Hysteresis Modeling of Reinforced Concrete Members Subjected to Combined Loading

By

Mr. Yasuo Murayama
Mr. Seiji Tokuyama
Mr. Kosuke Furuichi
Mr. Hachiro Ukon
Mr. Yoshihiro Hishiki
Dr. Norio Inoue

Prof. Stephen Mahin
Prof. Christopher Thewalt
Mr. Changrui Yin
Mr. Božidar Stojadinović

Report No. CK 91-03
February 1991

California Universities for Research in Earthquake Engineering (CUREe)  Kajima Corporation
CUREe (California Universities for Research in Earthquake Engineering)

- California Institute of Technology
- Stanford University
- University of California, Berkeley
- University of California, Davis
- University of California, Irvine
- University of California, Los Angeles
- University of California, San Diego
- University of Southern California

Kajima Corporation

- Kajima Institute of Construction Technology
- Information Processing Center
- Structural Department, Architectural Division
- Civil Engineering Design Division
- Kobori Research Complex
Hysteresis Modeling of Reinforced Concrete Bridge Members Subjected to Combined Loading

Final Project Report

Božidar Stojadinović
Changrui Yin
Christopher R. Thewalt
Stephen A. Mahin

Earthquake Engineering Research Center,
Department of Civil Engineering,
University of California, Berkeley

March 1, 1991
Abstract

This report presents the work done during the CUREe-Kajima research project “Hysteresis Modeling of Reinforced Concrete Bridge Members Subjected to Combined Loading”. Two major tasks are addressed in this report:

- the verification of the fiber model for modeling the hysteresis of reinforced concrete columns subjected to bilateral loading and varying axial load, and
- the development of an improved stand alone cross-section analysis program, AfS.

The report concludes with a summary of the results achieved and outlines of research opportunities arising out of this project.
Acknowledgment

The authors gratefully acknowledge the support of the CUREe-Kajima Project. The help of C. A. Zaris from the Department of Civil Engineering, National Technical University of Athens, Greece, in performing the verification study is greatly appreciated.

The findings, observations and conclusions stated in this report are those of the authors alone.

Product and brand names mentioned herein may be trademarks and registered trademarks of their respective holders.
Contents

Abstract ii
Acknowledgment iii

1 Introduction 1

2 Verification of the Fiber Element 3
  2.1 Theory of the Fiber Finite Element Models 3
    2.1.1 Fiber Beam-Column Element 4
    2.1.2 Fiber Spring-Joint Element 6
  2.2 Description of Experiments 7
  2.3 Element Verification 14
    2.3.1 Introduction 14
    2.3.2 Modeling 14
    2.3.3 Comparison of Experimental and Analytical Results 16
  2.4 Parametric Studies 17
    2.4.1 Modeling of Steel Properties 17
    2.4.2 Number and Distribution of Sections 18
    2.4.3 Number and Distribution of Fibers In Cross-Section 20
    2.4.4 Inclusion of Spring-Joint Model 23
    2.4.5 Numerical Sensitivity of Results 26
  2.5 Conclusions and Recommendations 32

3 AcS—the Section Analysis Program 33
  3.1 Functional Specification 33
    3.1.1 Cross-Section Analysis 34
    3.1.2 Functional Summary 38
    3.1.3 Development Schedule 39
  3.2 Design Description 39
    3.2.1 Analysis Module 41
    3.2.2 User Interface Module 42
  3.3 Status and Pending Tasks 43
  3.4 Summary 43

4 Conclusion 44

References 45
A Sample Problem Definition in AcS
A.1 Cross-Section Geometry ........................................... 48
A.2 Cross-Section Materials ............................................. 53
A.3 Cross-Section Analysis Task Definition .......................... 53
List of Figures

2.1 Fiber Beam-Column Element ........................................ 4
2.2 Fiber Spring-Joint Element ........................................ 6
2.3 Specimen Cross-section ........................................... 8
2.4 Specimen Elevation: X Direction .................................. 9
2.5 Specimen Elevation: Y Direction .................................. 10
2.6 Experimental Setup: Elevation .................................... 11
2.7 Experimental Setup: Top View .................................... 11
2.8 Uniaxial Bending Load ............................................. 12
2.9 Diagonal Biaxial Bending Load ................................... 12
2.10 Clover-Leaf Lateral Bending Load ............................... 13
2.11 Linearly Varying Axial Load ..................................... 13
2.12 Section Distribution Along Specimen ......................... 14
2.13 Fiber Distribution Across Cross Section ....................... 15
2.14 Concrete Model for Fiber Column Element ..................... 16
2.15 Menegotto-Pinto Steel Model .................................... 17
2.16 Material Model for Steel Spring ............................... 18
2.17 Moment/Displacement Diagram for Specimen 4 ............... 19
2.18 Moment/Displacement Diagram for Specimen 5 ............... 19
2.19 Effects of Steel Modeling on Specimen 1: Fixed Base Condition 19
2.20 Effect of Number of Sections on Specimen 1: Fixed Base Condition 20
2.21 Effect of Number and Distribution of Fibers for Specimen 1: Fixed Base Condition 22
2.22 Effect of Joint-Spring on Specimen 1 Behavior .................. 23
2.23 Correlation with Fiber Model on Joint-Spring Element. Specimens 2 and 3 24
2.24 Correlation with Fiber Model on Joint-Spring Element. Specimens 4 and 5 25
2.25 Results of Specimen 4 Using Different $M$ and $b$ ............ 27
2.26 Results of Specimen 5 Using Different $M$ and $b$: Part 1 ... 28
2.27 Results of Specimen 5 Using Different $M$ and $b$: Part 2 ... 29
2.28 Comparison with 4 Spring Model Results ....................... 30
2.29 Results for Multiple Elements .................................. 31

3.1 Cross-Section Continuum Model ................................. 35
3.2 Cross-Section Fiber Model ....................................... 36
3.3 Modes of Deformation ............................................. 37
3.4 Fiber Material Models ........................................... 37
3.5 Communication Links in AE S .................................... 40

A.1 Sample Cross-Section ........................................... 47
A.2 Sample Load Pattern .............................................. 48
A.3 Quadrilateral Primitive ......................................... 49
Chapter 1

Introduction

The analysis of the non-linear and inelastic behavior of reinforced concrete columns has been the focus of research efforts for a long time. However, a wide variety of physical phenomena have made analytical modeling extremely difficult [Mahin 88]. Two dominant approaches are pursued in contemporary engineering practice: the finite element model approach and the phenomenological model approach. Experience with finite element models shows that they can reproduce fundamental flexural the inelastic behavior of concrete columns to the extent of the accuracy of the employed material model, often at a prohibitive computational cost.

Phenomenological models have proven to be more successful and, most importantly, more applicable in practice. These models can be classified into three categories:

- generalized plastic hinge models;
- concentrated spring models;
- fiber models.

The merits and shortcomings of each one of these categories have been discussed in an array of publications. Summaries are presented in several the state of the art reports [Mahin 88, Shibata 88].

The objective of this research project is to develop, implement, verify and apply a series of phenomenological models to the analysis of the non-linear and inelastic behavior of bridge and building column sections and members under multi-directional loading. Emphasis is placed on two principal topics. One is the development of computer-aided analysis tools. The other is improving understanding of the behavior of simple reinforced concrete structures under multi-directional loading.

A research plan consisting of seven interrelated tasks has been undertaken in cooperation with researchers at the Kajima Corporation. [Mahin 89]. These tasks are:

1. Formulation of the computing environment for the CURee-Kajima project;
2. Development of the stand-alone cross-section analysis computer program;
3. Implementation of generalized hinge, concentrated spring and fiber models for columns subjected to multi-directional loading;
4. Verification of these models against the available experimental data;
5. Studies to assess the numerical aspects of modeling;
6. Application of the developed models to prototype structures;
7. Documentation of results.
Chapter 1. Introduction

The tasks performed during the first half of the research period concentrated to the following areas:

- verification of the fiber finite element model on the basis of the experimental data obtained by S. S. Low and J. P. Moehle [Low 87];
- studies of numerical efficiency of the fiber model;
- development of the functional specification for the user interface and analysis part of the stand-alone cross-section analysis program;
- documentation of the achieved results.

The results were reported in the Interim Project Report [Mahin 90].

The tasks performed during the second half of the research period are:

- extensive verification of the fiber element model and studies of numerical stability with respect to:
  - cross-section discretization;
  - longitudinal discretization;
  - importance of the spring-joint model;
  - material modeling;
- analysis of the optimal discretization, material modeling and iteration parameters with respect to total computation costs and quality of obtained results;
- design and development of the stand-alone cross-section analysis program AfeS;
- additional detailed documentation of the achieved results.

This report presents an overview of the results achieved to date. Chapter 2 addresses the verification and numerical stability investigation of the fiber finite element model. Chapter 3 describes the functional specification, design and current status of AfeS, the cross-section analysis program. A summary of the results achieved and the recommendations for future work in this field are outlined in Chapter 4. Finally, a sample cross-section analysis problem definition is presented in the Appendix.
Chapter 2

Verification of the Fiber Element

The primary goal of the first part of this research program was to verify the fiber-based models of reinforced concrete beam-column members proposed in [Zeris 86]. The verification is conducted on the basis of the experimental results of biaxial bending of a quarter scale beam-column element reported in [Low 87]. The success of this verification is crucial for the acceptance of the fiber modeling approach for reinforced concrete columns.

2.1 Theory of the Fiber Finite Element Models

Framed structures generally consist of beam-column, bar and connection elements. Beam-columns and bars are three-dimensional continua, with a pronounced longitudinal dimension, significantly larger than the cross-sectional dimensions. Connections are, in most cases, assumed to be two-dimensional zero-length continua. The accepted practice of realistic three-dimensional framed structures analysis requires modeling of

- the axial force-deformation relation;
- the bending moment-rotation relation in both directions;
- the torque-twist relation.

Mathematical models of beam-column and bar elements can be treated, in accordance with the geometric idealizations introduced above, as one-dimensional continua, consisting of an infinite amount of "slices". Each slice is an infinitesimally thin three-dimensional element, with finite cross-sectional dimensions, consisting of infinitesimal differential volumes—fibers. Overall material properties of a beam-column element are obtained by integration, generally performed in two stages, first at the slice level, and then along the element. Material properties may be included at the element level, the slice level, or at the fiber level. The lower the level of introduction of material constitutive relations in the integration process, the better and more rational the element behavior becomes.

The fiber finite element models are based on the premise that the introduction of material constitutive relations at the fiber level will produce realistic element behavior, albeit at the cost of additional computing time. Increased realism and the ability of the element to perform under generalized excitation compensate for the increased computational effort.

Two finite element models used in this project belong to the class of fiber finite element models. One is the fiber beam-column element, hereafter referred to as the fiber element. The other is the fiber connection element, referred to as the spring-joint element. Both elements are implemented by C. A. Zeris [Zeris 86, Zeris 90a] within ANSR-I [Mondkar 75], a non-linear analysis finite element program.
2.1.1 Fiber Beam-Column Element

The fiber beam-column element models prismatic structural members with a straight longitudinal axis. Typically, the member is discretized longitudinally using several slices. At least two slices, one at each end, must be defined. Each slice—cross-section—is further discretized into a number of finite steel and concrete fibers (Figure 2.1).

The principal modeling assumption, at the slice level, is that plane sections remain plane. Longitudinally, the behavior of the element is governed by the assumption of linear flexibility variation between the monitored slices. Therefore, the slice locations (apart from the two compulsory end slices) along the element have to be established according to realistic assumptions about the actual flexibility distribution. Variations of slice properties are permitted along the length, with the provision that the interior slices must be at least as strong as the end slices in order to avoid convergence problems.

Interior kinematic transformations are defined assuming small displacements. Shear and torque deformations at the element level and internal shear-flexure interactions are ignored. Distributed transverse loads along the element are not allowed. In addition, the plane sections assumption precludes the inclusion of bond-slip influence on end slice rotations. Bond-slip effects can be included using the spring-joint element.

The discretization and analysis of a slice is similar to the analysis of the reinforced concrete cross-section presented in Section 3.1.1. A slice is an arbitrary shaped cross-section, defined as an assemblage of steel and concrete fibers in and orthonormal right-hand coordinate system, perpendicular to the longitudinal axis of the element. All fibers are assumed to be under uniaxial stress-strain state, thus requiring only uniaxial material models.

