \) ksi). This may be attributed to the coupling effect relating the hydrostatic stresses and the deviatoric stresses to the deviatoric strains and the hydrostatic strains, respectively. This effect was modeled by means of the two coupling moduli \( H \) and \( Y \). For this complex loading history the analytical prediction captured the essential characteristics of the experimentally observed response.
Chapter 4
Constitutive Modeling of Cracked Concrete

4.1 Introduction

When the capacity of concrete in tension-dominated regions exceeds the tensile strength, cracks form perpendicular to the maximum tensile stress (or strain) direction. Initiation of cracks results in physical discontinuities in the RC structural elements and brings reinforcement into action. These physical discontinuities, associated with the stress and strain discontinuities, make the simulation of cracked concrete in finite element model a difficult task to achieve. Different crack models and post cracking stress strain relationships can in some cases influence not only the predicted ultimate capacities but also the failure mechanisms of reinforced concrete structures. Nonlinear behavior of concrete after cracking is a major factor contributing to the global nonlinear response of an RC structural element. Thus selection of a suitable model for simulating cracked concrete is very important.

This chapter presents a detailed description of the constitutive model adopted for cracked concrete, numerical examples, results of analysis and a discussion of the findings.

4.2 Criterion for Cracking

Concrete cracks under tensile stresses. Cracking of concrete is not a stable process and hence failure theories derived for stable materials may not be applicable to concrete cracking. The common criteria for fracture initiation are the strength
based criteria and the maximum strain based criterion. A dual criterion based on both stress and strain is recommended by Chen (1982). Bazant and Cedolin (1979) recommended a criterion based on the critical strain energy release rate or the stress intensity factor.

Though cracking of concrete has a major influence on the member, fortunately, precisely when cracking starts in most reinforced concrete members is not of paramount interest; the fact it does occur, however, is important (ASCE, 1982). In the present work strength based criterion was used for fracture initiation. A crack opens at a material point of the structure if the stress reaches the failure surface and the maximal principal stress (taking tension as positive) is greater than half of \( f'_c \), the strength in uniaxial tension.

### 4.3 Cracking Representations

Over the past 25 years a number of different models have been developed to represent cracking in the finite element analysis of reinforced concrete members. Two different approaches have been employed to model concrete cracking: the discrete and the smeared crack representations.

In the discrete crack system, a crack is represented as inter-element discontinuities along their common boundary as shown in fig. 4.1. In finite element modelling this is achieved by disconnecting or separating adjacent elements on each side of a node using additional nodes (Ngo and Scordelis 1967; Nilson 1986; Saouma and Ingraffea 1981). In this procedure the direction of crack propagation is governed by the finite element mesh. The use of the discrete crack model is restricted due to the difficulties involved in redefining the finite element topology and the lack of generality in possible cracking directions.

The alternate smeared crack model fits the finite element computational scheme ideally. In the smeared crack system, introduced by Rashid (1968), cracked concrete is assumed to remain a continuum. A crack is not discrete but implies an infinite number of parallel fissures across that part of the finite element (fig. 4.2).
In this method an orthotropic material matrix is chosen to represent the effect of cracks at that sampling point. This approach allows complete freedom for crack propagation direction besides requiring no remeshing operations.

The particular cracking model to be selected from the different models available depends upon the purpose of finite element study and the nature of the output desired from that study. The smeared cracking approach has been widely used and with apparent success for a variety of structures (Hand, Pecknold and Schnobrich 1972; Dodds, Darwin and Leibengood 1984; Rots, Nauta, Kusters and Blaauwendraad 1985; de Borst and Nauta 1985) and was adopted in the present work.
4.4 Smeared Crack Concept

4.4.1 Strain Decomposition

Figure 4.3 shows a typical uniaxial tensile stress strain curve of concrete. The point marked 'A' corresponds to the tensile strength of the concrete. Beyond this point, the tensile stress decreases with increase in strain. This phenomenon, termed as strain softening is attributed to the development of macro cracks in the material and their propagation.

In the finite element method of analysis of concrete structures using the 'smeared crack' concept, a procedure of resolution of strains has been adopted by several investigators (Bazant and Gambarova 1980). In this procedure, the total strain increment at a cracked sampling point is decomposed into two parts. The first part is the concrete strain and corresponds to the actual strain in concrete. The second part is the crack strain which is a fictitious quantity obtained by normalizing
the crack displacements with respect to a size parameter. This size parameter is normally the width of the sampling point under consideration and is termed the 'crack band width'. For a stress state on the descending branch AC of the stress-strain curve, such as that represented by point 'B' the total strain can be assumed to comprise of the concrete strain $DD'$ and the crack strain $OD'$. This resolution is depicted pictorially in fig 4.3(a), (b), (c).

The strain resolution can also be made in terms of incremental values as

$$\{\Delta \epsilon\} = \{\Delta \epsilon^c\} + \{\Delta \epsilon^r\}$$  \hspace{1cm} (4.1)

where

$\{\Delta \epsilon\}$ : total strain increment vector
$\{\Delta \epsilon^c\}$ : concrete strain increment vector and
$\{\Delta \epsilon^r\}$ : crack strain increment vector

The advantage with such a resolution is the possibility of a subresolution of concrete strains into elastic, plastic and creep strains. Thus each part of the strain can be studied in isolation for the development of a combined constitutive relationship. Also particular crack laws, which start from the notion of crack strain rather than the total strain, can be incorporated in a transparent manner. The importance of the decomposition has been recognized by a number of researchers (de Borst 1985; Rots 1985a; Riggs 1986). It is in essence an attempt to come closer to the discrete crack concept which completely separates the solid material from the crack by using separate finite elements.
Figure 4.3: Resolution of Total Strain of a Fracture Zone into Concrete Strain and Crack Strain
4.4.2 Development of Constitutive Model

The strain vector in equation 4.1 relate to the global coordinate axes and for a three dimensional configuration they have six components. The global crack strain vector reads,

\[
(\Delta \varepsilon_s^c) = (\Delta \varepsilon_{xx}^c \quad \Delta \varepsilon_{yy}^c \quad \Delta \varepsilon_{zz}^c \quad \Delta \gamma_{xy}^c \quad \Delta \gamma_{yx}^c \quad \Delta \gamma_{xz}^c \quad \Delta \gamma_{zx}^c)^T
\]

where \(x,y\) and \(z\) refer to the global coordinate axes and the superscript \(T\) denotes a transpose.

When incorporating crack traction-crack strain laws it is convenient to set up a local \(n,s,t\) coordinate system which is aligned with the crack as shown in fig 4.4. In the local system, defining a local crack strain vector \(\Delta \varepsilon_s^c\) as

\[
(\Delta \varepsilon_s^c) = (\Delta \varepsilon_{nn}^c \quad \Delta \gamma_{ns}^c \quad \Delta \gamma_{nt}^c)^T
\]

Figure 4.4: Local Coordinate System and Traction Across a Crack
where $\varepsilon_n^c$, is the normal crack strain and $\gamma_{ns}^c$, $\gamma_{nt}^c$ are shear crack strains in local n,s,t - coordinate system.

The relation between local and global crack strains is given by

$$ (\Delta \varepsilon^c) = (N)(\Delta \varepsilon^c) $$

with $N$ being a transformation matrix reflecting the orientation of the crack. A fundamental feature of the present concept is that $N$ is assumed to be fixed upon crack formation, so that the concept belongs to the class of fixed crack concepts. For a three dimensional configuration N reads

$$
(N) =
\begin{pmatrix}
    l_x^2 & l_x l_y & l_x l_z \\
    m_x^2 & m_x m_y & m_x m_z \\
    n_x^2 & n_x n_y & n_x n_z \\
    2l_x m_x & l_x m_y + l_y m_x & l_x m_z + l_z m_x \\
    2m_x n_x & m_x n_y + m_y n_x & m_x n_z + m_z n_x \\
    2n_x l_x & n_x l_y + n_y l_x & n_x l_z + n_z l_x
\end{pmatrix}
$$

where $l_x, m_x$ and $n_x$ form a vector which indicates the direction of the local n-axis expressed in the global coordinates. In accordance with this convention, the direction cosines with subscript y indicate the local s-axis and those with subscript z indicate the local t-axis.

In case of multiple cracks at the same sampling point, each individual crack strain increment is assembled into the global crack strain increment vector after applying the appropriate transformation given by,

$$ (\Delta \varepsilon^c) = \sum_{i=1}^{n} (N)(\Delta \varepsilon_i^c) $$

here $[N_i]$ is the transformation matrix corresponding to the ith crack,$\{\Delta \varepsilon_i^c\}$ is the crack strain increment vector for ith crack, in its local coordinates, and n is the number of cracks at the sampling point.

In the local coordinate system, defining a vector $\Delta t^c$ of incremental tractions across the crack as
\[(\Delta t^\tau) = (\Delta t^\tau_1 \Delta t^\tau_2 \Delta t^\tau_3)^T \tag{4.7}\]

in which \(\Delta t^\tau_1\) is the normal stress and \(\Delta t^\tau_2, \Delta t^\tau_3\) are shear stress increments.

The relation between the global stress increment \(\{\Delta \sigma\}\) and the local traction increment is given by

\[(\Delta t^\tau_i) = (N_i)^T (\Delta \sigma) \tag{4.8}\]

To complete the system of equations, a constitutive model is needed for the intact concrete and a traction-strain relation for the smeared cracks. For the concrete between the cracks a relationship is assumed of the following structure.

\[(\Delta \sigma) = (D^\omega) (\Delta e^\omega) \tag{4.9}\]

with the matrix \([D^\omega]\) containing the instantaneous moduli of the concrete.

The crack interface stress increments and the corresponding crack-strain increments are assumed to be related through the stiffness matrix \([D^\tau]\)

\[(\Delta t^\tau_i) = (D^\tau_i) (\Delta e^\tau_i) \tag{4.10}\]

By properly combining equations 4.1, 4.4, 4.8, 4.9 and 4.10 the overall stress-strain relation for the cracked concrete with respect to the global coordinate system can be developed.

From the equation 4.9 we get

\[(\Delta e^\omega) = (D^\omega)^{-1} (\Delta \sigma) \tag{4.11}\]

Inverting the equation 4.10 and substituting for \(\{\Delta t^\tau_i\}\) from equation 4.8 we get,

\[(\Delta e^\tau_i) = (D^\tau_i)^{-1} (N_i)^T (\Delta \sigma) \tag{4.12}\]

Multiplying both sides by \([N_i]\) and summing up over the number of cracks we get,

\[\sum_{i=1}^{n} (N_i) (\Delta e^\tau_i) = \left[\sum_{i=1}^{n} (N_i) (D^\tau_i)^{-1} (N_i)^T\right] (\Delta \sigma) \tag{4.13}\]
Substituting for \( \{ \Delta e^{co} \} \) and \( \{ \Delta t^{cr} \} \) from equations 4.11, 4.6 and 4.12 in 4.1 we get

\[
(\Delta \varepsilon) = [(D^{co})^{-1} + \sum_{i=1}^{n} (N_i)(D_i^{cr})^{-1}(N_i)^T](\Delta \sigma)
\]  

or

\[
(\Delta \sigma) = [(D^{co})^{-1} + \sum_{i=1}^{n} (N_i)(D_i^{cr})^{-1}(N_i)^T]^{-1}(\Delta \varepsilon)
\]

### 4.4.2.1 Crack Stiffness Parameters

The crack interface stress increments and the corresponding crack-strain increments are related through the crack interface matrix \([D^{cr}]\) which is given by

\[
(D^{cr}) = \begin{pmatrix}
D_c & 0 & 0 \\
0 & G_c & 0 \\
0 & 0 & G_c
\end{pmatrix}
\]

in which

- \(D_c\) is the crack normal stress-crack normal strain modulus and
- \(G_c\) is the crack shear-stress shear-strain modulus.

Figure 4.5 indicates these moduli. For an opening crack \(D_c\) has a negative value \(D^o_c\) indicating a decrease in stress with increase in strain. For arrested cracks, i.e. cracks that are closing, \(D_c\) has a positive value \(D^r_c\).
Figure 4.5: Crack Interface Stiffnesses

(a) crack normal stress-normal strain modulus

(b) crack shear stress-shear strain modulus
The crack shear-stress shear-strain modulus $G_c$ is always positive and indicates the roughness of the crack surface. Thus the effect of aggregate interlock can be included in this parameter. $G_c$ depends on the crack opening displacement or in terms of smeared crack model, the crack normal strain. Within smeared crack models that are not based upon the resolution of strain (Suidan and Schnobrich 1973; Hand, Pecknold and Schnobrich 1972; Lin and Scordelis 1975) it has become an accepted practice to represent the shear stiffness of cracked concrete by means of a shear retention factor $\beta$, indicating the percentage of elastic shear capacity that is retained after cracking. The effect of shear degradation is represented by reducing the initial shear capacity $G$ to $\beta G$ once the material has cracked. The crack shear stiffness $G_c$ can be expressed in terms of $\beta$ and $G$ (Rots et al. 1985)

$$G_c = \frac{\beta}{1 - \beta} G$$

(4.17)

The shear retention factor $\beta$ can assume the values between 0.5 and 0.0, representing the effective shear stiffness of uncracked concrete and fully cracked concrete respectively. In finite element analysis, high value of $\beta$ (close to 0.5) results in extensive multiple cracking and very low value of $\beta$ (close to zero) results in numerical difficulties in solving the system of equations. Thus, in the present study, $\beta$ is varied between 0.2 and 0.001 for different numerical experiments.