The incremental slice kinematic degrees of freedom

\[ dr_s = \{d\varepsilon_s, d\psi_x, d\psi_y\}, \]
are the slice reference strain and the two orthogonal slice rotations. The corresponding incremental slice resistance degrees of freedom

\[ dR_s = \{dN_x, dM_{xz}, dM_{zy}\} \]

are the slice axial force and the two orthogonal slice moments. Under the plane section assumption, the slice tangent stiffness matrix is computed as

\[
k_{st} = \sum_i \begin{bmatrix} 1 \\ -y_i \\ x_i \end{bmatrix} E_i A_i \text{d}x \left[ 1 - y_i x_i \right],
\]

where
- \(E_i\) is the fiber current tangent modulus;
- \(A_i\) is the fiber area;
- \(x_i\) and \(y_i\) are the coordinates of the fiber centroid with respect to the slice principal axis.

The flexibility of a slice is obtained by inverting the slice stiffness matrix, i.e.

\[ f_{st} = k_{st}^{-1}. \]

The available material models include:
- bilinear hysteretic steel model;
- explicit exponential Menegotto-Pinto steel model [Menegotto 77];
- multilinear concrete model, with no tensile strength and the unloading modulus equal to the initial elastic modulus [Kent 71].

The fiber beam-column element kinematic degrees of freedom at element ends \(i\) and \(j\) are

\[ d\mathbf{r}_e = \{d\delta, d\theta_x^i, d\theta_y^i, d\theta_x^j, d\theta_y^j\} \]

and the corresponding element resistances are

\[ dR_e = \{dN, dM_x^i, dM_y^i, dM_x^j, dM_y^j\}. \]

The element tangent stiffness matrix, relating these two vectors is formed by inverting the element tangent flexibility matrix. The element tangent flexibility matrix, \(F_{et}\), is evaluated by an integration of tangent slice flexibilities, under the assumption of longitudinal linear flexibility distribution. Thus,

\[
F_{et} = \int_0^L b(z)f_s(z)b(z)dz
\]

where

\[
b(z) = \begin{bmatrix}
1 & 0 & 0 & 0 & 0 \\
\frac{z}{L} - 1 & \frac{z}{L} & 0 & 0 & 0 \\
0 & 0 & \frac{z}{L} - 1 & \frac{z}{L} & 0
\end{bmatrix}
\]

is the equilibrium transformation matrix of a simply supported member, following the principles of virtual work.

The state determination process is based on the flexibility approach to the formulation of variable element shape functions, outlined in [Mahasuverachai 82]. Once the element nodal displacements \(r_e\) to be imposed on the element are determined, the following steps are performed:

- the current flexibility distribution is used to compute the variable shape function matrix

\[ a(z) = f_s(z)b(z)F_e^{-1}; \]
Chapter 2. Verification of the Fiber Element

- the deformations of end slices are computed as
  \[ dr_s(z) = a(z)dr_e; \]
- the resistances of the two end slices, \( dR_s \), are computed using a modified event-to-event step advancement;
- the required resistances of the remaining interior sections are computed using the equilibrium equations for a, now statically determined, element;
- the deformations of the interior slices are updated according to the imposed equilibrium slice resistances;
- the flexibilities of slices and the element flexibility are updated.

Finally, the element resistances \( R_e \) are returned to the host analysis program. Details of the elaborate equilibrium iteration strategies are presented in [Zeris 86].

This finite element model formulation permits analysis of members under large deformations that result in localized disintegration of members and cause softening mechanical properties. Most of the earlier analytical models have not been able to treat these conditions in a realistic manner. The numerical stability issues pertaining to the softening behavior of the member are treated in [Zeris 88a, Zeris 88b].

2.1.2 Fiber Spring-Joint Element

The fiber spring-joint element models the beam-column connection of infinitesimal thickness (Figure 2.2). It is developed by generalizing the concepts of four and five spring element joints [Lai 84]. The spring idealization is refined by adopting the fiber model, analogous to the slice idealization used for the fiber beam-column element. This enables modeling of arbitrary shaped and arbitrary oriented cross-sections.

![Figure 2.2: Fiber spring-joint finite element.](image)
Chapter 2. Verification of the Fiber Element

The spring-joint element has three kinematic degrees of freedom

$$r = \{\epsilon, \psi_x, \psi_y\},$$

and the corresponding joint resistances

$$R = \{N, M_x, M_y\}.$$

The fibers of the spring-joint element can have only axial deformation. Under the assumption of plane sections remaining plane, the kinematic transformations and the stiffness matrix formulation are the same as for a slice of a beam-column element. Since the stiffness matrix is readily available and corresponds directly to the external degrees of freedom, there is no need for additional transformations.

Material models currently available are the ones implemented for the beam-column element. In determining spring properties, uniform bond resistance is generally assumed over the development length of reinforcement bars, implying a linear steel stress distribution in the joint. The same slice equilibrium iteration strategies used in the beam-column element are implemented here.

The spring-joint element is implemented within the framework of ANSR-I analysis program. More details about the element can be found in [Zeris 86].

2.2 Description of Experiments

The experimental investigation used for a verification study in this project was conducted by S.S. Low at under the supervision of professor J. P. Moehle at the Department of Civil Engineering, University of California, Berkeley. The goal of the investigation was to observe the behavior of a column under different multi-axial load histories and assess the adequacy of the current design methods.

The experiments provide an excellent source of test data for the verification of the analytical reinforced concrete column models. This was the first time that a set of concrete columns with a rectangular cross-section was subjected to a variety of lateral and axial load histories in a well controlled experiment. This is in contrast to a great majority of previous experiments where only circular or square fully symmetric cross-sections have been considered.

The test program featured five nominally identical reinforced concrete column specimens (Figures 2.3 – 2.5). The columns were designed as quarter-scale models of a typical column and satisfy the major requirements of current codes for design of lateral load resisting columns in regions of high seismic risk. The average compressive strength of concrete was approximately 5000 psi (34.48 MPa). The type of reinforcement used was Grade 60 (414/620 MPa). Additional data on the characteristics of the materials used is presented in [Low 87].

The column specimens were tested as cantilevers projecting from stiff foundation blocks. The experimental setup is shown in Figures 2.6 and 2.7. The hydraulic jacks are positioned to enable the application of different combinations of lateral and axial loads. The instrumentation of the specimens was designed to provide direct measurements of the column tip displacement and the rotation of the specimen base cross-section.

The specimens were tested for three lateral load patterns, namely:

- uniaxial bending about the weaker cross-section axis (Figures 2.8);
- diagonal biaxial bending (Figures 2.9);
- clover-leaf lateral bending load (Figures 2.10);

and and two compressive axial load patterns:
Chapter 2. Verification of the Fiber Element

- constant axial load of 10 kips (44.48 kN);
- linearly varying axial load (Figure 2.11).

The experiment was controlled by load magnitude during the initial small displacement cycles. The large displacement cycles were performed under displacement control.

The specimens are differentiated according to the imposed loading pattern, as follows:

specimen 1: uniaxial bending about the weaker axis under a constant compressive axial load;
specimen 2: diagonal biaxial bending under a constant compressive axial load;
specimen 3: clover-leaf lateral bending load pattern under a constant compressive axial load;
specimen 4: diagonal biaxial bending under a linearly varying axial load;
specimen 5: clover-leaf lateral bending load pattern under a linearly varying axial load.

An interpretation of the recorded data and visual observations of the specimen behavior are presented in [Low 87]. The verification of the fiber model will be based only on the moment/displacement relations of all specimens. This data was chosen for comparison with the analytical fiber model because the measurements of the base cross-section rotation and the observed crack pattern show that the behavior of the entire column was predominantly governed by the rotation of the base cross-section and its load/deformation properties.
Figure 2.4: Specimen elevation: X direction (1 in. = 2.54 cm).
Figure 2.5: Specimen elevation: Y direction (1 in. = 2.54 cm).
Figure 2.6: Experimental setup: elevation (1 in. = 2.54 cm)

Figure 2.7: Experimental setup: top view (1 in. = 2.54 cm)
Chapter 2. Verification of the Fiber Element

Figure 2.8: Lateral displacement (inches) applied to achieve uniaxial bending of the column about the weaker cross-section axis.

Figure 2.9: Lateral displacement (inches) applied to achieve diagonal biaxial bending of the column.
Chapter 2. Verification of the Fiber Element

Figure 2.10: Lateral displacement (inches) applied to achieve clover-leaf biaxial bending load pattern.

Figure 2.11: Linearly varying axial load (1 in. = 2.54 cm; 1 kip = 4.448 kN).
Chapter 2. Verification of the Fiber Element

2.3 Element Verification

2.3.1 Introduction

To assess the accuracy and reliability of the fiber column and joint-spring elements, a series of analytical studies were performed for Specimens 1 through 5 tested by Low and Moehle. It was found that very good correlations could be obtained. However, to achieve satisfactory agreement of experimental and analytical results, the joint-spring element was necessary to account for the large contributions to lateral displacement of fixed end rotations associated with bar slip.

2.3.2 Modeling

The test specimens were modeled for these verification studies as follows:

Section distribution along member. Each specimen is subdivided into a number of sections along its length according to our need. The distribution of sections must take into account the damage and cracking pattern expected. The joint-spring element is included at the base of the column. The section distribution employed in the verification studies is shown in Figure 2.12.

![Figure 2.12: Section distribution along specimen.](image)

Fiber distribution of cross section. The fiber distribution shown in Figure 2.13 is used for the verifications. A single fiber is employed for each steel bar. The confined concrete is represented by a 6x8 grid of fibers. Unconfined concrete in the cover is represented by four groups of fibers. Each of these groups of fibers consists of 10 fibers laid out in a 2x5 grid. The fiber distribution for joint-spring element is similar to that shown in Figure 2.13, but concrete fibers are positioned in an 8x10 grid spanning the entire cross section.
Concrete material model for fiber column element. Two different material models are used for confined and unconfined concrete. They are shown in the Figure 2.14 and are based on reported test results [Low 87].

Steel material model for fiber column element. The Menegotto-Pinto model [Menegotto 77] is used for the steel fiber of the fiber column element. This model computationally attractive since the stress is explicitly defined in terms of the strain according to the following equation:

\[
\bar{\sigma} = \frac{(1 - b)\varepsilon}{(1 - \varepsilon/R)R} + b\bar{\varepsilon} 
\]  

(2.1)

denoting \( \bar{\sigma} = \sigma/\sigma_y, \bar{\varepsilon} = \varepsilon/\varepsilon_y \) where \( \varepsilon_y \) is the yield strain, \( \sigma_y \) is the yield stress, \( b \) the ratio of hardening to elastic modulus and \( R \) the exponent controlling the rate of nonlinearity (Figure 2.15).

Steel material model for joint-spring element. Due to the complex unknown nature of the rebar slippage of the R/C beam-column joints, a simple idealization is currently used. It is based on work by [Lai 84].

Concrete material model for joint-spring element. A elastic perfectly plastic material model is employed to represent the concrete spring of joint-spring element. The full concrete cylinder strength was used to estimate its yield capacity. Unloading under the initial modulus and cracking at zero force are assumed.

Assuming the steel stress distribution in the anchorage zone varies linearly along the development length of bar, the concrete bond stress is constant, the initial effective stiffness of the bar at
the joint interface can be written as:

\[ K_{se} = (2P_{sy}E_s)/(L_dF_y) \] (2.2)

\[ K_{sy} = 0 \] (2.3)

in which \( K_{se} \) = the initial elastic stiffness; \( E_s \) = modulus of elasticity of steel; \( L_d \) = development length for slippage; \( P_{sy} \) = force at yield for the effective steel spring; \( F_y \) = specified yield strength of reinforcement; \( K_{sy} \) = post yielding stiffness. A bilinear model is employed to represent this steel spring. Empirical adjustment factors \( M \) and \( b \) are introduced in this model to better match experimental data:

\[ K'_{se} = MK_{se} \] (2.4)

\[ K'_{sy} = bK'_{se} \] (2.5)

Figure 2.16 shows the models employed in the verification studies.

2.3.3 Comparison of Experimental and Analytical Results

The most complex tests were performed on Specimens 4 and 5 in which the axial load varied at the same time that lateral displacements were imposed. These specimens posed a severe test for the elements. The results plotted in Figures 2.17 and 2.18 show very good agreement between the analytical and experimental results. Experimental results are shown as dotted lines while solid lines represent computed responses. The computed results exhibit the complex hysteretic loop shapes corresponding to two different bi-lateral displacement histories and capture the influence of varying axial load.

It was found that the results were sensitive to various modeling assumptions. A parametric study was carried out to assess these sensitivities.
2.4 Parametric Studies

A number of analyses were performed to identify the sensitivity of computed responses to variations in modeling. Parameters considered include:

- Material models
- Number and distribution of sections
- Number and distribution of fibers across section.
- Inclusion/exclusion of joint-spring elements.

Analyses were performed for all specimens, so that the effect of different displacement/load history was also considered in the studies. Since cases without the joint-spring element attached at the base of the column would be expected to concentrate more damage within the column, such cases were more critical for assessing the sensitivity of the column modeling parameter. Thus, for the results shown below, a fixed column base is assumed for the analyses of material and geometric modeling alternatives. The effect of the joint-spring model is examined in Section 2.4.5.

2.4.1 Modeling of Steel Properties

For cyclic loading conditions it has been shown [Kaba 83] that material must be modeled properly in order to capture the inelastic cyclic behavior of reinforced concrete columns. Kaba showed, however, that analytical results in the inelastic range were insensitive to various concrete models as long as the initial modulus of elasticity and strength were represented properly. Predicted results were found to be very sensitive to steel modeling.

Because of computational considerations it would be desirable to minimize the number of
changes in stiffness on the material. For steel this would encourage the use of bi-linear, elasto-plastic models. However, a model capable of capturing Baushinger effects would be more realistic. To assess the effect these two steel models have on computed results a study was performed on Specimen 1. This specimen was discretized longitudinally and transversely as described in Section 2.3.2. Concrete properties were also taken identical to the previous verification examples. However, the base of the column was fixed. In one set of analyses the previous Menegotto-Pinto type was again used and in the other a bilinear model was employed. The initial modulus of the steel is the same for both steel models, and the strain hardening modulus was taken as 5% of the initial value.

The results are shown in Figure 2.19. It is clear from these figures that the bilinear results have the larger discrepancy. Errors are most significant in the representation of the stiffness degradation that occurs on reloading following a cycle of inelastic deformation oriented in the opposite direction. Thus, the Menegotto-Pinto type model produces more realistic results. The analyses using the Menegotto-Pinto model require approximately 35% more computational effort than do the analyses using the simpler bilinear model.

It can be noted that neither of the correlations shown in Figure 2.19 are as good as shown in Figures 2.17 and 2.18 for Specimens 4 and 5. This is attributable to the absence of the joint-spring element in these models of Specimen 1.

2.4.2 Number and Distribution of Sections

To achieve a realistic distribution of curvature (and damage) along the members, a sufficient number of sections must be incorporated into the model. If too few sections are used, inaccurate results would be expected; if too many are used, the analysis would be computationally prohibitive.

To assess this, the Specimen 1 model with the Menegotto-Pinto steel representation is again considered. Three alternative distributions of sections along the member are considered. In the first, only two sections are considered; one located at each end. In the second the first segment considered in the standard analysis is subdivided in half to get a better distribution of curvature in the plastic hinge zone. This gives 5 sections. In the third alternative, the end region is divided into
Chapter 2. Verification of the Fiber Element

Figure 2.17: Moment[kip-in]/displacement[inch] diagram for Specimen 4: (a) $M_x/D_y$ relationship; (b) $M_y/D_x$ relationship.

Figure 2.18: Moment[kip-in]/displacement[inch] diagram for Specimen 5: (a) $M_x/D_y$ relationship; (b) $M_y/D_x$ relationship.

Figure 2.19: Effects of steel modeling on Specimen 1: Fixed base condition: (a) Menegotto-Pinto model; (b) Bilinear model.
six 1-inch long segments for a total of 9 sections.

The results of these analyses are compared in Figure 2.20. The same basic trends are seen. However, it should be noticed that the case with only two sections achieves the poorest correlation and might not be expected to work as well under more complex boundary condition (i.e., double curvature along the column).

The results for 9 sections show some changes in shape not seen in the other cases. It is believed that this is a result of a tendency of models with close section spacing to concentrate damage in one segment when the cross sectional mesh is not correspondingly refined.

It appears that the standard distribution of 4 sections adequately produce the behavior of the specimen. The distribution of sections should vary as the expected damage along the member. In this case the damage concentrated in the bottom 1/3rd of the column. It appears that a section located at a distance equal to the width of the member away from the end of member may be sufficient to represent basic behavior.

Figure 2.20: Effect of number of sections on Specimen 1: Fixed base condition: (a) 2 sections; (b) 4 sections; (c) 5 sections; (d) 9 sections.

2.4.3 Number and Distribution of Fibers In Cross-Section

For computational efficiency it is desirable to minimize the number fibers considered. Typically, each steel rebar is represented by a fiber. The discretization of the concrete must be selected
considering a number of factors. First, the fiber idealized section should have similar elastic properties. If too few fibers are used, the effective moment of inertia of the idealized section will differ appreciably from that of the actual section. To see this, a rectangular section is considered. The moment of inertia, \( I \), is \( bh^3/12 \) where \( b \) and \( h \) are the width and height of the section, respectively. Considering the section discretized in the vertical direction into \( n \) equally spaced fiber (where \( n \) is even), the moment of inertia of the idealized section, \( I' \), is

\[
I' = (1 - \frac{1}{n^2})I
\]

(2.6)

Considering various \( n \) value, one has:

<table>
<thead>
<tr>
<th>( n )</th>
<th>( I'/I )</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.75</td>
</tr>
<tr>
<td>4</td>
<td>0.94</td>
</tr>
<tr>
<td>6</td>
<td>0.97</td>
</tr>
<tr>
<td>10</td>
<td>0.99</td>
</tr>
</tbody>
</table>

A second consideration is that the strains in the critical concrete fibers be realistically represented. Since the centroid of the critical fiber is not at the top of the section, strains related to onset of spalling and so on may not be accurately captured by the model. In this case the strain at the top fiber, \( \varepsilon_{max} \), is related to the strain at the top of the section, \( \varepsilon_{max} \), when pure moment is considered by:

\[
\varepsilon_{max}' = \left[ \frac{n}{(n + 1)} \right] \varepsilon_{max}
\]

(2.7)

For various values of \( n \), one obtains:

<table>
<thead>
<tr>
<th>( n )</th>
<th>( \varepsilon_{max}' / \varepsilon_{max} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.67</td>
</tr>
<tr>
<td>4</td>
<td>0.80</td>
</tr>
<tr>
<td>6</td>
<td>0.86</td>
</tr>
<tr>
<td>10</td>
<td>0.91</td>
</tr>
</tbody>
</table>

The error in strains is greater than in moments of inertia. This problem is accentuated once a section cracks and the neutral axis moves much closer to the compression face.

This points out another consideration is selecting mesh refinement. It would be expected that because of the probable location of the neutral axis, fibers should be concentrated near the perimeter of the section. This will best represent the concrete properties (more fibers will be used for the portion of the section that is in compression). If spalling is expected, more fibers should also be provided near the perimeter of the confined concrete.

To assess these considerations the standard representation of Specimen 1 is compared to 3 other mesh distributions. In the first the density of fibers is increased to a 9x10 grid in the core and the same cover grid is used (5x2). In the second alternative the core is modeled by a 5x6 grid of fibers, and the 5x2 cover grid is maintained. In the third alternative, both cover and core grids are simplified. The core is reduced to a 3 by 3 grid and the cover is represented as a 5x1 grid for this third alternative.

The results are shown in Figure 2.21. As expected the finer mesh produces somewhat better correlation. The correlation deteriorates as the grid becomes coarser. Surprisingly, the simplest grid is still able to capture the basic shape of the hysteretic loops rather well.
Chapter 2. Verification of the Fiber Element

Figure 2.21: Effect of number and distribution of fibers for Specimen 1: fixed base case. (a) 9x10 grid for center; (b) 6x8 grid for center; (c) 5x6 grid for center; (d) 3x3 grid for center.
2.4.4 Inclusion of Spring-Joint Model

In the cases presented so far correlations have been reasonably good, but significant discrepancies still exist. To assess the causes of these discrepancies, additional analyses were performed in which the joint-spring element is included at the base of the test specimens. Previous research on fixed end rotations (associated with bar slip) indicate that nearly half of the inelastic deformation in a member may be contributed by this source [Mahin 88]. The joint-spring element described in Section 2.1 allows these deformations to be approximated. The model used for these springs is based on one developed by Lai [Lai 84].

As indicated above the spring stiffness suggested by Lai may be incorrect and an empirical factor M is introduced. Filippou [Filippou 83] has shown that bond stress are not uniform over the length of anchorage and that bond stress in the unconfined concrete near the base of a column is sharply reduced after several cycles of inelastic behavior. Assuming that the unconfined concrete in this case is about \( L_d/3 \) one would obtain:

\[
K_{se}' = \frac{(P_{sy} E_s)}{((L_d F_y/2) + (L_d F_y/3))}
\]

which is

\[
K_{se}' = 0.6 K_{se}
\]

Thus the empirical coefficient, M, introduced in Equation 2.2 might be expected to have values near 60%.

To assess this, Specimen 1 is reanalyzed with the joint-spring included. The results in Figure 2.22 show marked improvement in the correlation. In this analysis, the parameter M was taken to be 42% based on several iterations.

![Figure 2.22: Effect of joint-spring on Specimen 1 behavior: (a) Fiber element; (b) Fiber element & joint spring element.](image)

To look at cases with more severe bidirectional loading, Specimens 2 and 3 were investigated. Results for these specimens are shown in Figure 2.23 and 2.24. Again, good correlation with experimental results is seen. In these cases M is taken 42%.

To assess the effect of the parameter M as well as of other joint-spring parameters, several analyses were performed. These were carried out considering Specimens 4 and 5. In addition to varying M, the strain hardening ratio for the steel was also varied in a few cases:
Figure 2.23: Correlation with fiber model supported on joint-spring element. Specimens 2 and 3. M=0.42: (a) Specimen 2: $M_x/D_y$; (b) Specimen 2: $M_y/D_z$; (c) Specimen 3: $M_x/D_y$; (d) Specimen 3: $M_y/D_z$. 
Chapter 2. Verification of the Fiber Element

Figure 2.24: Correlation with fiber model supported on joint-spring element. Specimens 4 and 5. M=0.42 (a) Specimen 4: $M_x/D_y$; (b) Specimen 4: $M_y/D_z$; (c) Specimen 5: $M_x/D_y$; (d) Specimen 5: $M_y/D_z$. 
Results for Specimen 4 which was subjected to skewed lateral loading and variable axial load are shown in Figure 2.25. Results for Specimen 5 which was subjected to the clover-leaf lateral displacement pattern and variable axial load are shown in Figures 2.26 and 2.27.

From these figures it is clear that values of M between 0.4 and 0.6 produce good results for large inelastic cycles. At smaller displacement levels, the higher value of M achieves better correlation. It is believed for general analysis values of M varying between 0.5 and 0.6 may be preferable.

Results obtained in this study are in better agreement with experimental data than results obtained using simplified multi-spring models [Lai 84]. For example, in a recent study of these specimens using a 4-spring model [Jing 90], good results are reported. However, the accuracy is not as good as achieved using the fiber model shown in Figure 2.28. Moreover, the multiple spring model does not treat variation of axial load, spalling of concrete and irregular shaped sections as well.

### 2.4.5 Numerical Sensitivity of Results

To assess numerical respects of the model, another set of analyses was performed. In this case, multiple elements were used to model the distribution of sections located along the length of the member. The fiber model incorporates internal degrees of freedom that automatically incorporate the distribution of damage along the element. Other types of models do not have this power and refined modeling of the plastic hinge region can be done only by using several elements over the length of the member. This results in increased computational effort at the global level and possible difficulties in achieving a converged stable solution when when the host program must eliminate equilibrium errors that may develop at nodes during a step.

The fiber element considered in this study reduce these global problems by a variety of techniques. The nature of the mixed formulation is that it inherently eliminate equilibrium errors entirely at internal sections.