The tensile strain-softening diagram (fig. 4.6) is characterized by

1. the tensile strength $f_t$ at which cracking starts,

2. the area under the strain softening curve, $g_f$ and

3. the shape of the strain softening curve

(1) The strength limit $f_t$ may be assumed to be a constant or it may be varied under the effect of lateral stress fields. In this study the maximum principle stress of the stress state that violates the criteria for cracking explained in section 4.2 was taken as the strength limit $f_t$. This implies that the maximum value of $f_t$ is $f^U_t$, the strength in uniaxial tension and the minimum value of $f_t$ is $\frac{f^U_t}{2}$.

(2) The area $g_f$ under the strain softening curve can be expressed as
Figure 4.6: Strain Softening Curve and Fracture Energy

Figure 4.7: Idealized Strain Softening Diagram
\[ g_f = \int \sigma_{nn} \cdot \delta \epsilon_{nn} \]  \hspace{1cm} (4.18)

If \( \delta_n \) is the crack opening displacement and \( \epsilon_{nn}^{cr} \) is the smeared crack strain assumed to be uniformly smeared over width \( h \),

\[ \delta_n = h \cdot \epsilon_{nn}^{cr} \]  \hspace{1cm} (4.19)

\[ d\delta_n = h \cdot d\epsilon_{nn}^{cr} \]  \hspace{1cm} (4.20)

or

\[ d\epsilon_{nn}^{cr} = \frac{d\delta_{nn}}{h} \]  \hspace{1cm} (4.21)

In section 4.5 it is explained that the actual magnitude of \( h \) depends on the chosen element size, element shape and the integration scheme. Now the fracture energy \( G_f \), defined as the energy required to create a crack of unit area is given by

\[ G_f = \int \sigma_{nn} \cdot d\delta_{nn} \]  \hspace{1cm} (4.22)

From equations 4.18, 4.20 and 4.22 we get

\[ G_f = h \cdot g_f \]  \hspace{1cm} (4.23)

(3) The actual shape of the descending branch of the stress strain curve is idealized for incorporation into the model. A linear, (Petterson 1981), bilinear (Hillerborg 1984), quadratic or an exponential curve can be used. In the present work a linear and a bilinear model (Rots et al. 1985) were employed. These models are shown in figure 4.7.

From figure 4.7(a) the crack normal stiffness \( D_v^c \) for the linear model is given by

\[ D_v^c = -0.5 f_s^2 \cdot h \]  \hspace{1cm} (4.24)

\[ \epsilon_u = 2 \frac{G_f}{h \cdot f_s} \]  \hspace{1cm} (4.25)

and for the bilinear model (fig. 4.7(b)),
A common feature of models employing the smeared crack approach is their lack of objectivity with respect to size of the finite element mesh (Bazant and Cedolin 1979, 1983). The crack normal strain $\varepsilon_{nn}$, given in equation 4.3, acts only over a limited width, normally the width of the fracture zone or, in the case of finite element modelling, the width of the finite element over which the micro cracks are smeared out. As a consequence of this the crack strain should always be envisaged in close relation to this width, and this introduces a size effect into the strain-softening formulation.

The objectivity with respect to mesh refinement can be achieved by modifying the constitutive law and the fracture energy dependent on the mesh size by introducing a parameter called ‘crack band width’ (Bazant and Oh 1985). In fact, the strain softening modulus has to be adjusted to the chosen size of the finite elements, or otherwise the fracture energy release would be mesh dependent. The dependence of the strain softening modulus $D_c^o$ on crack band width ($h$), hereafter referred as characteristic length of the crack, reflects the adjustment of $D_c^o$ to the size of the finite elements.

Different methods have been employed in determining the magnitude of $h$. In regular meshes this parameter can be determined intuitively, but for irregular
meshes and cracks skewed to the element sides the determination of the parameter \( h \) is not direct. The upper limit for crack band width is a point to be considered in the analysis of large structures where limit may be exceeded and a mesh dependent strength limit is a possible alternative (Bazant and Cedolin 1979, 1983). Bazant (1984) and Bazant, Belytschko and Chang (1984) have stated that there is a lower limit for the crack band width, which roughly equals about three times the size of the aggregate. The lower limit considers the fact that the present formulation of the crack model is based on the local continuum theory.

Bazant and Oh (1983) proposed to include the orientation of the crack in evaluating the crack band width \( h \). Rots (1985) estimated the value of \( h \) based on the width of the fracture zone in the finite element mesh. In case of two dimensional applications \( h \) may be assumed to be the side of an equivalent square having the same area as that of the sampling point under consideration (Channakeshava 1987).

\[
h = \sqrt{DA}
\]

(4.29)

where \( DA \) is the area of the sampling point.

In three dimensional applications the characteristic length \( h \) becomes the side of an equivalent cube having the same volume as that of the sampling point under consideration (Cervera 1986).

Recently, Oliver (1989) proposed a general expression for the characteristic length which is derived from certain hypotheses on the behavior in the fracture localization zone. A brief explanation of the proposed consistent characteristic length, \( I^* \) (Oliver 1989) for two dimensional applications may be given as follows.

The characteristic length is the ratio between dissipated energy per unit surface area (fracture energy) and per unit volume (specific energy) within the band and is given by

\[
I^* = \frac{G_f}{g_f}
\]

(4.30)

By assuming an idealized behavior of a singular band for the cracked elements an expression for the characteristic length can be deduced (Oliver 1989) which depends on the mesh size, crack direction and spatial position.
The expression for characteristic length for 2-D problems, as given reference by Oliver (1989), reads as

\[ l^*(\xi_j, \eta_j) = \left[ \sum_{i=1}^{n_c} \left( \frac{\partial N_i(\xi_j, \eta_j)}{\partial x} \cos \theta_j + \frac{\partial N_i(\xi_j, \eta_j)}{\partial y} \sin \theta_j \right) \phi_i \right]^{-1} \]  

(4.31)

where

- \( N_i \) are the standard \( C^0 \) shape functions of an element of \( n_c \) corner nodes,
- \( \theta_j \) defines the direction of the crack normal,
- \( \phi_i \) takes the value of +1 if the corner \( i \) is ahead of the crack, and 0 otherwise as shown in figure 4.8, and
- \( \xi, \eta \) refer to the local isoparametric coordinate directions of the element.

Generalizing the equation 4.31 to three dimensional problems we get

\[ l^*(\xi_j, \eta_j, \zeta_j) = \left[ \sum_{i=1}^{n_c} \left( \frac{\partial N_i(\xi_j, \eta_j, \zeta_j)}{\partial x} l_1 + \frac{\partial N_i(\xi_j, \eta_j, \zeta_j)}{\partial y} m_1 + \frac{\partial N_i(\xi_j, \eta_j, \zeta_j)}{\partial z} n_1 \right) \phi_i \right]^{-1} \]  

(4.32)

in which \( l_1, m_1 \), and \( n_1 \) are the direction cosines of the crack normal with respect to the global coordinate axes \( x, y, \) and \( z \) respectively.

In computing the characteristic length using eqn. 4.32, the determination of \( \phi_i \) values was found to be numerically difficult in certain three dimensional problems. One such case was while analyzing a plain concrete beam subjected to pure torsion, where the torsional cracks occurred in a plane making exactly 45° with all three coordinate axes, and the finite element used to represent a portion of the beam was a cube. A two dimensionalized version of this is shown in fig. 4.9.
Finite element band modeling a singular band (Oliver 1989)

Figure 4.8: Computation of $\phi$ Values
When a crack occurs at the sampling point which is exactly at the center of the square (cube in 3-D) finite element with the crack normal at an angle 45°, the determination of \( \phi_i \) values at the two corner nodes (fig. 4.9), B and D is not so obvious as at the corner nodes A and C. Incorrect evaluation of \( \phi_i \) at corner nodes B and D resulted in the wrong estimation of the characteristic length from eqn. 4.32. Though this is a special case, it occurred in a torsion beam example (discussed later) and may result in incorrect evaluation of characteristic length in a larger problem. Also, as can be seen from the eqn. 4.32 the determination of the characteristic length at a sampling point requires the evaluation of the derivatives of the shape functions in the global directions at all the corner nodes of the finite element. In this study, the above mentioned observations resulted in finding an alternative interpretation of the characteristic length.

![Figure 4.9: Determination of Characteristic Length in a Square Element](image)

The consistent characteristic length \( l^* \) for two dimensional problem computed using eqn. 4.31, can be geometrically interpreted as the length 'PQ' (fig. 4.8). Alternatively the length PQ can be found by determining the length of the straight
line, defined by the crack normal direction, cut by the finite element boundaries. In a two dimensional problem this requires the straight line, defined by the crack normal, to be extended till it intersects the lines defining the finite element edges and finding the segment contained in the finite element under consideration. In case of three dimensional problems the extended crack normal is made to intersect with the planes defining the faces of the finite element under consideration.

In the present study the characteristic length was determined using the geometrical intersection of the extended crack normal with the finite element faces. In this method it was not necessary to evaluate the derivatives of the shape functions. Also no numerical difficulties were observed in implementing the concepts and determining the characteristic length in the special cases cited earlier.

4.6 Verification Studies

The smeared cracking model and characteristic length concepts described above were implemented in the finite element software 'INARCS'. In order to verify the concepts described in this chapter, two numerical examples were considered. The following gives the geometry description, finite element idealization and results of analysis for each of the numerical examples.

4.6.1 Constant Stress Field

A bar subjected to an ideal constant stress path was considered to investigate the ability of the model to simulate the global response involving softening due to localized fracture. Figure 4.10(a) describes the geometry of the bar considered. Constant stress field could be induced in the bar by applying consistent nodal loads in the axial direction at the free end or by applying constant displacement in axial direction at the free end. In this example, since global softening was expected because of localized fracture, the loading of the bar was by application of displacement at the free end. Two different meshes are selected to evaluate the effect of characteristic length on the global response of the axial bar under constant stress.
field. Fig. 4.10(b) shows the regular and irregular finite element meshes used to represent the axial bar.

For a constant stress field, the solution obtained using \( C^0 \) elements is exact and independent of the mesh size. However, this constant stress field does not produce localization of fracture, which has to be included artificially. In the meshes shown in figure 4.10(b), the elements in which fracture localization was induced by means of a small reduction (5%) of the tensile strength \( f'_t \) are shaded. The elements in the fracture zone of the regular mesh were distorted to obtain the irregular finite element mesh.

Fig. 4.10(c) shows the material properties and stress-strain law adopted for cracked sampling points. The numerical analysis was carried out under displacement control in 30 increments. Size of the displacement increment was determined in such a way that at the stage of crack initiation in the specimen only the low strength elements crack in a given increment. The number of displacement steps were determined such that the local softening in the cracked elements was complete i.e. globally the stiffness was almost zero. To plot the force-displacement response of the specimen, the force was determined as the sum of the resulting reactions at the fixed end of the specimen in axial direction.

The force-displacement response of the bar as obtained from the finite element analysis is shown in figure 4.11. For both the meshes, regular and irregular, three different integration schemes viz. 2x2x2, 3x3x3 and Irons 15 point rule (Irons 1971), were used in the numerical analysis. Observed results for both the meshes were identical for all the three integration schemes and are shown in fig. 4.11. This demonstrated the ability of the smeared cracking model in localizing the fracture and modeling the subsequent global softening of the axial bar.

4.6.2 Plain Concrete Notched Beam

From among the great variety of fracture experiments a notched beam of unreinforced concrete, tested by Malver (1990), was selected for analysis. The main reasons for choosing this experiment were: (a) the test was repeated several times
(a) Description of geometry

(b) Different finite element meshes

(c) Material description

$E = 10,000 \text{ N/mm}^2$

$f_t = 1.0 \text{ N/mm}^2$

$G_f = 0.125 \text{ N/mm}$

$\nu = 0.0$

Figure 4.10: Axial Bar in Constant Stress Field
(12 times); and (b) the necessary material parameters, such as $G_f$, were carefully specified. This allowed the numerical results to be matched fairly well with the experimental observations. The notched beam was simply supported at ends and loaded at the center under displacement control.

In order to assess the objectivity, in the results of analysis, with respect to element sizes two different finite element meshes were considered for the analysis. A coarse mesh with two elements across the notch and a fine mesh with four elements across the notch were selected. Figure 4.12 shows the geometry and the two finite element meshes used for the notched unreinforced concrete beam considered. The material properties for the cracked and uncracked concrete are shown in fig 4.13.