A series of analyses was carried out on Specimen 1. In this series the standard model was used and a joint-spring (M=0.42) was employed. The standard model considers a single element with 4 sections along the length (2 at the ends and two interior sections). For comparison, three elements, each having 2 sections (one at each end), were used. An analysis with four elements was used as well. As would expected little difference in results is obtained (Figure 2.29) for the well conditioned behavior exhibited by the column analyzed.

Differences are attributable to the need for the global iterative solution strategy to resolve nodal equilibrium errors within the plastic hinge for the cases with sections only at element ends. In the case of severe spalling leading to softening of the sections close to the base of the column, the global program would not be expected to be able to maintain numerical stability. The refined fiber element, on the other hand, was formulated especially to handle such severe loading conditions. Moreover, it does so with less computational effort for a given level of member refinement.
Figure 2.25: Calculating results of Specimen 4 using different $M$ and $b$: (a) $M_x/D_y$, $M=0.80$, $b=0.05$; (b) $M_y/D_x$, $M=0.80$, $b=0.05$; (c) $M_x/D_y$, $M=0.60$, $b=0.03$; (d) $M_y/D_x$, $M=0.60$, $b=0.03$; (e) $M_x/D_y$, $M=0.42$, $b=0.05$; (f) $M_y/D_x$, $M=0.42$, $b=0.05$. 
Figure 2.26: Results of Specimen 5 using different $M$ and $b$: (a) $M_x/D_y$, $M=0.80$, $b=0.05$; (b) $M_y/D_x$, $M=0.80$, $b=0.05$; (c) $M_x/D_y$, $M=0.80$, $b=0.02$; (d) $M_y/D_x$, $M=0.80$, $b=0.02$. 
Figure 2.27: Results of Specimen 5 using different $M$ and $b$: (e) $M_y/D_y$, $M=0.60$, $b=0.05$; (f) $M_y/D_x$, $M=0.60$, $b=0.05$; (g) $M_x/D_y$, $M=0.42$, $b=0.05$; (h) $M_y/D_x$, $M=0.42$, $b=0.05$. 

Chapter 2. Verification of the Fiber Element
Figure 2.28: Comparison with 4 spring model results: (a) Specimen 4: fiber model; (b) Specimen 5: fiber model; (c) Specimen 4: 4 spring model; (d) Specimen 5: 4 spring model [Jing 90].
Figure 2.29: Calculating results of multiple elements: (a) 1 element, 4 sections; (b) 3 elements, 2 sections for each element; (c) 4 elements, 2 sections for each element.
Chapter 2. Verification of the Fiber Element

2.5 Conclusions and Recommendations

The refined fiber column model and the associated joint-spring element were able to simulate the inelastic behavior of rectangular reinforced column subjected to complex patterns of bidirectional lateral load as well as to simultaneously varying axial loads. A minimum of assumptions were required to perform these analyses and results could be interpreted in terms of local strains and damage to the member. The behavior of the sections were complex and simplified stiffness degrading models would not have been able to predict response. No numerical instabilities were detected and reasonable computational efforts were required. Sensitivities studies identified several parameters that were important.

- Baushinger effects in steel were found (as expected) to have an important influence on computed response.
- In many cases only a few (possibly two) internal sections need be located along an element. The number of sections and their distribution must be based on considerations of expected inelastic curvature (damage) distributions. The linear variation of section flexibility assumed by the element between sections minimizes the number of sections that need to be considered.
- In many cases only a limited number of fibers need be considered in modeling sections. Fiber must be distributed to achieve realistic bending stiffnesses and strain distributions. For inelastic action fibers should be concentrated near the parameter of the section.
- In was found to be very important to consider the fixed end rotation at the base of the column due to rebar pullout. The simplified joint-spring element was able to provide realistic results provided an empirical parameter was introduced to reduce the theoretical stiffness of the springs.

The fiber model was found to be reliable and versatile. Computational effort was consistent with the level of refinement but less than expected for similar types of refined models. Additional future work remains to be done to apply the element to the analysis of complete bridge and building structures subjected to static and dynamic loading. Some initial work in this topic is reported elsewhere [Zeris 90b, Zeris 90a]. Improvements in the element are desirable especially with regards to facilitating input of data and interpretation of results. The simple hysteretic models used for representing the joint-springs should be improved to achieve even better correlations of experimental and analytical responses.
Chapter 3

AfC—The Section Analysis Program

A major task undertaken in this CUREe-Kajima project is the development of a computer program for non-linear analysis of reinforced concrete sections AfC. The main purpose of the program is to provide a powerful analysis tool, flexible enough to satisfy the needs of researchers, designers and students. In addition, this program is a platform for testing prototype solutions for user interface and data input problems that arise within the scope of the CUREe-Kajima unified computing environment.

The analytical procedures used in AfC are based on the displacement formulation for the phenomenological fiber cross-section model. The same theoretical foundations were used for developing two cross-section analysis programs, UNCOLA [Kaba 83, Kaba 84] and BICOLA [Zeris 86, Zeris 87]. Both programs are capable of performing non-linear analysis of an arbitrary shaped reinforced concrete cross-section under uniaxial (UNCOLA) and multi-axial (BICOLA) loading. The fiber material models incorporated in these programs allow for the application of non-proportional, path dependent and cyclic loading histories. The use of UNCOLA and BICOLA demonstrated the potentials of this kind of software in research and design. However, the shortcomings of the existing user interfaces, lack of portability and occasional problems with the iterative solution strategy were obvious.

The goal of this part of the CUREe-Kajima project is to develop a new cross-section analysis program—AfC. Particular emphasis is placed on achieving the ease of use by designing the new application in a multiple window environment and inclusion of various pre- and post-processing routines. The analytical capabilities of UNCOLA and BICOLA are preserved and enhanced by improving the iteration strategy and consistency in material model definitions. Finally, the object-oriented development environment provides for a high level of portability across commonly available computing platforms.

The first section of this chapter addresses the functional specification of AfC. Section 3.2 presents the design and the modular structure of AfC. Section 3.3 discusses the current state of the development effort and the immediate future goals. Finally, the work on AfC is summarized in Section 3.4.

3.1 Functional Specification

AfC is designed to perform the non-linear analysis of reinforced concrete cross-sections subject to arbitrary axial force and bending moment load histories. The primary design goal is to allow real-time interaction between the user and the application. Therefore, the application is made to perform fast and reliable analysis and provide the flexibility needed for building the user interface. The second design goal is to make the application portable across different computing platforms.
Portability is achieved through careful design of data structures and use of the object-oriented programming paradigm.

AeS is aimed at three different groups of users: researchers, designers and students. The cross-section analysis capabilities required by all three groups is the same. However, the tasks where cross-section analysis is required for each of the groups have different complexity levels and require very different data manipulation and visualization functionality. In order to satisfy these needs, AeS is composed of two independent modules:

- the analysis module and
- the user interface module.

The analysis module provides non-linear concrete and steel material models and a non-linear iteration strategy that is capable of dealing with non-proportional loading applied using mixed force and displacement control. The user interface module enables real-time user interaction with the analysis module.

### 3.1.1 Cross-Section Analysis

The analysis of the reinforced concrete cross-section is based on the event-to-event non-linear iteration strategy, the deformation approach and the fiber discretization method.

A cross-section is modeled as a three dimensional continuum with an infinitesimal thickness $dz$ (Figure 3.1). This continuum consists of an infinite number of differential volumes. The global cross-section properties are obtained by integrating the actions and deformations of individual differential volumes. For linear or simple non-linear material behavior it is possible to formulate the exact equations and integrate them in closed form to obtain the exact cross-section properties. In reality, mathematical models of material behavior are prohibitively complex. Therefore, numerical integration based on the interpolated properties of a finite (usually small) number of monitored differential volumes is the method of choice for this kind of analysis.

A natural method for interpolation of material properties is to divide the cross-section into a number of finite volumes—fibers—and monitor the material properties of differential volumes found at the centroids of fibers, as shown in Figure 3.2. Thus, it is assumed that material properties do not vary significantly within the volume of the fiber. In addition, it is assumed that the only significant geometric property of a fiber is its area.

AeS assumes that a cross-section is subject only to axial and bending loads. Therefore, the only deformation a fiber is exposed to is the axial strain. Fundamental deformation modes of a cross-section are shown in Figure 3.3. The relation between the cross-section axial deformation and curvature and the fiber axial strain is established by assuming that plane sections remain plane. This relations is expressed as:

$$
\epsilon_i = \left[ 1 - y_i x_i \right] \begin{bmatrix} \epsilon \\ \psi_x \\ \psi_y \end{bmatrix}
$$

(3.1)

where:

- $\epsilon_i$ is the fiber strain (positive in tension);
- $x_i$ and $y_i$ are the coordinates of the fiber centroid with respect to the cross-section principal axis;
- $\epsilon$ is the cross-section axial extension (positive in tension);
- $\psi_x$ is the cross-section rotation about $x$ axis;
- $\psi_y$ is the cross-section rotation about $y$ axis.
Chapter 3. ACS—the Section Analysis Program

The response of a fiber to a given strain is defined by the fiber material, i.e. the uniaxial constitutive relationship assumed for that material (Figure 3.4). In most cases, these constitutive relationships are empirical. They consist of a set of states where a particular material can be found during the loading process and a set of rules that govern transitions from one state to another. The constitutive material model must behave “sensibly”. Namely, a material model must be

- **physically viable**, providing physically sound behavior (i.e. a fiber must not absorb negative energy during cycling);
- **logically complete**, insuring that at each state the rules of the model are sufficient to uniquely determine the next state;
- **realistic**, making the behavior of the model as close to the experimentally observed material behavior as possible.

The fiber property of utmost importance for general non-linear analysis is the tangent modulus, defined (for the i-th fiber) by

$$d\sigma_i = E_{ti} \, d\epsilon_i. $$

(3.2)

Based on the above relations, the force-deformation relationship at the cross-section level is given by

$$\begin{bmatrix} 
\frac{dP}{dM_x} \\
\frac{dM_x}{dM_y} \\
\frac{dM_y}
\end{bmatrix} = K_t \begin{bmatrix} 
\frac{d\epsilon}{d\psi_x} \\
\frac{d\psi_x}{d\psi_y}
\end{bmatrix} $$

(3.3)
where $K_t$ is the cross-section tangent stiffness matrix, defined as

$$
K_t = \sum_i \begin{bmatrix}
1 & -y_i \\
-x_i & 
\end{bmatrix} E_i A_i [1 - y_i x_i] 
$$

(3.4)

and

$P$ is the cross-section axial force;

$M_x$ is the cross-section moment about $x$-axis;

$M_y$ is the cross-section moment about $y$-axis;

$A_i$ is the area of the $i$-th fiber.

The cross-section deformation state is defined by the triplet

$$
r = \{\epsilon, \psi_x, \psi_y\}
$$

and the cross-section force state is defined as

$$
R = \{P, M_x, M_y\}.
$$

The goal of the analysis procedure in AcS is to find all cross-section deformation and force states occurring during any given load history, assuming non-linear hysteretic material behavior. A load history is specified as a series of load steps. A load step is defined as a triplet of generalized load component increments along the axial and two rotational cross-section degrees of freedom. A generalized load component increment can be either a force or a displacement.
Figure 3.3: Fundamental deformation modes of a cross-section: a) axial extension; b) rotation about $x$ axis; c) rotation about $y$ axis.

![Deformation Modes](image)

Figure 3.4: Possible fiber material models: a) steel; b) concrete.

<table>
<thead>
<tr>
<th>generalized load:</th>
<th>axial</th>
<th>$x$ rotation</th>
<th>$y$ rotation</th>
</tr>
</thead>
<tbody>
<tr>
<td>force</td>
<td>$\Delta P$</td>
<td>$\Delta M_x$</td>
<td>$\Delta M_y$</td>
</tr>
<tr>
<td>displacement</td>
<td>$\Delta \epsilon$</td>
<td>$\Delta \psi_x$</td>
<td>$\Delta \psi_y$</td>
</tr>
</tbody>
</table>

It is obvious that the cross-section tangent stiffness matrix can be formed at any given cross-section deformation state. Thus, it possible to solve two important problems.

The first problem concerns the application of mixed load and deformation load steps. Given the tangent stiffness matrix for a particular deformation state and a load step the equations

$$K_{\Delta r} = R$$

can be rearranged and solved for the unknown increments of deformation state variables.