Number of numerical experiments were carried out to determine the right combination of the parameters, such as type of mesh, shear retention factor, idealized strain softening law (linear or bilinear). As in the physical experiment, the specimen was loaded at the center under displacement control. Total displacement of 0.7 mm was applied in 50 increments.
Figure 4.12: Unreinforced Concrete Notched Beam
Figure 4.13: Notched Beam Material Properties

In the coarse mesh the bottom element was completely cracked and the observed peak load occurred at a stage where cracking propagated into the top element. Analytically obtained crack pattern in the two elements of the notch is shown in fig. 4.14. Vertical cracks were seen at the sampling points at the center of the element. This is expected due to the symmetry in the notched beam. Symmetric inclined cracks were observed at the sampling points away from the center line. The inclination of the cracks was due to the presence of shear at the points away from the center line. As could be seen from the crack pattern, there was absolute symmetry in the cracks in both the elements across the notch. The ability of the 20 noded quadratic element to simulate a strain gradient was demonstrated clearly by the top element in the notch region which was subjected to both compression and tension.

Figure 4.15 shows the deflected shape of the notched beam under loading. The load-deformation response obtained from the analysis of notched beam using coarse mesh (42 elements) is shown in figure 4.16. Two values of $\beta$ were considered with the idealized linear softening model. The bilinear softening model was considered with a low value of $\beta$ (0.001).

\[ E = 21,700 \text{ N/mm}^2 \]
\[ f_c' = 29.0 \text{ N/mm}^2 \]
\[ f_t' = 3.1 \text{ N/mm}^2 \]
\[ G_f = 0.0763 \text{ N/mm} \]
\[ v = 0.20 \]
Figure 4.14: Crack Pattern in Notched Beam

Figure 4.15: Deformed Shape of the Notched Beam
In the analysis with linear softening model, both values of $\beta$ resulted in overestimation of the peak load by 11.5%, however, the lower value of $\beta$ resulted in a better post peak response. The use of idealized bilinear model for crack softening resulted in a response that matched well with the experimental values. Both the peak load and the post peak response were simulated well. The kinks in the post peak response of the analytical results were due to sudden release of energy when cracks were initiated at many sampling points lying at the same horizontal level. Thus the results of analysis for coarse mesh indicated that a low value of $\beta$ with bilinear model would be a better combination for the response that matches well with the experimental values.

The results of analysis for fine mesh are compared with that of coarse mesh in figure 4.17. Complete post peak response of the notched beam could not be simulated because of divergence in the iterative procedure. As could be observed from the figure, analysis using fine mesh resulted in overestimating the peak load by 5.4% and only a part of the post peak response could be simulated. The adopted incremental procedure was not capable of simulating a very sudden change in the stiffness such as the one indicated by the post peak response for fine mesh. The failure of the numerical procedure also indicated that too much refinement in finite element mesh may not always give better results, particularly when mesh dependent models, such as the current smeared crack model, are used. Also the refinement used in the FE mesh was not appropriate if the lower limit on the crack band width is considered. In the fine mesh the depth of the element across the notch was 12.75 mm which was only half the size of the aggregate used in concrete.

Figure 4.16 shows the analytically observed load-deformation response of the notched beam analyzed using the coarse mesh (42 elements). With the idealized linear softening model two values of $\beta$ are considered. As can be seen from the observed load-deformation response (fig. 4.16), lower value of $\beta(=0.001)$ resulted in better post peak response. The bilinear softening model when considered with low value of $\beta$ resulted in much better post peak response. The results of analysis using a coarse mesh indicated that the low value of $\beta$ with bilinear model is a better combination for predicting a response that matches with the experimentally observed one.
Figure 4.16: Load-Deformation Response for Coarse mesh

In the analysis of the notched beam using fine mesh low value of $\beta$ and bilinear softening model are used and the resulting load-deformation response is shown in fig. 4.17.
Figure 4.17: Load-Deformation Response Comparison
Chapter 5
Constitutive Modeling of Reinforcing Steel and Bond Slip

5.1 Introduction

In contrast to concrete, the material behavior of reinforcing steel is well known. Techniques for modeling the reinforcement rarely receive as much emphasis as do techniques for modeling the concrete component of reinforced concrete. Some investigators apparently consider their representation of reinforcing steel so unimportant as to provide only a limited details in their description of the finite element model (Balakrishnan and Murray 1988, Chang et al. 1987, Gajer and Dux 1990, Wu et al. 1991). This may be due to the fact that there are few differences in the properties of reinforcing steels, which are typically modeled as uniaxially loaded, elastic-perfectly plastic or elastic strain hardening materials.

This chapter presents the constitutive relationship adopted for the steel, detailed description of the finite element modeling of steel and its numerical implementation, and a numerical example verifying the applicability of the concepts presented herein.

5.2 Constitutive Relationship for Steel

Typical stress-strain curves for steel reinforcing bars loaded monotonically in tension are shown in fig 5.1. For low Carbon reinforcing steels the curves exhibit a linear elastic region followed by a yield plateau and then a long strain hardening...
region. The length of the yield plateau and strain hardening regions typically decrease as the strength of the steel increases.

For monotonic loading, reinforcing steel is typically represented as either an elastic perfectly plastic material (fig. 5.2(a)) or as an elastic strain hardening material (fig. 5.2(b)). Occasionally, it is represented using a tri-linear stress strain curve (fig. 5.2(c)) or with a representation of complete stress strain curve (fig. 5.2(d)). Of the four methods, the elastic-perfectly plastic representation was the most often selected, followed by elastic strain hardening representation. Where the numerical stability of the solution is of concern, the latter is the representation of choice because of the positive slope of the stress strain curve, once the steel becomes nonlinear. In the present study the idealized elastic-strain hardening representation (fig. 5.2(b)) of steel was adopted with the strain hardening modulus $E_{sp}$ taken as 2% of the elastic modulus $E_t$. Unloading and reloading of steel stress in the plastic range is allowed along a path parallel to the direction of initial modulus of elasticity.

### 5.3 Reinforcement Representations

In representing the reinforcement in the finite element model of a reinforced concrete member three alternate approaches can be used: (1) smeared or distributed; (2) discrete; and (3) embedded. In the smeared representation (fig. 5.3(a)), which is suitable for modeling surface-type reinforced concrete structures, the steel is assumed to be uniformly distributed over the concrete element, with a particular orientation angle $\theta$. The smeared steel element is obtained by employing a composite constitutive matrix for concrete. If $\rho_s$ is the steel ratio, $[D_s]$ is the constitutive matrix of steel and $[D_c]$ is the constitutive matrix for concrete, the composite material matrix $[D_{comp}]$ is obtained as

$$[D_{comp}] = [D_c] + \rho_s[D_s]$$  \hspace{1cm} (5.1)

As can be inferred from this equation, the smeared representation of steel assumes full compatibility between concrete and steel and hence bond slip can not be considered.
Figure 5.1: Stress-Strain Curves for Steel Reinforcing Bars (Nilson and Winter 1968)
Figure 5.2: Idealizations for Stress Strain Curves for Reinforcing Steel in Tension Compression
When the rebars are sparsely located in the RC member, either the discrete or embedded representation become more effective. In the discrete model (fig. 5.3(b)) space truss elements are used to represent the reinforcing bars. Although this model is conceptually simple, its use is constrained by the facts that the concrete finite element mesh patterns are restricted by the location of the rebars and numerically this is less efficient than the other two approaches. Discrete model allows only a few representative rebars to be included in the finite element model. Inclusion of all bars, especially in 3D finite element models, leads to a large number of concrete elements whose size and shape are controlled by the geometry of the bars rather than by the need to model the stress gradient in the structure. Also the elements used to represent the cover concrete veer towards high aspect ratios. In most cases of discrete representation the secondary and/or shear reinforcement is ignored. A significant advantage of discrete representation, in addition to its simplicity, is that it can account for possible displacement of the reinforcement with respect to the surrounding concrete (bond slip).

In the embedded model, (fig. 5.3(c)), the reinforcing bar is represented as an axial bar embedded in the concrete finite element and the stiffness of the bar is evaluated in conjunction with the isoparametric shape functions of concrete finite element (Phillips and Zeinkiewicz 1976). This model allows the rebars that are located and oriented arbitrarily, to be represented within the concrete finite element at their exact locations.

5.4 Choice of Reinforcement Representation

Conventional finite element analysis of reinforced concrete structures frequently make an assumption that severely restricts the accuracy in simulating the reinforcement. The rebars are assumed to lie along the edges of concrete finite elements. This assumption poses a difficult constraint which often leads to only a few representative bars being included in the finite element model.

Although, in recent years, finite element analysis of RC structures has developed to a relatively high degree of sophistication (ASCE 1982,1985) in view of
advances in constitutive modeling and numerical techniques, in most examples of analysis meshes with a high degree of regularity are favored. This is because of the mesh dependent nature of finite element analysis procedures such as smeared representation of cracking in concrete. Countering the choice of regular meshes are constraints imposed by the modeling of reinforcement.

In order to model complicated reinforcement details, while retaining the advantages of a regular mesh, an embedded representation of reinforcement appears to be a preferred approach and the same was used in the present work. Especially in three dimensional applications where the rebars are oriented in different directions, the smeared representation of steel cannot be used effectively. The use of discrete model results in a large number of finite elements and the application of embedded representation serves the objective rather than just being advantageous.
Figure 5.3: Alternate Representations of Steel (ASCE 1982)
5.5 Bond between Concrete and Reinforcing Steel

Bond between concrete and steel reinforcement is of fundamental importance to most aspects of localized reinforced concrete behavior especially when cracks have taken place. The bond depends on many factors associated with properties of concrete and steel and also with the geometry, the loading and boundary conditions. The characteristics of the interface can affect the location, spacing and width of cracks in members, the internal distribution of stresses in both concrete and steel, and the effective stiffness of the members (ASCE 1982).

When a reinforced concrete member is subjected to loading, the steel reinforcing bar has a tendency to slip through the surrounding concrete. This slip is resisted by a combination of adhesion, friction and the mechanical interlock of the protruding bar ribs with the surrounding concrete. In a finite element analysis this effect is introduced by using either linkage elements or special interface elements (Nilson 1968; ASCE 1982). The stiffness characteristics of these elements are derived from experimental bond stress-slip curves. However in many analyses, bond slip and bond degradation are only of secondary importance and may not affect overall structural behavior significantly, especially for monotonic loading cases (Gerstle 1981).

In the present study the bond-slip is simulated by assigning a slip degrees of freedom associated with the rebar nodes on interelement boundaries. The nonlinear characteristic associated with the bond slip stiffness is derived from the relationship given by Nilson (1968) and is given as

\[
\tau_b = 980.93d - 57361.0d^2 + 837383.0d^3
\]  

(5.2)

and the corresponding stiffness is

\[
E_b = A_b \frac{d\tau_b}{dd} = A_b [980.93 - 1.147 \times 10^5 d + 2.52 \times 10^4 d^2]
\]  

(5.3)

where \(\tau_b\) is the local bond stress in MPa,
\(d\) is the slip in mm,
\(A_b\) is the contributing area in \(mm^2\) and
\(E_b\) is the slip modulus.
5.6 Finite Element Modeling of Embedded Reinforcement and Bond Slip

Over the past decade, a number of embedded representations for reinforcement were published. Phillips and Zeinkiewicz (1975) and Elwi and Murray (1986) separately developed embedded representation in which the virtual work integration is performed along the reinforcing bar and the reinforcement is aligned with one of the local isoparametric element coordinate axes. Embedded representation by Chang et al. (1987) allows for a rebar placed at an angle to the local isoparametric element axes but restricted to problems having rectilinear meshes.

When using the embedded formulation, for simulating the complete nonlinear response of RC structural elements such as beams, the current practice is to reduce the groups of bars at a given level to one, and restrict them to lie along the local coordinate lines of the parent concrete element (Balakrishnan and Murray 1986; Cervera 1986). In RC structural elements, such as a column, with diagonal hoop reinforcement, the ties cannot be forced to lie along the local coordinate axes, and hence are ignored in the finite element analysis (Abdel-Halim and Abu-Lebdeh, 1989). This precludes the examination of different finite element meshes for concrete without changing the rebar geometry definition.

A formulation presented by Elwi and Hrudey (1989) for a general reinforcing bar embedded in a higher-order two dimensional finite element assumes that the global coordinates of the points on the reinforcing bar at which the bar intersects the element edges are available. In three dimensional applications, the determination of the points of intersection of a rebar with solid element faces in global coordinates is a highly involved task which makes the use of embedded approach in 3D a difficult task. In order to overcome this difficulty, in the present study, a mesh mapping procedure that gives the points of intersection of rebars with the solid element faces in global coordinates was developed and is presented in detail in the next chapter. In the present study the finite element model for embedded reinforcement proposed by Elwi and Hrudey (1989) for 2D applications was extended to 3D cases.
5.6.1 Geometric Formulation

The parent element, shown in fig. 5.4, is described in global coordinates \((x,y,z)\). The normalized coordinates of the element are \((\xi, \eta, \zeta)\). In the finite element isoparametric formulation (Zienkiewicz 1977) the global coordinates of any point in the element are expressed in terms of a vector of interpolation functions \(\{\phi\}\) and the nodal coordinates \(\{z\}\), \(\{y\}\) and \(\{x\}\) as

\[
\begin{bmatrix}
  x \\
  y \\
  z
\end{bmatrix} =
\begin{bmatrix}
  \phi(\xi, \eta, \zeta) & 0 & 0 \\
  0 & \phi(\xi, \eta, \zeta) & 0 \\
  0 & 0 & \phi(\xi, \eta, \zeta)
\end{bmatrix}
\begin{bmatrix}
  \{z\} \\
  \{y\} \\
  \{x\}
\end{bmatrix}
\]  

(5.4)

Noting that

\[
\phi(\xi, \eta, \zeta) = \{\phi\}^T = ((\phi_1(\xi, \eta, \zeta), \phi_2(\xi, \eta, \zeta), \ldots, \phi_n(\xi, \eta, \zeta)))
\]  

(5.5)

and

\[
\{\mathbf{z}\} = \{\mathbf{z}\}^T = ([z_1, z_2, \ldots, z_n])
\]  

(5.6)

where \(n\) is the number of nodes in the parent finite element and in the present work corresponding to a 20 noded element \(n = 20\).

The corresponding local to global transformation is given by

\[
\begin{bmatrix}
  dx \\
  dy \\
  dz
\end{bmatrix} = [J]
\begin{bmatrix}
  d\xi \\
  d\eta \\
  d\zeta
\end{bmatrix}
\]  

(5.7)

where the Jacobian \([J]_{3x3}\) is given by
Figure 5.4: Embedded Representation of Reinforcement
The parent element (concrete finite element) mesh is established without giving particular consideration to the location and geometry of the reinforcing bars. Once the global mesh has been created, the reinforcing bars are specified by locating a set of rebar nodes. Automatic generation of these rebar nodes for a given rebar grid is discussed in detail in the next chapter. To represent the continuity in the rebar, rebar nodes such as $P_B$ and $P_E$ as shown in fig. 5.4, are defined at points where the reinforcement crosses the parent element boundaries. To represent the higher order variation in the strain along the rebar segment, additional nodes can be defined along the segment between the two rebar nodes $P_B$ and $P_E$. In the present study, since the quadratic 3D finite elements are employed to represent concrete, an additional node $P_C$ (fig. 5.4) is defined at the center of the segment in order to achieve the strain compatibility between the parent finite element and the rebar segment.

The coordinates of points, such as $P_C$ on the reinforcing bar between the end nodes $P_B$ and $P_E$ (hereafter referred as rebar segment) are obtained by interpolation. Letting $\{x^*, y^*, z^*\}$ be vectors containing the global coordinates of all the rebar nodes associated with a single parent element, the coordinates of any other point, $s$, on the rebar are then given by

$$
\begin{align*}
\begin{bmatrix}
  x \\
  y \\
  z
\end{bmatrix} &=
\begin{bmatrix}
  \langle \psi \rangle & 0 & 0 \\
  0 & \langle \psi \rangle & 0 \\
  0 & 0 & \langle \psi \rangle
\end{bmatrix}
\begin{bmatrix}
  x^* \\
  y^* \\
  z^*
\end{bmatrix}
\end{align*}
$$

(5.9)

The one dimensional interpolation functions $\langle \psi \rangle = (\psi_1, \psi_2, \ldots, \psi_m)$ are expressed in terms of an independent normalized coordinate $\gamma$, (fig. 5.4), $-1 \leq \gamma \leq$
1. The number of nodes, $m$, on the rebar segment within the parent element is taken as 3 in order to represent the quadratic variation in the displacement field and the corresponding interpolation functions are taken as

$$
\psi_1 = \frac{1}{2}(\gamma^2 - \gamma) \\
\psi_2 = (1 - \gamma^2) \\
\psi_3 = \frac{1}{2}(\gamma^2 + \gamma)
$$

Various stiffness terms associated with the reinforcing bar and bond slip require that integration be performed along the rebar. Denoting a differential element of length $ds$, along the rebar, the direction cosines of the tangent at any point on the rebar (fig. 5.5) are given by

![Figure 5.5: Global and Local Coordinates Along a Rebar](image)

$$
l = \frac{dx}{ds} \\
m = \frac{dy}{ds}
$$
\[ n = \frac{dz}{ds} \]  

(5.15)

Since \(l^2 + m^2 + n^2 = 1\), equation 5.11 yields

\[
\left(\frac{dx}{ds}\right)^2 + \left(\frac{dy}{ds}\right)^2 + \left(\frac{dz}{ds}\right)^2 = 1
\]

(5.16)

Multiplying the above equation both sides by \((\frac{ds}{d\gamma})^2\); eqn. 5.16 becomes

\[
\frac{ds}{d\gamma} = J^* = \sqrt{\left(\frac{dx}{d\gamma}\right)^2 + \left(\frac{dy}{d\gamma}\right)^2 + \left(\frac{dz}{d\gamma}\right)^2}
\]

(5.17)

Where \(J^*\), the Jacobian, is the mapping factor of the transformation from the global to local coordinate system of the rebar. The terms of the Jacobian can be evaluated from equation 5.9 as

\[
\frac{dx}{d\gamma} = (d\psi/d\gamma)\{x^*\}
\]

(5.18)

\[
\frac{dy}{d\gamma} = (d\psi/d\gamma)\{y^*\}
\]

(5.19)

\[
\frac{dz}{d\gamma} = (d\psi/d\gamma)\{z^*\}
\]

(5.20)

and from equation 5.11

\[
(d\psi/d\gamma) = (\gamma - \frac{1}{2}, -2\gamma, \gamma + \frac{1}{2})
\]

(5.21)

The Jacobian \(J^*\) can now be calculated at any point along the rebar. The direction cosines of the tangent are therefore written as

\[
l = \frac{dx}{d\gamma} \cdot \frac{d\gamma}{ds} = \frac{1}{J^*} \cdot \frac{dx}{d\gamma}
\]

(5.22)

\[
m = \frac{dy}{d\gamma} \cdot \frac{d\gamma}{ds} = \frac{1}{J^*} \cdot \frac{dy}{d\gamma}
\]

(5.23)

\[
n = \frac{dz}{d\gamma} \cdot \frac{d\gamma}{ds} = \frac{1}{J^*} \cdot \frac{dz}{d\gamma}
\]

(5.24)
A differential element of volume $dV_s$ and a differential element of surface area $dS_s$ at the reinforcing bar can be expressed in terms of length $ds$; the cross-sectional area of the rebar, $A$; and the perimeter of the rebar, $O_s$, as follows:

$$dV_s = A_s ds = A_s \left( \frac{ds}{d\gamma} \right) d\gamma$$ (5.25)

$$dS_s = O_s ds = O_s \left( \frac{ds}{d\gamma} \right) d\gamma$$ (5.26)

5.6.2 Evaluation of Strain along the Rebar

A general expression for the strain (Elwi and Hrudey 1989) at any sampling point along the rebar in incremental form can be written as

$$\Delta \varepsilon_s = \frac{d\Delta w_b}{ds} + \Delta \varepsilon_x l_x^2 + \Delta \varepsilon_y m_y^2 + \Delta \varepsilon_z n_z^2 +$$
$$\Delta \gamma_{xy} l_m + \Delta \gamma_{yz} m_n + \Delta \gamma_{xz} l_n$$ (5.27)

The strain increments $\Delta \varepsilon_x, \Delta \varepsilon_y, \Delta \varepsilon_z, \Delta \gamma_{xy}, \Delta \gamma_{yz},$ and $\Delta \gamma_{xz}$ are obtained directly from the parent displacement field. The first term in equation 5.27 corresponds to the contribution of bond slip. The slip increment along the rebar is interpolated as

$$\Delta w_b(\gamma) = \{\psi(\gamma)\}\{\Delta w_b^*\}$$ (5.28)

in which $\delta w_b^*$ indicates the nodal values of the slip associated with the rebar. The incremental strain due to slip is

$$\frac{d\Delta w_b}{ds} = \frac{d\{\psi(\gamma)\}}{ds} \cdot \{\Delta w_b^*\}$$
$$= \frac{d\gamma}{ds} \cdot \frac{d\{\psi(\gamma)\}}{d\gamma} \{\Delta w_b^*\}$$
$$= \frac{d\gamma}{ds} \cdot \{\psi, \gamma\}\{\Delta w_b^*\}$$ (5.29)
Rewriting the equation 5.27 in matrix form we get

\[
\Delta \epsilon = \langle B_b \rangle \{ \Delta w_b \} + \left[ l^2 \ m^2 \ n^2 \ lm \ mn \ nl \right] \{ \Delta u \} \quad (5.30)
\]

The incremental strain vector for the parent finite element can be expressed in terms of the incremental displacement \( \{ \Delta u \} \) and \( [B] \) the strain-displacement matrix (Zienkiewicz 1977) as

\[
\{ \Delta \epsilon \} = [B] \{ \Delta u \} \quad (5.31)
\]

therefore equation 5.30 becomes

\[
\{ \Delta \epsilon \} = \langle \langle B_b \rangle \langle B_s \rangle \rangle \left\{ \begin{array}{c}
\{ \Delta w_b \} \\
\{ \Delta u \}
\end{array} \right\} \quad (5.32)
\]

where

\[
\langle B_s \rangle = \left[ l^2 \ m^2 \ n^2 \ lm \ mn \ nl \right] [B] \quad (5.33)
\]
5.6.3 Virtual Work Formulation

In formulating the expressions for virtual work, it is assumed that in the reinforcement the strain occurs only in the direction along the rebar. Using the incremental form of the principle of virtual work, the incremental internal virtual work may be written as

\[ \delta \Delta W = \int_{V} (\sigma_{e} + \Delta \sigma_{e}) \delta \Delta \varepsilon_{e} dV + \int_{S} (\sigma_{b} + \Delta \sigma_{b}) \delta \Delta \omega_{b} dS \]  \hspace{1cm} (5.34)

in which \( \sigma_{e} \) and \( \sigma_{b} \) represent the axial stress in the rebar and the bond stress respectively. \( V \) and \( S \) represent the volume and the surface area of the rebar.

In incremental form the constitutive equation for the rebar may be written as (Balakrishnan and Murray, 1986)

\[ \Delta \sigma_{e} = E_{e} \Delta \varepsilon_{e} \]  \hspace{1cm} (5.35)

and

\[ \Delta \sigma_{b} = E_{b} \Delta \omega_{b} \]  \hspace{1cm} (5.36)

in which \( E_{e} \) and \( E_{b} \) are the tangential moduli for the normal stress-strain relation and the bond stress-slip relationship respectively. Substituting for various terms from equations 5.25, 5.26, 5.35 and 5.36, eqn. 5.34 can be rewritten as

\[ \delta \Delta W = \int_{S} (\delta \Delta \varepsilon_{e} E_{e} \Delta \varepsilon_{e} A_{e} + \delta \Delta \omega_{b} E_{b} \Delta \omega_{b}) ds \]
\[ + \int_{S} (\delta \Delta \varepsilon_{e} \sigma_{b} A_{b} + \delta \omega_{b} \sigma_{b} O_{b}) ds \]  \hspace{1cm} (5.37)

substituting for \( \Delta \varepsilon_{e}, \Delta \omega_{b}, \delta \Delta \varepsilon_{e} \) and \( \delta \Delta \omega_{b} \) from equations 5.28 and 5.32, the incremental internal virtual work may be written as
\[
\delta \Delta W = \langle \delta w_5^* \rangle \langle \delta \Delta u \rangle \begin{pmatrix}
[K_{bb}] & [K_{bs}] \\
[K_{sb}] & [K_{ss}]
\end{pmatrix}
\begin{pmatrix}
\{\Delta w_5^*\} \\
\{\Delta u\}
\end{pmatrix} + \begin{pmatrix}
\{F_b\} \\
\{F_s\}
\end{pmatrix}
\]

(5.38)

where the b-partitions are associated with slip degrees of freedom, and the s-partitions are associated with the degrees of freedom of the parent element and

\[
[K_{bb}] = \int_\gamma \{\psi\} E_b(\psi) O_s + \{B_b\} E_s(B_b) A_s \frac{ds}{d\gamma} \cdot d\gamma
\]

(5.39)

\[
[K_{bs}] = [K_{sb}]^T = \int_\gamma \{B_b\} E_s(B_s) \frac{ds}{d\gamma} \cdot A_s d\gamma
\]

(5.40)

\[
[K_{ss}] = \int_\gamma \{B_s\} E_s(B_s) \frac{ds}{d\gamma} \cdot A_s d\gamma
\]

(5.41)

\[
\{Q_b\} = \int_\gamma \{B_b\} \sigma_s A_s + \{\psi\} \sigma_b O_s \frac{ds}{d\gamma} \cdot d\gamma
\]

(5.42)

\[
\{Q_s\} = \int_\gamma \{B_s\} \sigma_s A_s \frac{\Delta \sigma}{d\gamma}
\]

(5.43)

In the above equations, the submatrices $[K_{bb}]$, $[K_{bs}]$ and $[K_{ss}]$ represent the stiffness contributions and the vectors $\{Q_b\}$ and $\{Q_s\}$ are internal forces associated with the stresses $\sigma_b$ and $\sigma_s$.