Once the load step is converted, its has to be applied to the cross-section. The non-linear behavior of the fiber material models creates the second problem. Fortunately, this non-linear problem is commonly found in engineering practice and there are several iterative strategies for solving it [Simons 82]. The strategy implemented in AcS is the modified event-to-event strategy. This strategy is chosen because it is conceptually simple and reliable. It is a natural way of following the complete load-displacement path of the analyzed cross-section because it closely simulates the processes that take place in the actual cross-section.
3.1.2 Functional Summary

Although seemingly simple, the problem of non-linear cross-section analysis is extremely data intensive. The definition of the cross-section geometry and material characteristics requires considerable data. Long loading history definitions make interactive use of the program tedious and error-prone. Finally, large amounts of data produced during the analysis makes monitoring and post-processing prohibitively slow and difficult. The experience in using BICOLA strongly suggests that these problems can severely degrade the performance of an otherwise adequate program.

Data required for an analysis of a cross-section can be classified as:
- cross-section geometry data (fiber centroid position and fiber area data);
- material model data;
- load history data (specification of a series of load steps to be applied to the cross-section).

Results of cross-section analysis can be divided into the following two groups:
- global cross-section response data (the history of cross-section force and deformation states);
- local individual fiber response (the history of fiber stress-strain states).

Actions performed by the user during an analysis of a cross-section are:
- definition of the cross-section geometry;
- definition of the cross-section materials;
- definition of the analysis tasks (either user specified load history or an ultimate load locus analysis using the predefined load patterns);
- execution of the chosen task, i.e. analysis the cross-section under a chosen load history;
- monitoring, analysis and manipulation of results.

ACeS provides the data structures for storing the analysis input and output data and methods for performing the required actions. The principal requirement for the behavior of ACeS is that it is an interactive application, providing immediate response to user actions as well as easy and intuitive data manipulation. The routines of the analysis module should be fast, robust and very efficient in using the available data structures. Also, care is taken to properly design the data-handling routines in the user interface module.

The user interface module consists of routines that enable interactive, fast and robust access to data structures and methods provided in ACeS. The current standards of user interface design prescribe a user-friendly application environment. The routines should be robust (with respect to user errors) and provide for easy undoing of accidentally taken actions. On the structural side, ACeS is developed as a modular component of the overall CUREe-Kajima computing environment. It is easily expandable, allowing addition of new material models and iteration strategies, and portable across a variety of computing platforms. Also, ACeS user interface module is independent enough to serve as a front end for specifying geometric and material data for a general finite element program.

In order to satisfy these functional and structural specifications ACeS is designed using the object-oriented design paradigm. Data structures and functionality of the analysis module are implemented as a set of classes, discussed in some detail in Section 3.2. The user interface module is designed under the blanket assumption that the application will function in an event driven, window-oriented environment with a mouse as the primary interaction device. Therefore, a single main application window encapsulates the entire interaction between the user and ACeS. The user interacts with ACeS via a set of menus and pop-up dialog boxes. Problem definition data is supplied in files (using a special-purpose data definition language), through appropriate form-filling dialog boxes or by direct manipulation of graphic images. The results of the analysis can be viewed using graphs displayed in sub-windows or color-coded images of the analyzed cross-section and exported in plain ASCII
files, enabling easy processing by a variety of database and graphing software.

### 3.1.3 Development Schedule

The modular structure of A&s made it possible to develop the program in stages. The following items describe the phases of application development:

1. Design of data structures and functions related to material models in the analysis module. Formulation of several basic material models for testing the iteration strategy.

2. Implementation of data structures and functions related to cross-section discretization, stiffness formulation and non-linear iteration strategy in the analysis module.

3. Completion of the analysis module. Implementation of the minimal functionality in the user interface module, allowing problem definition and result analysis only through formatted ASCII files.

4. Addition of special-purpose language extensions for data definition in the user interface module. The language for specification of cross-section geometry, material model states and step-by-step load history are processed by command line parsers.

5. Implementation of graph objects for graphic presentation of analysis results. The capabilities of graph objects in this phase include only two-dimensional line-graph drawing.


7. Addition of an interactive, step-by-step cross-section loading driver, with an “undo” facility. This makes interactive, real-time cross-section analysis of “what-if” type possible.

8. Implementation of a cross-section geometry and fiber layout drawing capability. The purpose of this capability is two-fold: it provides the means of checking the definition of the cross-section geometry and options for color-coded real-time monitoring of fiber states.


10. Development of a material model library manager that includes material model definition, finite state machine code generation and dynamic library linking capabilities.

11. Development of an interactive graphics-oriented cross-section geometry editing facility.

Beyond level 6 A&s can be used as an analysis tool. The program provides adequate facilities for testing of the behavior of iteration strategy, new material models and the functionality of the user interface. Further development of the application hinges on the results of these performance tests. Interactive definition of cross-section geometry depends decisively on the resolution and accuracy of available display and pointing devices.

### 3.2 Design Description

An important concept in the design of A&s was the division of functionality and data structures into two independent modules, the analysis module and the user interface module. The process of
modularization is extended further to accommodate the functional requirements and the application development phases.

The interaction between the analysis and the user interface module can be designed using the Model-View-Controller programming concept. This concept is based on the separation of an application into three distinct layers, namely:

- the model, encapsulating the application's functionality;
- the view, defining a way the model is presented to the user;
- the controller, encapsulating the methods of user interaction with the model.

The key idea in the Model-View-Controller concept is the separation of an application into interchangeable layers so that how a user sees and interacts with an application is independent of the central structure of the application. This provides the flexibility of changing the layers of the application to accommodate different user perspectives of the model layer and porting to different computing platforms.

The structure of AfcS readily supports the MVC concept, as shown in Figure 3.5. The analysis module is organized as the model layer of the application, the user interface module presents the view layer of the application and the controller layer is provided by the command language parser and the window environment, accessed through a class library. Following the communication protocol specifications embedded into the MVC concept, AfcS analysis module is designed as a closed, self-contained entity. It interacts with the user interface module exclusively by answering model access and update messages. In order to do this, the classes that form the analysis module implement appropriate "callback" methods. The user interface module refers to these callback methods when it receives view protocol messages. These messages are generated by the user in the command parser. Once the program obtains the answer from the analysis module, the user interface module informs the user about the outcome using the controller messages.

![Figure 3.5: Communication links between modules in AfcS.](image-url)
3.2.1 Analysis Module

The analysis module consists a group of classes that provide the data structures required by the material model data and by the cross-section geometry data and implement the functionality of the analytical module.

Material State Buffer Class: this class implements an entity of a buffer for states an instance of a material model goes through during a loading sequence. A material state is defined as an ordered pair of current stress and strain values. This class is implemented as a list.

Material Model Class: this class implements an entity of a uniaxial material model with a linearized stress-strain constitutive relation. It combines data the local, fiber specific, material data and the global material description data, common to all fibers.

Some of the functions included in this class perform:

• reporting of material state (stress, strain, stiffness, failure);
• reporting of the material event factor;
• change in the direction of (displacement-driven) loading;
• application of a, possibly scaled, displacement step.

Fiber Class: this class implements an entity of a fiber, combining the fiber geometry data with an instance of its material model. Fiber geometry is defined by its area and position of its centroid with respect to the principle cross-section axes.

Cross-Section State Buffer Class: this class implements an entity of a buffer for states a cross-section goes through during a loading sequence. A cross-section state is defined as a set

\[ \{ P, M_x, M_y, \epsilon, \psi_x, \psi_y, \} \]

of current cross-section force and deformation values. This class is implemented as a list.

Cross-Section Class: this class implements an entity of a cross-section analytical model. This class is at the top of the analysis model class hierarchy. The data part of the cross-section model consists of:

• an ordered collection of fibers;
• an ordered collection of material global data containers;
• an instance of a Cross-Section State Buffer Class.

The functions provided in the cross-section model can be divided into two groups. The first group contains the callback methods that respond to messages form the user interaction module. This group includes methods for:

• initializing the cross-section from a formatted stream (both geometry and material data);
• reporting the cross-section state;

The second group implements the non-linear iterative solution strategy of the event-to-event type. It contains functions for:
Chapter 3. AcS—the Section Analysis Program

- cross-section stiffness computation;
- load step conversion to deformation-only form;
- global event factor evaluation;
- load step application and cross-section state determination;

At this phase of development AcS has:
- a generic elastic material model;
- a generic elastic-perfectly plastic material model;
- a generic bilinear material model;
- the modified Kent-Park concrete model [Kent 71], implemented using a 6-line linearized envelope;
- the Cook and Gerstle steel model with variable elastic unloading length and modulus [Cook 85]

3.2.2 User Interface Module

The user interface module is built upon the interaction mechanism and data structure classes provided in the window-system environment. It consists of the following classes (in development phase 6):

**Graph Class:** this class implements an entity of a two-dimensional line-graph object. The graphing data is obtained by reading a collection of points, making the direct use of material and cross-section state buffers possible. The object implements the basic functionality of generic line-graph drawing programs and a capability of automatic axes re-scaling if the new graphing data is out of the current range.

**Material Definition Language Parser:** this class implements the material definition language and provides a parser that converts a stream (file) written in the material definition language to a formatted stream readable by the Cross-Section Class constructor.

**Cross-Section Definition Language Parser:** this class implements the cross-section definition language and provides a parser that converts a stream (file) written in the cross-section definition language to a formatted stream (a list of fiber specifications) readable by the Cross-Section Class constructor.

A cross-section is generated as a set of geometric objects. An object is either a geometric primitive (a rectangle, triangle, quadrilateral, circle, pie or arc) or a group of geometric primitives. The fibers are generated from the mesh descriptions for each of the primitives. A description of the cross-section definition language and a detailed example are presented in the Appendix.

**Load Definition Language Parser:** this class implements the load definition language and provides a parser that converts a stream (file) written in the load definition language to a formatted stream of load steps readable by the Cross-Section View Class functions.

A loading history is defined as a sequence of load steps. In turn, a load sequence is composed of load steps or other load sequences. A load step defines a triplet of generalized load component increments, following the definition of page 36. Load definition language provides ways of scaling, repeating and grouping load
steps and enables easy formulation of complex load histories. A sample load history definition and a detailed description of the load definition language are presented in the Appendix.

Cross-Section View Class: this class implements an application's view layer. It is at the top of the class hierarchy in the user interface module. As such, it implements mechanisms for:

- interaction with the user, using menus and various types of dialog boxes;
- opening, parsing and connecting streams among different entities in the analysis module of AoS;
- directing result to various output devices, graph objects or ASCII files.

3.3 Status and Pending Tasks

Currently, we are proceeding with extensive testing of the application. Initial tests are performed with the simple elastic material model, to examine the overall behavior of the analysis module. Next, the event-to-event iteration strategy and the material model state transition process will be tested using the bilinear model. Following these tests, verification will be done using the modified Kent-Park concrete model and Cook-Gerstle steel model will be performed and compared against the S. S. Low and J. P. Moehle beam-column and Kajima beam experimental data.

3.4 Summary

A cross-section analysis program called AoS has been developed. The development process proceeded from the definition of the computing platform and functional specification, presented in the CUREe-Kajima Interim Report to the current implementation and testing phase. In its present form, AoS is a useful analysis tool. Furthermore, it is a good base for future development. One possible direction is the development of a cross-section analysis tool specialized for design or instructional use. Another is the incorporation of AoS into a more complex structural analysis application.
Chapter 4

Conclusion

The research performed to date has been concentrated on the verification of the fiber finite element model for the analysis of reinforced concrete columns and the developments of the program for the analysis of reinforced concrete cross-section, AcS. The verification study showed good agreement of the fiber model with the experimental results of a rectangular specimen under various types of biaxial bending and axial loads. The work on AcS resulted in the development of the application up to level 4.
References


References


Appendix A

Sample Problem Definition in ACcS

Problem definition consists of three phases (page 38):
- definition of the cross-section geometry;
- definition of the cross-section materials;
- definition of the analysis tasks (either a user specified load history or an ultimate load locus analysis).

This appendix presents a sample problem definition. The sample cross-section geometry is shown in Figure A.1. The sample loading consists of a variable axial force and a clover-leaf displacement pattern, shown in Figure A.2.

Figure A.1: Sample cross-section.
Appendix A. Sample Problem Definition in AcS

A.1 Cross-Section Geometry

A cross-section is a planar (two-dimensional) geometric object. A \textit{planar geometric object} is a group of planar objects and/or planar geometric primitives. A \textit{planar geometric primitive} is an atomic planar geometric object. \texttt{AcS} supports the following geometric primitives:

- quadrilateral;
- triangle;
- rectangle;
- arc;
- pie;
- circle;
- reinforcement bar.

Each planar geometric object has its own, local, coordinate system origin. A \textit{point} in the local coordinate system can be specified either in Cartesian coordinates, as

\[ x_y \]

using the underscore (\_) as a separator, or in polar coordinates, as

\[ r\phi \]
using the less-than sign (\(<\)) as a separator. The lengths are specified in drawing length units. The default length unit dimension is metric (meter), but the user can specify the desired length dimension using the \texttt{units} command. Internally, length units are converted to metric units to preserve consistency. In polar coordinates, angles are specified in degrees in either decimal or degrees-minutes-seconds form.

Geometric primitives are specified by parameters in the geometric primitive definition argument list. The parameters refer to the local coordinate system origin of each primitive. A primitive definition list consist of:

- a compulsory set of geometric parameters;
- a compulsory material specification;
- an optional primitive mesh specification.

Each primitive is specified with respect to its own local origin. This local origin is the \textit{reference point} of the geometric primitive. Following is a list of possible geometric primitive specifications. The symbol \texttt{pt} means that a point specification is required.

\textbf{Quadrilateral definition:} the \texttt{quad} command defines an arbitrary quadrilateral by specifying four points in the counter-clockwise direction (Figure A.3). If three points are specified, the first point is assumed to be the local origin.

\begin{verbatim}
quad( pt1, pt2, pt3, pt4, material, mesh)
quad( pt1, pt2, pt3, material, mesh)
\end{verbatim}

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{quadrilateral.png}
\caption{Quadrilateral primitive.}
\end{figure}

\textbf{Triangle definition:} the \texttt{triangle} command defines an arbitrary triangle by specifying three points in the counter-clockwise direction (Figure A.4). If two points are specified, the first point is assumed to be the local origin.

\begin{verbatim}
triangle( pt1, pt2, pt3, material, mesh)
triangle( pt1, pt2, material, mesh)
\end{verbatim}

A triangle is a special case of a quadrilateral primitive.

\textbf{Rectangle definition:} the \texttt{rect} command defines an arbitrary rectangle by specifying two points on the diagonal (Figure A.3). If only one point is specified, local origin is assumed to be the first point.

\begin{verbatim}
rect( pt1, pt2, material, mesh)
rect( pt1, material, mesh)
\end{verbatim}

A rectangle is a special case of a quadrilateral.
Appendix A. Sample Problem Definition in AFS

Figure A.4: Triangle primitive.

Figure A.5: Rectangle primitive.

**Arc definition:** the arc command defines an arbitrary arc. The first group of arc specifications calls for the external and internal radial of the arc and the angles specified with respect to the horizontal axis in the counter-clockwise manner. The second group of arc specifications is relative to the specified internal radius and smaller outer arc angle. If some of the specification elements are omitted they are taken to be zero.

- `arc( pt, r1, r2, phil, phi2, material, mesh)`
- `arc( r1, r2, phil, phi2, material, mesh)`
- `arc( r1, r2, phi, material, mesh)`
- `arc( pt, r, thickness, phi, delta_phi, material, mesh)`
- `arc( r, thickness, phi, delta_phi, material, mesh)`
- `arc( r, thickness, delta_phi, material, mesh)`
Pie definition: the pie command defines an arbitrary pie slice of a circle. A pie is specified by the circle radius and the two pie angles with respect to the horizontal axis. If some of the specification elements are omitted they are taken to be zero.

pie( pt, r, phi1, phi2, material, mesh)
pie( r, phi1, phi2, material, mesh)
pie( r, phi, material, mesh)  A pie is a special case of an arc primitive.

Circle definition: the circle command defines a circle of a given radius, centered at a given point. If the center point is not specified the center of the circle is assumed to be at the local origin.

circle( pt, r, material, mesh)
circle( r, material, mesh)  A circle is a special case of an arc primitive.
Reinforcement Bar definition: the \texttt{rbar} command defines a reinforcement bar. The diameter of the bar is specified using the bar numbers. If the overall length dimension is metric, then the bar numbers are in millimeters. Otherwise, the bar numbers represent eights of an inch. The bar material is specified by a material name.

\texttt{rbar( bar_number, material_name )}

The material for a geometric primitive is specified by a material name given in the parameter list. The mesh specification is optional. If desired, a mesh can be specified as

\begin{verbatim}
integer x integer
\end{verbatim}

using the symbol (x) as a separator. The first \texttt{integer} specifies the number of mesh divisions in \textit{x} direction in Cartesian or radial direction in polar coordinate system. The second \texttt{integer} specifies the number of mesh divisions in \textit{y} direction in Cartesian or angular direction in polar coordinate system. If the mesh is not specified, it is determined by A\textit{c}S using the moment of inertia heuristic.

A group of planar objects is constructed by \textit{placing} the constituent objects reference points at specified coordinates in a local coordinate system of a group.

An object reference point can be placed into the group directly as

\begin{verbatim}
object @ pt
\end{verbatim}

using @ as a separator, or by automatic generation, specified by

\begin{verbatim}
integer object @ lineSegment
\end{verbatim}

where the \texttt{integer} specifies the number of objects to be placed on a line segment. The reference point of the first object coincides with the first point, and the reference point of the last object coincides with the last point of the line segment. A \textit{line segment} is defined by two end points, as

\begin{verbatim}
pt~pt.
\end{verbatim}
using '-' as a separator. If both specified points are in Cartesian coordinates, the generation is performed on a line. If both points are in polar coordinates, the generation is performed on an arc, along a radius or along a spiral.

A planar objects can be declared either as positive (default), or negative (by preceding the object name with a minus sign) to provide for the specification of holes, openings and complex geometric cross-section shapes.

Therefore, BNF grammar specification for object placement is:

\[ [\text{integer}] \ [\pm] \ \text{object} \ [\# \ \text{pt} \ | \ \text{pt}^-\text{pt}] \]

A group of planar objects is specified by enclosing a list of objects in a pair of curly brackets. This group of objects can be assigned to a group name, using the equals (\(=\)) sign, or appended to an existing group of objects, using the increment symbol (\(+\)=). If the objects name is not specified, it is assumed that the group is a part of the cross-section definition.

The BNF grammar specification for object placement is:

\[ [\text{object name}] \ [\pm] \ [\pm] \ { [\text{object} \ [\text{primitive}] \ [, \ \text{object} \ [\text{primitive}] \ ]] \}

A planar object can be geometrically transformed to produce a new planar object. A\(\text{C}\)S supports the following geometric transformations:

- **Rotation**: the rotate command performs a planar rotation of an object about the local origin by a specified angle in the counter-clockwise direction.
  
  \[ \text{rotate( object, angle)} \]

- **Reflection**: the reflect command performs a planar reflection of an object about the specified line in the local coordinate system. A line is specified by a set of two points, using the same syntax as in object placement commands.
  
  \[ \text{reflect( object, line)} \]

- **Scaling**: the scale command enables a uniform scaling of an object in its local coordinate system by a specified factor.
  
  \[ \text{scale( object, factor)} \]

The cross-section geometry is defined in a cross-section geometry definition file. A geometry definition file defining to the sample cross-section (Figure A.1) is shown in Table A.1. First, the materials are defined. Then, the geometric primitives are specified, according to the cross-section partition shown in Figure A.9. Following this, the objects are assembled. Finally, the cross-section itself is defined.

### A.2 Cross-Section Materials

Materials are defined by invoking the initialization routines supplied with each material model. A file containing the definition of a particular material is supplied and material initialization routine is responsible for reading it. For example, a definition file for a generic bilinear material is shown in Table A.2.

### A.3 Cross-Section Analysis Task Definition

Two types of analysis tasks are supported in A\(\text{C}\)S. The first task is the analysis of a cross-section under a given load history. The second is the construction of the ultimate load locus of a given cross-section.

A **load history** is a sequence of load steps or previously defined load sequences. A **load sequence** consists of scaled and/or multiply repeated load steps or previously defined load sequences. A **load
Appendix A. Sample Problem Definition in AcS

Figure A.9: A possible method of cross-section decomposition into primitives and objects.

*step* is the atomic load sequence. It specifies a triplet of generalized loads along the three cross-section degrees of freedom (one axial, and two rotational degrees of freedom).

The cross-section degrees of freedom are specified with respect to a chosen set of coordinate axes. Two possible sets of coordinate axes are supported in AcS, through the *axes* command. One is the set of cross-section principal axes, with the origin placed at the cross-section centroid. The other is the original cross-section local coordinate system used during the assembly process.

A load step is specified by the following command:

```
```

A load step *component* has the form

```
type = value
```

where the component *type* is chosen according to the following table:
Appendix A. Sample Problem Definition in *AeS*

<table>
<thead>
<tr>
<th></th>
<th>axial dof</th>
<th>rotation dof</th>
<th>rotation dof</th>
</tr>
</thead>
<tbody>
<tr>
<td>force</td>
<td>P</td>
<td>Mx</td>
<td>My</td>
</tr>
<tr>
<td>displacement</td>
<td>eps</td>
<td>psix</td>
<td>psiy</td>
</tr>
</tbody>
</table>

and the component *value* is a number specifying either the increment of the respective generalized load component, or its absolute value.

The intensity of a generalized load component is specified in general force units, and general length units. The dimensions of these units is specified during loading initialization using the *units* command. They are internally converted to metric load (N) and length (m) units in order to preserve consistency. Rotations are specified in radians. The output of the analysis is converted back to user-specified units.

A load step must specify at least one and at most three load step components, each one corresponding to a distinct generalized load degree of freedom. If less than three components are specified, *AeS* retains the unspecified component value reached at the previous step. This is accomplished by internally setting the increment value of that generalized load step component to zero.

A load step can be scaled and/or repeated. Scaling is specified by a scale factor that precedes the load step command. Load step repetition is specified by a caret (^) followed by an integer multiple.

A load sequence is defined by enclosing a series of load steps or predefined load sequences in a pair of curly brackets. Once defined, a load sequence can be assigned to a load sequence name, using the equal sign (=), for absolute assignment, or the increment sign (+=) sign for appending the new load sequence to the end of the previously defined load sequence.

Therefore, the BNF grammar definition of a load step is

\[ \text{[scale factor]}^*\text{step( component [, component] [, component])}[^*\text{multiple}] \]

and of a load sequence

\[ \text{[load sequence name]} [= l +] \{ \text{[load sequence l step]} [, \text{load sequence l step}] ... \} . \]

The cross-section state must be initialized before the loading sequence is applied using the *initialize* command. The user can chose to initialize the cross-section to the virgin state or start from the current state. Starting from a desired state is accomplished by initializing to the virgin state and applying the load corresponding to the desired state in the first load step.

A load history is defined in a cross-section load definition file. A load definition file corresponding to the sample loading history (Figure A.2) is shown in Table A.3. First, load step sequences defining each leaf of the clover-leaf pattern are defined (Figure A.10). Then, these sequences are combined into a clover leaf sequence. Finally, the load history is constructed by scaling and repeating the clover leaf sequence.