In forming the global stiffness matrix, the slip degrees of freedom $\{\Delta w_5^*\}$ may be treated as global degrees of freedom such that those degrees of freedom associated with nodes on interelement boundaries are shared by more than one element. Using this approach the compatibility of the slip between adjacent elements is maintained. However in practical applications, the introduction of additional global degrees of freedom results in substantial increase in the size of the stiffness matrix. Thus activation of the slip degrees of freedom should be done based on the need to model the effect of the bond slip in a reinforced concrete structural element.
5.7 Determination of Normalized Coordinates of a Point on the Rebar

In order to evaluate the integral for the incremental internal virtual work in the rebar segment, it is necessary to determine the strain in the present element at sampling points on the rebar segment. Also the various stiffness terms associated with the reinforcing bar and bond slip require that integration be performed along the rebar segment in the parent element. Thus for a point on the rebar segment with global coordinates \((x, y, z)\), it is necessary to determine the associated normalized coordinates \((\xi, \eta, \zeta)\) in the parent element.

The mapping between the local (normalized) and global coordinates for isoparametric elements (fig. 5.4) is given by equation 5.4. The inverse relationship of this equation is not directly available in an explicit form. Thus the inverse transformation must be done numerically in an indirect manner. For this purpose, two methods viz., integration method and iterative method are available in literature (Elwi and Hrudey 1989). In the present work the iterative method was used because of its simplicity.

The coordinates of any sampling point, \(s\), on the rebar are given by equation 5.9. The corresponding local coordinates \((\xi, \eta, \zeta)\) are given by the roots of the vector function:

\[
f(\xi, \eta, \zeta) = \begin{bmatrix} x \\ y \\ z \end{bmatrix} - \begin{bmatrix} \phi & 0 & 0 \\ 0 & \phi & 0 \\ 0 & 0 & \phi \end{bmatrix} \begin{bmatrix} \{x\} \\ \{y\} \\ \{z\} \end{bmatrix}
\]

These roots can be evaluated using the Newton-Raphson iterative scheme from which the solution, after \(n+1\) iterations with an initial guess of \(\xi = \eta = \zeta = 0\) is given by
\[
\begin{align*}
\begin{bmatrix}
\xi \\
\eta \\
\zeta
\end{bmatrix}^{n+1} &= 
\begin{bmatrix}
\xi \\
\eta \\
\zeta
\end{bmatrix}^{n} + 
\begin{bmatrix}
\Delta \xi \\
\Delta \eta \\
\Delta \zeta
\end{bmatrix}^{n+1}
\end{align*}
\]  
(5.45)

where

\[
\begin{align*}
\begin{bmatrix}
\xi \\
\eta \\
\zeta
\end{bmatrix}^{n+1} &= [J^n]^T^{-1} \begin{bmatrix}
x \\
y \\
z
\end{bmatrix} - 
\begin{bmatrix}
\phi^n & 0 & 0 \\
0 & \phi^n & 0 \\
0 & 0 & \phi^n
\end{bmatrix} \begin{bmatrix}
x \\
y \\
z
\end{bmatrix}
\end{align*}
\]  
(5.46)

with

\[
[J^n] = [J(\xi^n, \eta^n, \zeta^n)]
\]  
(5.47)

and

\[
(\phi^n) = (\phi(\xi^n, \eta^n, \zeta^n))
\]  
(5.48)

The above procedure based on the iterative method was found to converge within a relatively small number of iterations and yield accurate values of the coordinates.

**5.8 Verification Analyses**

All the concepts, related to the embedded representation of the steel reinforcement and the slip, presented in this chapter were implemented in a finite
element program "INARCS". In order to test the validity of the embedded representation of steel, two numerical examples were considered. The following sections present a description of the geometry and discusses the finite element idealization, material properties and the results of analysis for each numerical example.

5.8.1 Constant Stress Field

A reinforced concrete axial bar subjected to an ideal constant stress path was considered to verify the embedded representation of the steel and slip between the concrete and steel. This example is same as the one considered in section 4.6.1, except that the axial bar is reinforced in the present case. The loading of the specimen was caused by applying by displacement at free end. Figure 5.6 describes the geometry of the bar, material properties and finite element mesh used. The boundary conditions at the fixed end were simulated by arresting the translational degrees of freedom in the axial direction for all nodes at the fixed end including the end nodes of rebars.

Figure 5.7 shows the results of analysis in the form of load displacement response of the RC axial bar. Response of the plain concrete specimen was also shown in the same figure for comparison. As can be seen from the predicted response, the analysis yielded a stiffer response and higher peak load for the reinforced specimen in comparison with the unreinforced one. This was due to the presence of steel which contributed to the stiffness and strength of the specimen. Two cases of analysis, with and without bond slip, were considered to determine the effect of bond slip on the global load-deformation response and stress distribution in steel. The results have indicated that inclusion of bond slip in the analysis did not affect the strength or stiffness to a noticeable extent. However, as expected, the stress distribution in the steel was affected by the bond slip.

The observed variation of the bond slip along the length of a reinforcing bar before and after cracking is shown in figure 5.8. There was no slip observed at the fixed end as suggested by the imposed boundary condition. At the free end of the rebar, where the concrete pulled out, maximum slip was observed indicating
the physical movement of the steel inside the concrete when it is loaded. After cracking, which was basically localized in the elements at the fixed end, there was a sudden variation of the bond slip in the fracture zone. This indicated the interaction between cracked concrete and steel. At the other points along the rebar the variation in the slip was identical to the response before cracking except for the increase in free end slip of the bar due to additional straining of the member.

Figure 5.9 shows the variation of stress in the rebar before and after cracking of concrete. Before cracking the stress in steel was constant in the region where the bond slip was zero. Also near the free end there was reduction in steel stress due to bond slip. At the free end, the reinforcing bar did not move along with the strained concrete because of the bond slip between the concrete and steel. This resulted in less strain and thus lower stress in steel at the free end. After cracking, the concrete elements in the fracture zone were unloading. This offered less resistance to the straining of the specimen resulting in force transfer from concrete to steel. The observed increase in steel stress at the fixed end indicated the load transfer from concrete to steel.

From the results of analysis it can be concluded that the variation of the stress in steel and the global load deformation response were simulated consistent with the physical response of the specimen.

5.8.2 Reinforced Concrete Beam

An experimentally tested reinforced concrete beam without shear reinforcement (Bresler and Scordelis 1963) was analyzed to further validate the applicability of the concepts presented in this chapter. Fig. 5.10 gives the description of the member, details of reinforcement and material properties for the specimen.

Taking advantage of the symmetry of the geometry and load position, only half of the beam was modeled in the finite element analysis. Finite element mesh used for simulating the deformational response is shown in fig. 5.11. Four elements of equal size were used across the depth of the beam to represent concrete. Along the length, 24 elements of equal size were used in the FE model. Boundary conditions
(a) Description of geometry

(b) Different finite element meshes

(c) Material description

Figure 5.6: Reinforced Axial Bar in Constant Stress Field
Figure 5.7: Load Displacement Response of Axial Bar

Figure 5.8: Bond Slip along the Rebar
Figure 5.9: Stress Distribution Along the Rebar
(a) Elevation

(b) Cross section

(c) Material properties

Figure 5.10: Reinforced Concrete Beam without Stirrups

$E = 20,000 \text{ N/mm}^2$

$f_c' = 22.5 \text{ N/mm}^2$

$f_t' = 2.25 \text{ N/mm}^2$

$G_f = 0.250 \text{ N/mm}$

$\nu = 0.20$

$f_y = 444.0 \text{ N/mm}$

$\epsilon_y = 0.00222$
along the line of symmetry were simulated by restraining the translational degrees of freedom in horizontal direction. Central point load was represented as a distributed load over the top face of the top element near the line of symmetry.

Figure 5.12(a) and (b) shows the observed crack pattern in the beam at early stages of cracking and near failure load respectively. Vertical cracks were seen at the center of the beam where flexure and shear are present. But towards the support the cracks were inclined due to the dominance of shear. Figure 5.12(c) shows the observed stress pattern prior to failure. As can be seen from the stress contours the top half of the members in compression with maximum stress occurring at the center of the beam. The analytical failure was recorded at a load level of 352kN due to crushing of the concrete at the top. Experimentally observed ultimate load was 334kN which is 5.8% lower than the ultimate load predicted by the finite element analysis.

The predicted load-deformation response upto failure was compared with that obtained experimentally in figure 5.13. As can be seen from the figure the analytical load-deformation response shows good agreement with the actual response upto 50% of the ultimate load. After this load level a somewhat softer response was predicted by the analysis till failure. Inclusion of bond slip in the analysis did not affect the load deformation response and ultimate load in any appreciable manner.
Figure 5.11: Finite Element Mesh for RC Beam
Figure 5.12: Crack Pattern and Stress Contour in RC Beam
Figure 5.13: Load Deformation Response of Reinforced Concrete Beam
Chapter 6
Reinforcement Mesh Mapping

6.1 Introduction

The applicability of the embedded approach to represent reinforcement in finite element analysis of reinforced concrete structures can be extended to 3D problems, provided the difficulty involved in preparing data for the FE model is reduced. As explained in section 5.6 the embedded representation of steel requires the points of intersections of rebars with the 3D finite element faces in global coordinates. This is an involved task and often limits the application of the embedded approach. Also in cases where the reinforcement is distributed all over the structural member and no bars can be omitted from the finite element model, the application of embedded approach to three dimensional problems is almost impossible. For example, in a structural element like RC column the reinforcement consists of longitudinal bars and lateral ties, distributed all over the column, yet no portion of the reinforcement can be omitted in the finite element model as it would change the entire behavior of the member. In the present study, in order to overcome the above mentioned drawback of the embedded approach, a global rebar mesh mapping technique was developed and implemented.

This chapter presents a mesh mapping algorithm, its implementation and a analytical example showing the usefulness of the procedure in reducing the data preparation required for a finite element model.
6.2 Mesh Configuration

Usually, the information on any 3D finite element mesh (fig. 6.1) is available in the form of data containing the nodal coordinates and the element incidences. The 3D finite elements can be either 8 noded or 20 noded solid elements and in the present study only 20 noded elements were employed. From the given element connectivity data, element face information consisting of nodal coordinates and node numbers of the four corner nodes is generated for all the elements. For the purpose of locating the intersection points of rebars with the finite element faces each face is treated as a plane defined by a closed polygon of four sides and four corner nodes. This assumption is not a limitation considering the fact that most often regular finite element meshes are used to model RC structural elements. Even in cases where irregular meshes are used, the 3D solid elements used to represent the concrete are not generally curved.

![Figure 6.1: Rebar Intersection with Concrete Finite Element Face](image)

The input data for the global reinforcing bars is assumed to be consisting
of the two end point coordinates for each rebar. All the rebars are assumed to be represented by a straight line defined by the two end points of the rebar. This seems to be a reasonable assumption considering the fact that rarely the reinforcement is curved in a RC structural element. The coordinates of the back node \( P_1 \) and fore node \( P_2 \) (fig. 6.1) of each line segment, representing a straight rebar, is assumed to be given in the global \((x,y,z)\) coordinate system.

6.3 Rebar Intersection Points with 3D Finite Element Faces

After generating the desired concrete mesh using quadratic elements, the mesh of steel bar elements has to be generated using the points of intersections of the steel reinforcing bars with the 3D concrete element faces as the nodal points. For this purpose it is necessary to superimpose the reinforcement grid over the 3D finite element mesh and identify the exact locations on the finite element faces where a rebar enters and leaves a particular concrete element. This is done by considering each rebar, separately and represented by a straight line, for determining the intersection points. Each element may be cut by more than one rebar.

The procedure is begun by identifying the element containing the starting point \( P_1 \) of the straight line under consideration. For this purpose a procedure explained in the next section was used. After identifying the 3D element containing the starting point of the line segment under consideration, the location of the point where the line segment leaves the element is determined graphically by firing a ray in the direction of the line defining a rebar and finding its intersection with appropriate face of the current element. All the six faces of the 3D element are to be scanned for possible intersection with the fired ray. This may be explained analytically as follows.

The equation of the straight line, representing the orientation of a rebar, can be expressed in parametric form as (fig. 6.1)

\[
(P_2 - P_1)t + P_1 = P
\]  
(6.1)
in which \( P_1 \) and \( P_2 \) represent the back node and fore node of the line segment and \( P \) represents a generic point on the straight line. The parameter \( t \) assumes values ranging from zero to one. From equation 6.1 it could be seen that when \( t = 0 \) the generic point \( P \) corresponds to \( P_1 \) and when \( t = 1 \), \( P \) corresponds to \( P_2 \).

The equation of the plane, representing the element face, is given as

\[
(P - P_0) \cdot \vec{N} = 0
\]  

(6.2)

Where \( P_0 \) corresponds to a node on the face and \( P \) represents a generic point on the plane. \( \vec{N} \) represents the normal to the element face under consideration.

The spatial point of intersection of the line and plane is obtained by solving for the parameter \( t \) from the equations 6.1 and 6.2, as given by

\[
t = \frac{(P_0 - P_1) \cdot \vec{N}}{(P_2 - P_1) \cdot \vec{N}}
\]  

(6.3)

The value of \( t \) from equation 6.3 is substituted in equation 6.1 to obtain the point of intersection of the straight line representing the rebar and the parent element face. At this stage the portion of the length of the rebar segment contained in the current 3D finite element (parent element) and the face number of the element from which the rebar leaves the element are available.

The algorithm proceeds by shortening the line segment by an amount \( P_1P \) and shifting \( P_1 \) to \( P \) (fig. 6.1). Now the next element to be scanned, for intersection with the remaining line segment, is identified as the one which is sharing the face of the element from which the line segment left. This is done by scanning the six faces of all elements excluding the current element and picking the match.