The ultimate load locus analysis consists of applying a series of predefined load histories. Given the space in which the ultimate locus is to be found (e.g. $P$ vs. $M_y$ space for a given value of the bending moment $M_x$), each load history is formed so as to trace a different linear (proportional) load path in that space until the failure of the cross-sections detected. Combined together, these load histories form rays in the given ultimate locus space and the detected failure points form the ultimate load locus. The accuracy of the locus is specified by the user by providing the number of rays in a quadrant of the coordinate system.
Appendix A. Sample Problem Definition in NcS

# Sample cross-section definition
# initialization
units(cm)

# define materials
confConcrete = KPConcrete("confConcr.mdt")
unconfConcrete = KPConcrete("unconfConcr.mdt")
mpSteel = MPSteel("mpSteel1.mdt")
bilinSteel = Bilin("bilinSteel.mdt")

# define objects -- concrete
a1 = rect(15_45, confConcrete, 3x9)
a2 = rotate(a1, -90)
b = rect(15_15, confConcrete, 3x3)
c1 = rect(21_3, unconfConcrete, 7x2)
c2 = rect(3_21, unconfConcrete, 2x7)
d = rect(3_63, unconfConcrete, 2x21)
e = rect(3_45, unconfConcrete, 2x15)
f = rect(39_3, unconfConcrete, 2x13)
g = rect(60_3, unconfConcrete, 20x2)
h = circle(6, confConcrete, 2x8)

concrete = { a1 @ 3_18, a2 @ 18_18 } # define objects -- concrete
concrete += { b @ 3_3, c1 @ 0_63, c2 @ 63_0, d @ 0_0, e @ 18_18, f @ 21_18, g @ 3_0, -h @ 10.5_40.5 } # define objects -- steel

bar18 = rbar(18, MPSteel)
bar8 = rbar(8, bilinSteel)

steel = { 4*bar18 @ 3_63*10_63, 3*bar18 @ 63_3*63_18 } # define cross-section
steel += { 3*bar8 @ 3_18*3_48,
3*bar8 @ 18_18*18_48,
2*bar8 @ 33_18*48_18,
2*bar8 @ 33_3*48_3,
3*bar8 @ 3_3*18_3,
bar8 @ 3_18 }

# define cross-section
(concrete @ 0_0, steel @ 0_0 )

# end

Table A.1: Cross-section definition file specifying the geometry of the sample cross-section.
Appendix A. Sample Problem Definition in A&ES

E = 30000
Eh = 1500
Sy = 60

Table A.2: Data file for a generic bilinear material.

Figure A.10: Load pattern decomposition.
Appendix A. Sample Problem Definition in NcS

# Sample load file
# initialization
units(N,m)
axes(local)
initialize(virgin)

# define objects (chunks of steps)
seq1 = { step( psix += 1, psiy += 0),
step( psix += 0, psiy += 1),
step( psix += -1, psiy += 0),
step( psix += 0, psiy += -1) }

seq2 = { step( psix += 0, psiy += -1),
step( psix += -1, psiy += 0),
step( psix += 0, psiy += 1),
step( psix += 1, psiy += 0) }

seq3 = { step( psix += 1, psiy += 0),
step( psix += 0, psiy += -1),
step( psix += -1, psiy += 0),
step( psix += 0, psiy += 1) }

seq4 = { step( psix += 0, psiy += 1),
step( psix += -1, psiy += 0),
step( psix += 0, psiy += -1),
step( psix += 1, psiy += 0) }

cloverLeaf = { seq1, seq2, seq3, seq4 }

# start loading

step( N = 100 )

0.003*cloverLeaf^2
0.009*cloverLeaf^3

step( N = 250 )

0.015*cloverLeaf^3
0.024*cloverLeaf^3
0.033*cloverLeaf^3
0.042*cloverLeaf^3

# end loading

Table A.3: Cross-section definition file specifying the geometry of the sample cross-section.
Kajima-CUREe Project

Hysteresis Modeling of Reinforced Concrete Members

Final Project Report

Yasuo Murayama
Seiji Tokuyama
Kosuke Furuichi
Hachiro Ukon
Yoshihiro Hishiki
Norio Inoue
(Kajima Corporation)

February 1991
# Contents

Abstract

1. Introduction .......................................................... 1

2. Experiment ............................................................ 3
   2.1 Outline of Experiment ........................................... 3
   2.2 Specimens and Materials Used .................................... 4
      (1) Specimen Details ............................................. 4
      (2) Materials Used .............................................. 4
   2.3 Method of Experiment ........................................... 6

3. Analysis ............................................................. 8
   3.1 Fiber Models .................................................... 8
   3.2 Forming of Analytical Cross Section .......................... 13
   3.3 Concrete Models ................................................ 14
      (1) Stress-strain Envelope .................................... 14
      (2) Hysteresis Modeling ....................................... 16
   3.4 Reinforcing Bar Models ........................................ 17
      (1) Stress-strain Envelope .................................... 17
      (2) Hysteresis Modeling ....................................... 17

4. Verification of Analysis Through Comparison by Experiment .......... 21
   4.1 Uniaxial Bending ............................................... 21
      (1) Results of Experiment ..................................... 21
      (2) Comparison Between Analysis and Experiment .............. 22
   4.2 Biaxial Bending ................................................ 27
      (1) Results of Experiment ..................................... 27
      (2) Comparison Between Analysis and Experiment .............. 28

5. Conclusion .......................................................... 32

Publication Plan
4.1.17 Stress-strain history of reinforcement in specimen U1 (cubic model) ................................................................. 64
4.1.18 Stress-strain history of reinforcement in specimen U1 (r/o model) ................................................................. 65
4.1.19 Stress-strain history of concrete in specimen U1 (r/o model) ................................................................. 65
4.1.20 Moment/curvature diagram for specimen U4 (bilinear model)
: Rotation about X axis ............................................................................................................................................... 66
4.1.21 Moment/curvature diagram for specimen U4 (cubic model)
: Rotation about X axis ............................................................................................................................................ 67
4.1.22 Moment/curvature diagram for specimen U4 (r/o model)
: Rotation about X axis ............................................................................................................................................ 67
4.1.23 Stress-strain history of reinforcement in specimen U4 (bilinear model) ................................................................. 68
4.1.24 Stress-strain history of reinforcement in specimen U4 (cubic model) ................................................................. 68
4.1.25 Stress-strain history of reinforcement in specimen U4 (r/o model) ................................................................. 69
4.1.26 Stress-strain history of concrete in specimen U4 (r/o model) ................................................................. 69
4.1.27 Moment/curvature diagram for specimen U3 (bilinear model)
: Rotation about X axis ............................................................................................................................................... 70
4.1.28 Moment/curvature diagram for specimen U3 (cubic model)
: Rotation about X axis ............................................................................................................................................ 71
4.1.29 Moment/curvature diagram for specimen U3 (r/o model)
: Rotation about X axis ............................................................................................................................................ 71
4.1.30 Ratio of analytical to experimental maximum moment .......................................................................................... 72
4.2.1 Biaxial histeretic response of moment for specimen B1 (experiment) ................................................................. 76
4.2.2 Biaxial histeretic response of curvature for specimen B1 (experiment) ................................................................. 77
4.2.3 Moment/curvature diagram for specimen B1 (experiment)
: Rotation about X axis ............................................................................................................................................... 78
4.2.4 Moment/curvature diagram for specimen B1 (experiment)
: Rotation about Y axis ............................................................................................................................................... 79
4.2.5 Biaxial histeretic response of moment for specimen B2 (experiment) ................................................................. 80
4.2.6 Biaxial histeretic response of curvature for specimen B2 (experiment) ................................................................. 81
4.2.7 Moment/curvature diagram for specimen B2 (experiment)
: Rotation about X axis ............................................................................................................................................... 82
4.2.8 Moment/curvature diagram for specimen B2 (experiment)
: Rotation about Y axis ............................................................................................................................................... 83
4.2.9 Biaxial histeretic response of moment for specimen B3 (experiment) ................................................................. 84
4.2.10 Biaxial histeretic response of curvature for specimen B3 (experiment) ................................................................. 85
4.2.11 Moment/curvature diagram for specimen B3 (experiment)
: Rotation about X axis ............................................................................................................................................... 86
4.2.12 Moment/curvature diagram for specimen B3 (experiment)
: Rotation about Y axis ............................................................................................................................................... 87
4.2.13 Biaxial histeretic response of moment for specimen B1 (cubic model) ................................................................. 88
4.2.14 Biaxial histeretic response of moment for specimen B1 (r/o model) ................................................................. 89
4.2.15 Biaxial histeretic response of curvature for specimen B1 (cubic model) ......................................................... 90
4.2.16 Biaxial histeretic response of curvature for specimen B1 (r/o model) ................................................................. 91
4.2.17 Moment/curvature diagram for specimen B1 (cubic model) : Rotation about X axis ........................................... 92
4.2.18 Moment/curvature diagram for specimen B1 (r/o model) : Rotation about X axis ........................................... 93
4.2.19 Moment/curvature diagram for specimen B1 (cubic model) : Rotation about Y axis ........................................... 94
4.2.20 Moment/curvature diagram for specimen B1 (r/o model) : Rotation about Y axis ........................................... 95
4.2.21 Biaxial histeretic response of moment for specimen B2 (cubic model) ................................................................. 96
4.2.22 Biaxial histeretic response of moment for specimen B2 (r/o model) ................................................................. 97
4.2.23 Biaxial histeretic response of curvature for specimen B2 (cubic model) ............................................................ 98
4.2.24 Biaxial histeretic response of curvature for specimen B2 (r/o model) ............................................................ 99
4.2.25 Moment/curvature diagram for specimen B2 (cubic model) : Rotation about X axis ......................................... 100
4.2.26 Moment/curvature diagram for specimen B2 (r/o model) : Rotation about X axis ......................................... 101
4.2.27 Moment/curvature diagram for specimen B2 (cubic model) : Rotation about Y axis ......................................... 102
4.2.28 Moment/curvature diagram for specimen B2 (r/o model) : Rotation about Y axis ......................................... 103
4.2.29 Biaxial histeretic response of moment for specimen B3 (cubic model) ................................................................. 104
4.2.30 Biaxial histeretic response of moment for specimen B3 (r/o model) ................................................................. 105
4.2.31 Biaxial histeretic response of curvature for specimen B3 (cubic model) ............................................................ 106
4.2.32 Biaxial histeretic response of curvature for specimen B3 (r/o model) ............................................................ 107
4.2.33 Moment/curvature diagram for specimen B1 (cubic model) : Rotation about X axis ......................................... 108
4.2.34 Moment/curvature diagram for specimen B1 (r/o model) : Rotation about X axis ......................................... 109
4.2.35 Moment/curvature diagram for specimen B1 (cubic model) : Rotation about Y axis ......................................... 110
4.2.36 Moment/curvature diagram for specimen B1 (r/o model) : Rotation about Y axis ......................................... 111
4.2.37 Ratio of analytical to experimental maximum moment ..................................................................................... 112
## List of Tables

<table>
<thead>
<tr>
<th>Table Number</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1.1</td>
<td>Test arrangement</td>
<td>33</td>
</tr>
<tr>
<td>2.2.1</td>
<td>Mix proportion of concrete</td>
<td>36</td>
</tr>
<tr>
<td>2.2.2</td>
<td>Results of concrete cylinder tests</td>
<td>37</td>
</tr>
<tr>
<td>3.3.1</td>
<td>Input values for analytical concrete models</td>
<td>47</td>
</tr>
<tr>
<td>4.1.1</td>
<td>Characteristics of analytical $M-\phi$ relation compared to experiment</td>
<td>73</td>
</tr>
<tr>
<td></td>
<td>(specimen U2)</td>
<td></td>
</tr>
<tr>
<td>4.1.2</td>
<td>Characteristics of analytical $M-\phi$ relation compared to experiment</td>
<td>74</td>
</tr>
<tr>
<td></td>
<td>(specimen U1)</td>
<td></td>
</tr>
<tr>
<td>4.1.3</td>
<td>Characteristics of analytical $M-\phi$ relation compared to experiment</td>
<td>75</td>
</tr>
<tr>
<td></td>
<td>(specimen U4)</td>
<td></td>
</tr>
<tr>
<td>4.2.1</td>
<td>Features of the analytical values as compared with</td>
<td>113</td>
</tr>
<tr>
<td></td>
<td>the experimental values in the $M-\phi$ relationships,</td>
<td></td>
</tr>
<tr>
<td></td>
<td>on the B1 specimen</td>
<td></td>
</tr>
<tr>
<td>4.2.2</td>
<td>Features of the analytical values as compared with</td>
<td>114</td>
</tr>
<tr>
<td></td>
<td>the experimental values in the $M-\phi$ relationships,</td>
<td></td>
</tr>
<tr>
<td></td>
<td>on the B2 specimen</td>
<td></td>
</tr>
<tr>
<td>4.2.3</td>
<td>Features of the analytical values as compared with</td>
<td>115</td>
</tr>
<tr>
<td></td>
<td>the experimental values in the $M-\phi$ relationships,</td>
<td></td>
</tr>
<tr>
<td></td>
<td>on the B3 specimen</td>
<td></td>
</tr>
</tbody>
</table>
Abstract

There may be aseismatic design concept in which the verification of the energy absorption of the structure is intended against an earthquake having an intensity greater than the design earthquake. In this case, a non-linear earthquake response analysis of the structure is needed and sometimes a bi-directional earthquake response analysis is performed.