As this algorithm proceeds, the line segment under consideration is completely cut into different segments contained in concrete elements. As and when the rebar is cut, the cut rebar segment is attached to the corresponding concrete finite element for considering the stiffness and forces in the concrete element. After defining the correspondence between the cut rebar segments and 3D finite elements, an element may contain one or more embedded rebar segments located in any arbitrary direction within the element.
6.4 Procedure to Identify the Concrete Element Containing a Given Point on a Rebar

Before each straight line, representing a global rebar, is considered for scanning, the element containing the starting point needs to be identified. Since it has been assumed that the given 3D finite element mesh contains 20-noded isoparametric elements, the task of identifying the element containing a given point is achieved by using the property of isoparametric finite element. For any global point lying within the element, the natural coordinates of the point will be within the range of -1 to 1 in all the three local coordinate directions \( \xi, \eta, \) and \( \zeta \) (fig. 5.4).

A given point in global coordinates, assuming that it lies in the 3D finite element under consideration, is mapped into the domain of the element under consideration by using the inverse transformation technique explained in section 5.7. While using the inverse transformation procedure, if the global point lies outside the concrete finite element and the element was severely distorted the mapping procedure did not converge. To overcome this problem of non-convergence, before proceeding with the inverse transformation procedure the global point has to be checked to find if it is contained within the global bounds of the geometry of the concrete finite element. The geometry bounds along \( x, y \) and \( z \) directions are defined by the minimum and maximum coordinate values of all the corner nodes of the 20 noded element under consideration. If the mapped point is found to be in the local domain of the finite element under consideration, i.e. all the three natural coordinates of the mapped point are within the range -1 to +1, this element is identified as the one containing the given global point. If not, the scanning proceeds with other elements.

While identifying the element containing a given point, difficulties are encountered when a rebar segment, such as a lateral tie, is located exactly on the face of a parent element or passing through the line of intersection of any two faces of the parent element. These circumstances are treated as special cases in the graphical procedure and considered appropriately. When a rebar segment is
found to be lying on the face of an element and the same face is shared by another element then a point on this segment is determined to be contained in two adjacent elements. In such case the first encountered element is identified as the element containing the point on the rebar.

6.5 Verification Analysis

The mesh mapping algorithm, presented in this chapter, was implemented in the finite element software “INARCS”. In order to verify the implemented procedure and to determine the points of intersections of the global rebars with the concrete finite element faces, an axially loaded test column (Scott et al. 1982) was considered. Figure 6.2 shows the details of the test column and finite element models used to represent the column. Considering the symmetry, one-eighth of the total column was represented by the finite element model.

Two meshes were considered to check the accuracy of the mesh mapping procedure. Three dimensional solid elements were used to represent the concrete. Coarse mesh consists of five elements of equal size along the length and four elements of equal size across the cross section. A three dimensional view of the finite element mesh for coarse mesh is shown in fig. 6.3.

The reinforcement was represented as a 3D mesh consisting of straight line segments as shown in fig. 6.3. Each longitudinal rebar was represented by a single line segment and each tie bar is represented by one or more straight line segments depending on the number of straight portions in the tie. For example a square tie, to be defined completely, would require four line segments to be specified. Any line segment, corresponding to a rebar or a portion of a rebar, was defined by two nodes, a back node and a fore node. All the nodes were specified in global coordinates.

The embedded rebar segments resulting from the mesh mapping procedure contain the information about their orientation and location in the given concrete finite element mesh. Each rebar segment in an element is viewed as a geometric line entity with back node and fore node defined in global coordinates. These cut
Figure 6.2: Column Details
Figure 6.3: Finite Element Model for Column and Reinforcement
rebar segments contained in all the finite elements of the mesh were assembled back to obtain the original rebar grid with the points of intersections indicated as rebar nodes. This was done only to verify the mesh mapping procedure and is not required for finite element analysis. A three dimensional view of the reinforcement embedded in the column, and the mapped reinforcement grid with intersection points are shown in fig. 6.4.

The significance of the implemented mesh mapping procedure is demonstrated further by considering the same concrete column with refined finite element mesh for concrete as shown in fig. 6.5. This implies a practical situation of studying the effect of mesh refinement without having to change the rebar grid data. Figure 6.5 also shows the mapped rebar segments in the fine mesh for the column.

The aim of this analysis was to demonstrate the ability of the mesh mapping procedure in allowing the researcher to choose the finite element mesh for concrete, independent of the geometry of the rebar. The observed results from the mesh mapping procedure have indicated that the exact locations of the rebars in the concrete elements could be obtained by the algorithm presented herein. The use of mesh mapping procedure simplified the data preparation involved in representing the reinforcement in the finite element model. The results of finite element analysis regarding the confinement effect and the load deformation response of the RC column are presented in the next chapter along with the additional numerical examples.
Figure 6.4: Mapped Rebar Segments for Coarse Mesh
Figure 6.5: Refined FE Mesh for Column and Mapped Segments
Chapter 7
Analyses of RC Structural Elements

7.1 Introduction

In chapters 3,4 and 5, the constitutive equations of plain concrete, cracked concrete and reinforcing steel were developed. These constitutive models were utilized in the theoretical evaluation of the behavior of concrete elements that were physically tested. Good correlation was obtained between the experimental results and the values predicted theoretically using these constitutive models.

In this chapter the selected constitutive models were used to analyze different RC structural elements to verify their ability to simulate the behavior of these elements. The following examples were considered to validate the implemented material models.

1. Plain concrete beam subjected to torsion

2. Reinforced concrete beam column

3. Reinforced concrete column

4. Beam-Column-Slab connection

Three dimensional solid (20-noded hexahedron) elements were employed to model the concrete elements in the above examples. The steel reinforcement was represented as embedded segments in concrete finite elements. The specimens were loaded under load or displacement control as appropriate for the analysis.
7.2 Plain Concrete Beam Subjected to Torsion

7.2.1 Introduction

A plain concrete cantilever beam subjected to torque at the free end was analyzed to demonstrate the ability of the cracking model to predict the torsional cracks. In order to verify the mesh independency of the results of analysis, three different finite element meshes were considered. This was a purely theoretical example constructed to test the objectivity in results of analysis. In following sections the member geometry, finite element idealization, load application and the results of analysis are discussed.

7.2.2 Finite Element Idealization

The geometry of the cantilever beam considered is shown in fig. 7.1(a). Material properties used in the analysis are listed in fig. 7.1(b). Finite element idealization of the plain concrete beam is shown in fig. 7.2. In all the finite element meshes considered the number of elements along the length of the beam was kept constant due to the fact that the torque is constant throughout the length of the cantilever beam. The refinement of the mesh was considered across the cross section in order to represent the stress gradient across the cross section. In all the three finite element meshes, the central band of elements (shown in shade in fig. 7.1(a)) was used to localize the fracture zone. The localization was induced by reducing the tensile strength of the concrete elements in the central band (fracture zone) by 10%.

In the finite element model the fixed end boundary condition was specified by restraining all the three translational degrees of freedom for all the nodes at the fixed end. Irons (1971) 15 point rule of integration was used in all the three analysis cases.
(a) Geometry description

\[ E = 20,000 \text{ N/mm}^2 \]
\[ f'_c = 35.0 \text{ N/mm}^2 \]
\[ f'_t = 3.85 \text{ N/mm}^2 \]
\[ G_f = 0.25 \text{ N/mm} \]
\[ u = 0.20 \]

(b) Material properties

(c) Loading

Figure 7.1: Torsion Beam Details
Figure 7.2: Finite Element Idealization of Torsion Beam
7.2.3 Load Application

The torque at the free end was applied by imposing nodal displacements as shown in fig. 7.1(c). A total displacement of 1mm in the direction perpendicular to the diagonal was applied at the four corner nodes in 120 steps. The size of the incremental step was chosen in such a way that the initiation of the cracking was confined to the fracture zone in a given increment.

7.2.4 Results of Analysis and Discussion

The specimen was analyzed using the three different FE meshes. The specimen was not loaded up to failure since the aim of this analysis was to check the objectivity in the response and ability of the smeared cracking model to represent diagonal cracks. Deformed shape of the torsion beam analyzed using the fine mesh (TOR180) is shown in figure 7.3.

Figure 7.3: Deformed Shape of Torsion Beam

The resulting crack pattern for the mesh TOR180 corresponding to a free
end rotation of 0.002 radians is shown in figure 7.4. As can be observed from the crack pattern in the fracture zone of the fine mesh, torsional cracks at 45° inclination occurred symmetrically across the cross section.

![Diagram of Crack Pattern in the Torsion Beam](image)

**Figure 7.4: Crack Pattern in the Torsion Beam**

Figure 7.5 shows the variation of maximum principal strain in the fine mesh corresponding to a free end rotation of 0.0035 radians. Localization of the fracture can be clearly explained by the strain contour showing the diagonal crack pattern in the central band of the elements. Few concrete elements outside the fracture zone were also found to have registered cracking. The loaded points at the free end were found to be severely strained. However, the stress concentration at the loaded points did not seem to affect the crack propagation in the fracture zone. The predicted cracking pattern was in accordance with the physical behavior of the specimen under torsion. The diagonal torsional cracks were simulated well by the current smeared crack model.

In order to plot the load deformation response, the applied torque at the free end was computed by taking the sum of two couples generated by the reacting
Figure 7.5: Maximum Principal Strain Contour for Torsion Beam
forces along the two diagonals. The free end rotation was taken as the ratio of the displacement $\delta$ at the corner node (fig. 7.1(c)) and the diagonal distance between the corner nodes (282.4mm). The observed load deformation response for all the three FE meshes is shown in fig. 7.6.

As could be seen from the predicted response, there was sudden change in the torsional stiffness of the beam when the free end rotation was approximately 0.001 radians. The kink in the deformation response was observed at the same load level for both the fine meshes TOR80 and TOR180. Also the post kink response was somewhat similar for both the fine meshes, when compared with the coarse mesh response. Analysis using the coarse mesh resulted in soft response in the final part of the loading. From the results of analysis for the two fine meshes, it can be concluded that physical behavior of the specimen would be closer to the response of fine mesh TOR180.

The difference in the load-deformation responses for TOR20 and other two fine meshes may be attributed to the choice of number of elements used across the cross section to represent the shear stress gradient. Referring to fig. 7.2(a), in the model with the coarse mesh, only one element used to represent the shear stress which varies from zero at center to maximum at the edge. This was certainly not a good mesh for representing the variation in the shear stress. Whereas in both the fine meshes, TOR80 and TOR180, the gradient in the shear stress across the cross section could be represented with reasonable accuracy.

Comparing the responses for the two fine meshes, it can be stated that refinement in the mesh yields a slightly stiffer response. This may be due to the propagation of the fracture, at higher load levels, from the central band of elements to the adjacent layers. However, the difference in response for the two meshes was not considerable. The characteristic length controlled the release of fracture energy in different meshes based on the location of the sampling point in an element and orientation of the crack.

It can be concluded that the used smeared cracking model was reasonably accurate in predicting the localized diagonal cracks in the beam. Also the results of analysis showed that the consistent evaluation of characteristic length based upon
the size of the finite element could be successfully used in achieving the objectivity in the results of analysis. Fig. 7.6 shows the degree of objectivity observed in the results of analysis.

![Figure 7.6: Load-Deformation response of Torsion Beam](image)

**Figure 7.6: Load-Deformation response of Torsion Beam**

### 7.3 Beam-Column

#### 7.3.1 Introduction

A reinforced concrete beam-column that was experimentally tested by Hisada et al. (1972) was analyzed. The test specimen was a half scale column subjected to repeated lateral forces and a constant axial force equal to one third of its compressive strength. In the present analysis the beam column was analyzed for monotonic loading only. The following sections give the details of geometry and finite element idealization and present the results of analysis.
7.3.2 Finite Element Idealization

The geometry and reinforcement details are given in fig. 7.7. Considering the antisymmetry, only half of the specimen was used in the analysis. The finite element mesh used to represent the beam column is shown in fig. 7.8. The boundary conditions at the fixed end were simulated by restraining the three translational degrees of freedom of all the nodes along the fixed edge. Also taking advantage of the symmetry across the cross section of the test specimen only half of the section was considered in the analysis. A three dimensional view of the reinforcement embedded in the concrete finite element model for the beam-column is shown in fig. 7.9.

7.3.3 Load Application

The analysis was carried out in two steps, first the axial load was applied to the specimen and then the transverse load was applied. Total axial force equal to one third of compressive strength 73.5t (720.3kN) was applied as uniformly distributed load on the faces of eight elements at the free end. The transverse load was applied as a uniform load, distributed over the two top elements at the free end. Total axial load was applied in two increments while the transverse load was applied in 80 increments. Material properties used in the analysis are given in tables 7.1 and 7.2.
Figure 7.7: Details of Beam-Column
Figure 7.8: Finite Element Idealization of Beam-Column
7.3.4 Results of Analysis and Discussion

Two step analysis of the model resulted in the failure at a load level of 82t (804.4kN). The experimentally observed failure was due to yielding of reinforcement.
Table 7.2: Material Properties of Reinforcement - Beam Column

<table>
<thead>
<tr>
<th>Reinforcement</th>
<th>Longitudinal</th>
<th>Transverse</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Bars</td>
<td>Bars</td>
</tr>
<tr>
<td>Yield Stress ($f_y$)</td>
<td>3700 kg/cm$^2$</td>
<td>3600 kg/cm$^2$</td>
</tr>
<tr>
<td>Yield Strain ($\epsilon_y$)</td>
<td>0.00176</td>
<td>0.00176</td>
</tr>
<tr>
<td>Ultimate Stress ($f_u$)</td>
<td>3890 kg/cm$^2$</td>
<td>3780 kg/cm$^2$</td>
</tr>
<tr>
<td>Ultimate Strain ($\epsilon_u$)</td>
<td>0.12</td>
<td>0.12</td>
</tr>
</tbody>
</table>

followed by crushing of concrete at a load of 77.2t (757.33kN). In the present analysis the yielding of steel was observed at 67t (657.27kN) as against the experimental load of 66.1t (648.44kN). Yielding of the steel in compression was followed by crushing failure of concrete at 82t (804.4kN) load. Yielding of the compression steel resulted in an increase in the number of iterations per increment by 3 for all the subsequent increments of loading. The load corresponding to the steel yielding was predicted with an error of 1.3% and the failure load with an error of 6.3%.

Figure 7.10 shows the deflected shape and the variation of normal stress in the specimen at 60% of the failure load. It can be seen from the stress contours, the elements at the bottom and top were subjected to compression and tension respectively. However, the severely loaded region was the junction of the main section and the experimental load application point.

The observed crack pattern at 70% of the failure load is shown in fig. 7.11. The crack pattern clearly indicates the fracture zone and its direction of propagation. The first cracks appeared at the junction of the main beam stub.

The load deformation response of the specimen is compared with the actual response in fig. 7.12. The specimen was also analyzed by considering the bond slip between the concrete and main reinforcement. The response obtained from the finite element analysis was relatively stiff in comparison with the actual response...
Figure 7.10: Stress Contour for Beam-Column
Figure 7.11: Crack Pattern in Beam-Column

Figure 7.12: Load Deformation Response for Beam-Column
for both cases, with and without the bond slip. This may be attributed to the fact that the actual test was carried out under cyclic loading and the damage to the concrete material due to cyclic loading was not considered in the present analysis. The bond slip did not seem to effect the global deformation response to a noticeable extent.

From the results of the analysis of the beam-column, it was observed that the load corresponding to the yielding of steel was predicted well (1.3% error) and the failure load was overestimated by 6.3%. However, considering the fact that damage to concrete due to cyclic loading also reduces the strength of the concrete, the error in the predicted failure load was not considerable. The crack pattern and the stress variation in the beam-column were in accordance with the physical behavior of the specimen.

### 7.4 Reinforced Concrete Column

The axially loaded test column (Scott et al. 1982) considered in the section 6.6 was analyzed to check the effectiveness of the embedded representation of steel in modeling the confinement of core concrete. The details of the geometry are shown in fig. 6.2. The material properties of concrete and steel used in the analysis are given in tables 7.3 and 7.4.

Table 7.3: Material Properties of Concrete - RC Column

<table>
<thead>
<tr>
<th>Concrete</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>2100 MPa</td>
</tr>
<tr>
<td>ν</td>
<td>0.2</td>
</tr>
<tr>
<td>$f'_{c}$</td>
<td>25.3 MPa</td>
</tr>
<tr>
<td>$f'_{t}$</td>
<td>2.53 MPa</td>
</tr>
</tbody>
</table>
Table 7.4: Material Properties of Reinforcement - RC Column

<table>
<thead>
<tr>
<th>Reinforcement</th>
<th>Main Bars</th>
<th>Ties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield Stress ($f_y$)</td>
<td>394 MPa</td>
<td>309 MPa</td>
</tr>
<tr>
<td>Yield Strain ($e_y$)</td>
<td>0.00186</td>
<td>0.00186</td>
</tr>
<tr>
<td>Ultimate Stress ($f_u$)</td>
<td>600 MPa</td>
<td>380 MPa</td>
</tr>
<tr>
<td>Ultimate Strain ($e_u$)</td>
<td>0.12</td>
<td>0.12</td>
</tr>
</tbody>
</table>

7.4.1 Finite Element Idealization

Taking advantage of the symmetry only one eighth of the column was considered in the analysis. The model and the details of the reinforcement are given in figure 6.3. In the present analysis coarse mesh was used to represent the concrete. The rebar mesh and finite element model for column are shown in figure 6.4.

7.4.2 Load Application

The specimen was loaded analytically under displacement control. This was necessary due to the fact that softening was expected in the global load deformation response for a RC column. In the present analysis a total displacem ent of 30mm was applied in 80 increments in the axial direction to all the nodes in the top face of the column. The boundary conditions were specified appropriately by restraining the translational degrees of freedom for all the nodes on the plane of symmetry.

7.4.3 Modeling of Crushed Concrete

The experimentally observed load deformation response of the test column is shown in fig. 7.13. These experimental results indicate a distinct post peak response for the test column. This response was due to the post peak softening
of concrete in compression. In order to simulate this softening, a stress reduction approach was employed in the present study.

Figure 7.13: Load Deformation Response of RC Column

After reaching the peak compressive stress at a material point, for a given strain increment, the corresponding stress decrement due to strain softening was computed as a fraction of the stress decrement resulting from an equal strain decrement. This was represented by a reduction factor, as explained below.

\[ \Delta \sigma = ([D]_T \cdot \Delta \epsilon) \cdot RF \]  

(7.1)

where

- \( \Delta \sigma \) is stress increment for strain increment of \( \Delta \epsilon \),
- \([D]_T\) is the tangent stiffness matrix
- \( RF = v_1 + (v_2 - v_1) \cdot \frac{\gamma_{su}}{\gamma_{el}} \) for \( \gamma_{cu} < \gamma < \gamma_w \)

in which \( \gamma_{cu} \) is octahedral shear strain corresponding to the peak compressive stress and \( \gamma_w \), taken as four times \( \gamma_{cu} \), is terminating the octahedral strain value for compression softening.
The value of the stress reduction factor $RF$ was evaluated based on the total octahedral shear strain at the crushed material sampling point. The values of the parameters $v_1$ and $v_2$ were chosen by trial and error procedure to fit the available experimental results (Kupfer 1968) for a uniaxial case and the results are shown in figure 7.14. The values of the parameters arrived at were $v_1 = -0.2$ and $v_2 = -0.12$ and the same were used in the analysis of test column.

![Stress (ksi)](image)

Figure 7.14: Comparison of Experimental and Analytical Stress Strain Relationship

7.4.4 Results of Analysis and Discussion

The analytically obtained load deformation response is compared with the experimental response as shown in fig. 7.13. The results of analysis of the column were found to be in good agreement with the observed experimental values. This example demonstrated the effectiveness of the embedded representation of lateral ties in representing the confinement of the core concrete. To isolate the effect of
inclusion of lateral ties in this example, the RC column was analyzed with and without the transverse reinforcement. The main reinforcement was considered in both the analyses.

The column without the lateral ties failed in compression as soon as the axial stress reached the uniaxial strength of the concrete. As expected, in this analysis case no confinement was observed in any of the concrete elements and all the elements were uniformly stressed in the axial direction. The load corresponding to the crushing was observed to be 6843kN.

In the analysis case including ties, the crushing failure was observed first in the elements near the cover at a higher load level. The elements in the cover region were found to be constrained in the lateral direction of the ties due to bond between concrete and ties resulting in a biaxial stress state. This resulted in the increased ultimate strength of the concrete elements in the cover region. The elements in the core region were found to be confined in both lateral directions uniformly. Though the core elements did not contain any lateral ties, the confinement resulted from the deformational resistance induced by the ties contained in the adjacent elements. As a consequence of the confinement of the core concrete, a higher peak load of 7240kN was observed as compared to the experimental peak load of 6780kN.

The analysis of the RC column demonstrated the ability of the embedded representation of steel in simulating the confinement effect of core concrete due to lateral ties. However the peak load was overestimated by 6.7% while the post peak response could be simulated well by a compression softening model presented herein.

7.5 Beam-Column-Slab Connection

7.5.1 Introduction

An exterior beam to column connection that was experimentally tested at Rice University (Zerbe 1985) was selected for the analysis. The test simulated the effect of an earthquake type loading on exterior beam to column connection of a
reinforced concrete frame building with a monolithic slab. In the present analysis the connection was loaded monotonically till failure and the results of analysis are discussed in detail in the following sections.

7.5.2 Finite Element Idealization

The schematic diagram of the test subassemblage is shown in fig. 7.15. Details of the geometry and the reinforcement in column and main beam are given in fig. 7.16. The details of the transverse beam and slab are shown in fig. 7.17. Taking advantage of the symmetry only half of the connection was modeled in the analysis. Details of the finite element idealization are shown in plan and elevation in fig. 7.18. Three dimensional views of the finite element model for the beam column slab connection are shown in fig. 7.19.

Three dimensional 20 noded solid elements were used to represent the concrete in the finite element model. Six, eight and nine elements were used across the cross sections of the column, main beam and transverse beam respectively. Across the depth of the slab two elements were used to represent the concrete. Since the 3D quadratic element is capable of representing a linear variation in the strain field, two elements were assumed to be enough to represent the strain gradient across the slab.
Figure 7.15: Specimen Configuration
Figure 7.16: Main Beam and Column Reinforcement Details
Figure 7.17: Slab Reinforcement Details
Figure 7.18: Finite Element Idealization (Elevation and Plan)
Figure 7.19: Finite Element Model for the Connection (3D-View)
The total number of 20 noded elements in the finite element model was chosen based on the strain variation at different locations in the connection, the limiting aspect ratio of the elements and the available computer memory for processing the data. In the present finite element model the total number of 20 noded elements was 344 and the total number of nodes associated with 3D elements was 2162. The maximum aspect ratio of the finite elements was limited to 3.44, which corresponds to the elements in the slab region.

The reinforcement was represented as embedded rebar segments at their exact locations. A three dimensional view of the mesh used to represent the reinforcement in the connection is shown in fig. 7.20. All the rebars were defined as geometric line entities as required by the mesh mapping procedure explained in chapter 6. A three dimensional view of the mapped reinforcement embedded in the concrete finite element mesh is shown in fig. 7.21. After mapping the rebar mesh into the 3D concrete finite elements the total number of segments was found to be 655 and the total number of nodes associated with the rebar segments was 1487.

7.5.3 Load Application, Boundary Conditions and Material Properties

In the experiment two types of load were applied to the test specimen. An axial compressive load (178kN) was applied to the column and cyclic load was applied at the free end of the main beam (fig. 7.22). The specimen was tested with column standing vertically and connected to the reaction frame through mechanical hinges.

In the present analysis a total axial load of 178kN was applied as uniform load on the top face of the column over six elements across the cross section. In order to facilitate the load application at the free end of the main beam a layer of finite elements were considered with high strength. The vertical load at the end of the main beam was simulated by displacing the nodes on top of the elements at the free end. The applied load was computed as the sum of the reactions generated at the nodes which were displaced.
Figure 7.20: 3D-Views of Reinforcement Mesh
Figure 7.21: 3D-View of Reinforcement Embedded in Concrete FE mesh
The boundary conditions along the central plane of symmetry were modeled by restraining the translational degrees of freedom in z-direction (refer fig. 7.19) for all the nodes on the plane of symmetry. The mechanical hinges at the top and bottom of the column were simulated by restraining the translational degree of freedom in y-direction for the nodes along the central line of the column face at the top and bottom. The material properties used in the analysis are as given in table 7.5 and 7.6.

7.5.4 Results of Analysis and Discussion

The analysis of the beam-column-slab connection was carried out by first applying the axial load on the column and then the vertical load at the free end of the main beam. Two cases were considered for the analysis, viz. the upward load at the free end and the downward load at the free end.

The upward loading, referred as positive loading, resulted in the slab being
Table 7.5: Material Properties of Concrete - RC Connection

<table>
<thead>
<tr>
<th>Concrete</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>38338.6 MPa</td>
</tr>
<tr>
<td>ν</td>
<td>0.2</td>
</tr>
<tr>
<td>$f'_c$</td>
<td>45.2 MPa</td>
</tr>
<tr>
<td>$f'_t$</td>
<td>4.97 MPa</td>
</tr>
<tr>
<td>$G_f$</td>
<td>0.3 N/mm</td>
</tr>
</tbody>
</table>

Table 7.6: Material Properties of Reinforcement - RC Connection

<table>
<thead>
<tr>
<th>Reinforcing Steel</th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Designation</td>
<td>Yield Stress</td>
<td>Yield Stress</td>
<td>Ultimate Stress</td>
<td>Ultimate Stress</td>
</tr>
<tr>
<td></td>
<td>$f'_y$ (MPa)</td>
<td>$\varepsilon'_y$</td>
<td>$f'_u$ (MPa)</td>
<td>$\varepsilon'_u$</td>
</tr>
<tr>
<td>#4</td>
<td>531.0</td>
<td>0.00256</td>
<td>751.7</td>
<td>0.0146</td>
</tr>
<tr>
<td>#6</td>
<td>413.8</td>
<td>0.00214</td>
<td>634.5</td>
<td>0.0121</td>
</tr>
<tr>
<td>#8</td>
<td>482.8</td>
<td>0.00241</td>
<td>717.2</td>
<td>0.0124</td>
</tr>
</tbody>
</table>

subjected to compression and effectively acting as a flange in compression for the main beam. The analysis resulted in numerical failure at a load level of 142kN as against the experimentally observed value of 129.9 kN. The failure of the numerical analysis was observed due to extensive cracking and yielding of reinforcement in the main beam and joint region, and subsequent divergence of the iterative procedure. The analytically obtained deflected shape of the connection near failure load is shown in fig. 7.23.
Figure 7.23: Deformed Shape of Connection under Positive Loading
Figure 7.24: Minimum Principal Stress Contour for Positive Loading
The observed stress pattern in the connection near the failure load is shown in fig. 7.24 by means of the minimum principal stress contour. As can be seen from the fig. 7.24 the slab was under compression, with increased levels of compressive stress in the region near the main beam. This indicates that some portion of the slab acts as a flange in compression for the main beam closer to the joint. Figure 7.25 shows the variation of normal stress along the length of the main beam. As it can be observed from this figure, the compression in the main beam is maximum at the top near the joint. Figure 7.26 shows the variation of the shear stress in the joint region.

Positive loading of the connection resulted in extensive cracking in the main beam, column and in the bottom elements of the slab. Analytically observed crack pattern in the slab at the failure load level is shown in figure 7.27. Only the top elements of the slab have shown cracking with most of the cracking concentrated at the junction of transverse beam and the slab. There was no cracking observed in the transverse beam. In the column horizontal cracks were seen at top and and bottom due to bending.

Figure 7.28 shows the observed crack pattern at the failure load level in the main beam and column. Cracks in the main beam appeared in a regular pattern with intensive cracking near the joint region. Flexural cracks in the column, shear cracks in the joint together with cracks in main beam formed a hinge like region at the junction of the column and main beam.

When a downward load was applied at the free end of the main beam, referred to as negative loading, the slab is subjected to tension and a considerable portion of the slab acts as flange in tension for the main beam. The analysis of the connection under negative loading resulted in numerical failure at a load level of 155.2 kN as against the experimentally observed load of 148.8 kN. The observed failure was due to extensive cracking in the main beam, slab, joint region followed by yielding of reinforcement in slab and subsequent divergence of the iterative procedure. Analytically obtained deflected shape of the connection at failure load level is shown in figure 7.29. From the deflected shape it could be seen that the column was bent in double curvature.
Figure 7.25: Normal Stress Variation in Main Beam for Positive Loading
Figure 7.26: Shear Stress Contour for Positive Loading
Figure 7.27: Crack Pattern in Slab for Positive Loading
Figure 7.28: Crack Pattern in Main Beam and Column for Positive Loading
Figure 7.29: Deflected Shape of Connection under Negative Loading
Figure 7.30: Normal Stress Contour for Negative Loading.
The observed stress pattern in the main beam and the column is shown in figure 7.30 by means of the normal stress contour. As it could be seen from the figure the main beam was subjected to compressive stress at the bottom with increased stress levels towards the joint. The tensile stresses were seen throughout the length at the top of the main beam extending into the joint region. Figure 7.31 shows the variation in shear stress in the connection at failure load level.

Negative loading of the connection resulted in extensive cracking in the slab, main beam, column and transverse beam. The observed crack pattern at the failure load level in the slab is shown in figure 7.33. Unlike in the positive loading case, both the elements across the depth of the slab have cracked in the region closer to the transverse beam.

Crack pattern in the main beam and column is shown in figure 7.33. Cracking in the main beam was limited to the slab above it. Cracks in the column and slab joined together at the joint region and formed a hinge like region at the face of the column. Figure 7.34 shows the crack pattern in the transverse beam. The transverse beam exhibited a diagonal crack pattern due to torsion.

Experimentally observed crack pattern in the slab (in the connection region) under negative loading is shown in figure 7.35. Comparing the figures 7.32 and 7.35 for crack patterns in the slab, it can be seen that the analytically obtained crack pattern was very close to that obtained experimentally. The transverse cracks in the slab near the joint-beam region were inclined and joined the torsional cracks in the transverse beam.
Figure 7.31: Shear Stress Contour for Negative Loading
Figure 7.32: Crack Pattern in Slab under Negative Loading
Figure 7.33: Crack Pattern in Main Beam and Column under Negative Loading
Figure 7.34: Crack Pattern in Transverse Beam under Negative Loading
Figure 7.36 shows the spalled concrete at the column back cover and the torsional cracks in the transverse beam. By comparing figures 7.34 and 7.36, it can be explained that the experimentally observed torsional cracks in the transverse beam were predicted with a reasonable accuracy, using a smeared approach by the numerical analysis. Figure 7.37 shows the failure surface of the connection that was observed in the test. Though the failure surface was not directly indicated by the analysis results, the analytically predicted crack pattern shows a close resemblance with the failure surface.

Figure 7.35: Transverse Cracks in the Slab
Figure 7.36: Torsional Cracks in Transverse Beam
Figure 7.37: Failure Surface in the Connection
Figure 7.38: Stiffness Degradation of the Connection

Envelopes of load-displacement curve and the analytically obtained deformational response are compared in the figure 7.39. Analytical failure load overestimated the experimental failure load by 9.3% and 4.2% for positive and negative loading cases respectively. In both the cases of analysis the analytical response was stiffer when compared with the experimental response. The stiff response could be explained by considering the experimentally observed degradation in the stiffness of the connection due to cyclic loading as shown in figure 7.38. As can be seen from this figure the damage to the concrete material due to cyclic loading was considerable. For example, there was a 40% reduction in the positive stiffness of the connection at the end of the second cycle. Since the stiffness degradation of the concrete due to cyclic loading was not accounted for in the present analysis, the analytical response was stiffer than that obtained from the experiment.
Figure 7.39: Load Deformation Response of Connection
Chapter 8
Summary and Conclusions

8.1 Summary

A comprehensive finite element model for studying the nonlinear three dimensional response of reinforced concrete structural elements has been developed in this study. For simulating the pre-cracking response of concrete under triaxial stress states, a slightly modified form of a hypoelastic constitutive model proposed by Stankowski and Gerstle (1985) is employed. This model is simple and requires few material parameters for its calibration. A five parameter strength envelope proposed by Willam and Warnke (1975) was used in conjunction with the constitutive model to detect failure in concrete. The stress-strain response of concrete under multiaxial loading conditions has been verified in the present study by comparing the analytical results with available test data from different experimental studies.

Post cracking strain softening behavior of concrete under triaxial loadings has been modeled in the context of a smeared crack approach following the procedure proposed by de Borst and Nauta (1985). The cracking model can simulate multiple non-orthogonal cracks at a sampling point. To maintain the finite element mesh objectivity with respect to crack propagation, a method proposed by Oliver (1989) has been extended to three dimensional applications.

In order to simulate the steel reinforcing bars in concrete, an embedded formulation proposed for two dimensional applications by Elwi and Hrudey (1989) has been extended to three dimensional cases. An automatic mesh mapping procedure has been developed to alleviate the difficulties associated with the preparation of data for computational models.
The mesh mapping procedure conveniently simulates individual steel bar elements in their spatial locations within the 20 noded solid isoparametric concrete finite elements and also determines the intersection points of those element faces with each rebar. The accuracy of the proposed procedure is verified by considering a reinforced concrete column with different finite element meshes.

Additional "slip degrees of freedom" have been introduced at the intersection of rebar elements with the concrete element in order to account for possible loss of bond between concrete and reinforcement. The effect of bond slip on global response is studied by analyzing a reinforced concrete beam and a beam-column.

Proposed analytical models for simulating the behavior of concrete are implemented in an integrated software package (INARCS) and the capabilities of analytical models have been examined by analyzing the response of various components of RC framed structures.

### 8.2 Conclusions

The hypoelastic model for concrete, with the proposed modification, is able to represent the stress-strain response of concrete under different loading conditions. The response of concrete under multiaxial loadings is represented well by the constitutive model up to 85% of the ultimate load, after that a slightly stiffer response is simulated. In triaxial loading cases the response is simulated well for both proportional and non-proportional loading cases. The constitutive model is numerically efficient and simple.

The strength envelope used in conjunction with the concrete constitutive model is accurate in predicting the failure of concrete under multiaxial stress states. The input parameters for the failure envelope, calibrated using the available experimental data, are able to represent the concrete strength under various loading conditions with reasonable accuracy.

The smeared cracking approach used to model the behavior of cracked concrete in three dimensions is able to represent the localized fracture as well as distributed cracking reasonably well. This is demonstrated by the analysis of plain
concrete notched beam and reinforced concrete beam without stirrups. The geometric interpretation of the consistent characteristic length has simplified its computation in three dimensional problems. The use of characteristic length to overcome the drawback of the smeared cracking approach (mesh dependency) has yielded objective results with mesh refinement. This is demonstrated by the analysis of plain concrete beam under torsion with different finite element meshes through the use of consistent characteristic length in conjunction with the smeared cracking technique.

The use of embedded formulation for the reinforcing steel has been found to be computationally advantageous. This approach permits all the reinforcing bars to be represented at their exact location without constraining the selection of size and shape of the finite elements used to model concrete. The analysis of reinforced concrete column demonstrated the ability of the embedded representation of steel in simulating the confinement effect on concrete. Modeling bond slip through the addition of slip degrees of freedom is feasible and efficient in the three dimensional applications.

The developed mesh mapping procedure made the use of embedded technique to represent the steel in three dimensional applications a feasible task. This procedure allows the complicated reinforcement configuration to be considered exactly in the analytical model and precludes the limitations on the choice of the gridwork for the parent concrete finite element mesh. Also the amount of work involved in preparing the data for a finite element model is significantly reduced by using the proposed mesh mapping procedure.

The analyses performed using the implemented procedures have simulated the response of RC structural components adequately. The gradual loss of stiffness due to cyclic loading has resulted in a softer experimental response than the analytical response. The results of analyses of beam-column-slab connection have indicated that the experimentally observed crack pattern in different regions are simulated reasonably well. The good agreement obtained in the examples between the numerical predictions and the experimental results establish the validity and accuracy of using the proposed procedures in modeling the reinforced concrete structures behavior.
8.3 Suggestions for Further Work

Some possible extensions to the present investigation and the analysis techniques are identified below:

1. The material models, presented in this study, for plain concrete, reinforcement and cracked reinforced concrete could be extended to include the effects of temperature, creep and shrinkage of concrete, and cyclic loading.

2. Develop numerical procedures that enable the analysis to be continued even after extensive cracking, yielding of steel and crushing of concrete have taken place.

3. Using the analytical model presented herein, a rigorous parametric study of beam-column-slab connections subjected to dynamic loading can be carried out.

4. Numerical studies to investigate the effect of spacing of main reinforcement in reinforced concrete columns can be carried out using the proposed mapping procedure.
References


ASCE Task Committee on Concrete and Masonry Structures. (1982). Finite Element Analysis of Reinforced Concrete Structures, ASCE, New York, N.Y.

ASCE Task Committee on Concrete and Masonry Structures. (1985). Finite Element Analysis of Reinforced Concrete Structures, ASCE, New York, N.Y.


Barzegar, F. (1989). "3-D Response of Reinforced Concrete Beam-column-slab Connections," Annual Report on a research project sponsored by the National Science Foundation, Dept. of Civil Engng., Louisiana State University, Baton Rouge, Louisiana, June, 90 pages.


Hsu, T. T. C. (1968). "Torsion of Structural Concrete-Behavior of Reinforced Concrete Rectangular Members," Torsion of Structural Concrete, sp-18, American Concrete Institute, Detroit, pp. 261-306.


Vita

Srinivas Maddipudi was born on 19th May 1965, in Andhra Pradesh, India. He graduated from the Andhra University, India, in June 1985 with a Bachelor of Technology degree in Civil Engineering. He entered the Civil Engineering Department at the Indian Institute of Technology Madras, India, in June 1985 and completed his post graduate work in December 1986, earning a Master of Technology degree in Civil Engineering. After receiving his Master's degree he worked as Senior Project Officer with Industrial Consultancy and Scientific Research at Indian Institute of Technology Madras for eight months. In August of 1987 he joined as Lecturer in Civil Engineering in College of Engineering, Gandhi Institute of Technology and Management, Visakhapatnam, India and worked for one year till August 1988. He joined the graduate program at Louisiana State University, Baton Rouge, in August 1988 to pursue Doctor of Philosophy degree in Civil Engineering. He is currently a candidate for the Ph.D. degree in Civil Engineering at Louisiana State University.
DOCTORAL EXAMINATION AND DISSERTATION REPORT

Candidate:  Srinivas Maddipudi

Major Field:  Civil Engineering

Title of Dissertation:  Three Dimensional Nonlinear Analysis of Components of Reinforced Concrete Framed Structures

Approved:

Vijaya K. A. Cepu
Major Professor and Chairman

Dean of the Graduate School

EXAMINING COMMITTEE:

Date of Examination:

8, May 1992