For the purpose of application of fiber model analysis for simulating the relationships between the bending moment and curvature of the reinforced concrete tower members (longitudinal steel ratio of 0.95%, compressive axial force of 80 kg/cm²) of cable-stayed bridges under biaxial bending, suitability of fiber models were investigated by paying attention to the reinforcing bar model.

Biaxial bending tests were performed on the RC-column specimens having rectangular cross section and the simulation analysis were performed on these specimens by using fiber model. As the consequence of the comparisons between analyses and experiments, the following facts have been clarified:

a. Uniaxial bending

<1> Whatever model, the Bilinear model, Cubic model, or Ramberg-Osgood (R/O) model, is used as the reinforcing bar model, can well simulate the $M-\phi$ relationships and can well express the influence of the magnitude of the axial force and steel ratio on the $M-\phi$ relationships.

<2> The degree of decrease in load after the maximum bending moment on the envelope of the $M-\phi$ relationships is slower in analysis than in experiment. Although the point where the load decreases steeply in experiment can be estimated on the Cubic model and R/O model, it is difficult to estimate on the Bilinear model.

<3> As far as the maximum bending moment is conserved, the Cubic model gives a higher accuracy of estimation as to tower members having a longitudinal steel ratio of 0.95%. On the other hand, the R/O model gives a higher accuracy of estimation in the case of columns of highrisied building, which have a higher steel ratio.

b. Biaxial bending

Examination on biaxial bending was performed on the specimens having a longitudinal steel ratio of 0.95% using the Cubic model and R/O model.

<1> As the result, although limited in above range, it has been confirmed that the results of the above mentioned uniaxial bending applies to biaxial bending also.

<2> Influence of biaxial loading hysteresis on the $M-\phi$ relationships can be expressed well on either model. However, the Cubic model gives a higher accuracy as to the magnitude of the maximum bending moment, so it is considered that the Cubic model is more suitable for analysis of the towers for cable-stayed bridges.
1. Introduction

The RC tower for cable-stayed bridges is a structural member not seen in conventional concrete-girder bridges, and is one of the most important structural members from the viewpoint of aseismatic design of cable-stayed bridges. While aseismatic design for bridges is normally performed by giving the bridge a required strength against the design seismic load in respective directions, longitudinal and transverse, of the bridge axis, there may be another design concept, in which the verification of the energy absorption of the bridge is intended against an earthquake having an intensity greater than the design earthquake. In this case, a non-linear earthquake response analysis of the bridge is needed and sometimes a bidirectional earthquake response analysis is performed. In the case of cable-stayed bridges having an A-shaped tower, in particular, the tower is always subjected to bending in the normal direction to the bridge axis, which is caused by the weight of girders via stay cables, and if a tower in this state is subjected to a seismic load in the longitudinal direction of the bridge, a biaxial bending moment occurs in the tower member even in the case of a unidirectional earthquake.

To perform this type of responses analysis, it becomes necessary to model analytically the biaxial restoring force characteristics of tower members. There are the following methods to model the column members: a method in which the relationships between lateral force and displacement (P-δ relationships) are formed into models, a method in which the relationships between the bending moment and curvature (M-θ relationships) at each portion of the member are formed into models and the curvature is integrated in the direction of the member axis, and a method using a spring model as a method between the above two, in which the relationships between the bending moment and angle of rotation (M-θ relationships) are expressed. These analytical models have been proposed mainly for RC columns of a building.

Unlike columns of a building, the tower members for cable-stayed bridges are slender, i.e., a large ratio of column height to side length of cross-section, so a dominant factor in their design is bending moment rather than shear force. Therefore, except for the quite limited parts near the joints with bridge piers, analytical models to express the M-θ relationships are more convenient rather than P-δ or M-θ relationships. Models to express the M-θ relationships are further divided into plastic models (macro models), in which a particular rule is given to hysteresis, and fiber models (micro models) which start from the hysteresis models of structural materials used. Of these models the latter is considered to have a wider range of applications to members with various specifications of cross section.

A tower member is, in terms of compressive axial force per unit area of cross section, on the upper side of the range of columns of a building. A tower member and the column of a building are largely different to each other in the shape of cross section and cross sectional specifications such as the amount of reinforcement and compressive strength of concrete. In the case of columns of a building, the cross section is normally square, the steel ratio of reinforcement is 1 to 2.5%, and the compressive strength of concrete in use is 210 to 400 kg/cm². In the case of tower members on the other hand, the cross section is normally rectangular, the steel ratio of reinforcement is
around 1%, and the compressive strength of concrete is about 400 to 600 kg/cm².

Further, if the steel ratio and compressive strength of concrete are converted together into steel coefficients (Pfy/f'c), the steel coefficient of tower members is significantly smaller than that of columns of a building.

A fiber model has so far been verified against the normal type of columns of a building through correlating the test results of short column specimens subjected shear and bending. Therefore, in this study the suitability of the fiber model to the M-ψ relationships of tower members for cable-stayed bridges and the effect of the difference of the analytical model of reinforcing bars on the M-ψ relationships are investigated.

To check the suitability of fiber model analysis, biaxial bending tests were performed for comparison, on tower model specimens subjected to axial force.

For the purpose of the experiment, compressive axial force was applied to the specimen via an unbonded tendon, which passed through the center of the cross-section of the member, so as to eliminate the so-called P-δ effect from the experiment. Further, the bending moment was applied as pure bending so as to eliminate the effect of shear force. Specimens were designed taking into consideration the features of the cross section of tower members as described above.

Prior to biaxial bending tests, a series of uniaxial bending tests were conducted to understand the features of influential factors in a simple condition.
2. Experiment

2.1 Outline of Experiment

In the experiment, bidirectional bending moment hysteresis was applied to test specimens under a constant compressive axial force. As ideal conditions, a bending moment was applied in pure bending, and compressive axial force was applied via unbonded tendons which pass through the center of the cross section.

Prior to a series of biaxial bending tests, a series of uniaxial bending tests were conducted to examine the effects of the magnitude of compressive axial force and the steel ratio of longitudinal reinforcements on the bending moment-curvature relationships. These tests were conducted under four different sets of conditions, in which the basic specimen with a 0.95% steel ratio was subjected to 80 kgf/cm² of axial force assumed for RC tower members, a specimen with the same steel ratio as the basic specimen was subjected to 7 kgf/cm² of axial force assumed for normal bridge piers, a specimen with a 2.85% steel ratio was subjected to same axial force as the basic specimen assumed for heavily reinforced RC columns such as columns of bundling, and a specimen with a 1.90% steel ratio, an intermediate value between those of the latter two specimens, was subjected to 80 kg/cm² of axial force.

In biaxial bending tests, which were conducted using specimens having the same steel ratios and axial force as for uniaxial tower model specimen, a bending moment was applied reversally at 45° from the principle axis, reversally around the X axis with a constant bending moment applied around the Y axis, and randomly in two directions by simulating the modified Elcentro earthquake acceleration records.

Details of each specimen and loading patterns are summarized in Table 2.1.1.
2.2 Specimens and Materials Used

(1) Specimens Details

Specimens with rectangular cross sections (160 x 250 mm) as shown in Figure 2.2.1, which are based on the 1/20-reduced models of actual towers of cable-stayed bridges, were used. On these specimens, a 500-mm length at the center was expected to be the measuring zone and both ends were tapered and reinforced by steel bars so as to protect the specimen against stress concentration.

The same external dimensions were used for all specimens, and the steel ratio of 0.95% in the longitudinal direction was same for each specimen except the U3 and U4 specimens. The steel ratio in the longitudinal direction of the U3 and U4 specimens was 1.90% and 2.85% respectively, which was two times and three times larger than the steel ratio of the basic specimen.

The hoop reinforcement ratio in the observation zone was 0.16% for all specimens. The interval between hoops was 35 mm. Further, a more condensed interval of 15 to 18 mm was used for the fixed end zones of the specimen.

(2) Materials Used

a) Reinforcing bars

The D6 steel bars (SD30) were used for reinforcing bars in the longitudinal direction and scaled deformed 3-mm diameter steel bars (SD30) were used for hoops.

According to the test results of reinforcing bars, the yield strength of the D6 steel bar was 3030 kgf/cm² (43.10 ksi) and its tensile strength was 4785 kgf/cm² (68.06 ksi). The yield strength of the scaled deformed 3-mm diameter steel bar was 3250 kgf/cm² (46.2 ksi) and its tensile strength was 4930 kgf/cm² (70.1 ksi). Figure 2.2.2 shows the stress-strain relationships of the D6 steel bar. In this case, the value of the load divided by the average cross section of the reinforcing bar, was used for the stress and the value measured with a strain gauge with a 5-mm length attached to the surface of the reinforcing bar was used for the strain.

b) Concrete

Table 2.2.1 shows the mix proportion of the concrete used for specimens. The compressive strength of the concrete cylinder (ø100 x 200 mm) during the experiment was within the range of 400 to 480 kgf/cm² as shown in Table 2.1.1. Another series of concrete tests was conducted by the strain control method (strain rate of 243 μ/min) to grasp the stress-strain relationships of concrete. The test results are shown in Table 2.2.2 and Figure 2.2.3.
Table 2.2.2 shows the results of the tensile tests (Brazilian tests) of concrete cylinder, which were conducted in parallel with the compression test. From these results, it was found that the tensile strength was about 1/13 less than the compressive strength.
2.3 Method of Experiment

(1) Loading method

Load was applied using the unit shown in Figure 2.3.1 and Figure 2.3.2 to produce combined axial force and bidirectional pure bending moments. The unit consists of a total of four jacks: one is used to give compressive axial force and the remaining three are push-pull jacks for applying bending force in arbitrary directions. The respective force of the three jacks for applying bending moments (J1, J2, and J3) were calculated using the following expressions as to an arbitrary bending moment \( M \), and the direction of bending moment vector \( \theta_M \).

\[
\begin{align*}
\text{<1> } & \text{ In the case where } 0^\circ \leq \theta_M \leq 45^\circ, 135^\circ \leq \theta_M \leq 225^\circ, 315^\circ \leq \theta_M \leq 360^\circ \\
J_1 &= (2/3) \cdot (M/r) \cdot \cos \theta_M \\
J_2 &= (\sqrt{3} \cdot \tan \theta_M - 1)/3 \cdot (M/r) \cdot \cos \theta_M \\
J_3 &= (\sqrt{3} \cdot \tan \theta_M + 1)/3 \cdot (M/r) \cdot \cos \theta_M \\
\text{<2> } & \text{ In the case where } 45^\circ < \theta_M < 135^\circ, 225^\circ < \theta_M < 315^\circ \\
J_1 &= (2/3) \cdot (M/r) \cdot \tan \omega \cdot \cos \theta_M \\
J_2 &= (\tan \omega + \sqrt{3})/3 \cdot (M/r) \cdot \cos \omega \\
J_3 &= (\tan \omega - \sqrt{3})/3 \cdot (M/r) \cdot \cos \omega
\end{align*}
\]

Here,

\( \omega = \theta_M - 90^\circ \)

\( r \) : Distance between the center of the cross section of specimen and the center of the jack ( \( r = 0.5 \) m)

Load control for each hydraulic jack was accomplished with a computer, by automatically controlling the open-close operation of the electromagnetic valve while metering the value indicated on the load cell attached to the jack.

(2) Measuring method

The average curvature \( \phi \) along the gauge length and the direction of curvature vector \( \theta \) were obtained with the following procedures:

\text{<1> Measurement of the relative displacement in the measuring zone}

Larger side length of the cross section multiplied by two (50cm) was selected as gauge length for relative displacement measurement (see Figure 2.2.1). By fastening band-like jigs around cross sections A and B and attaching four displacement ring gauges (of 1/1000-mm sensitivity) to each angle of the jigs, the relative displacement at each position \( (D_1, D_2, D_3, \text{ and } D_4) \) was measured.
<2> Calculation of the most probable value of each measured relative displacement

A plane most suitable for the four points is applied using the least square method. The most probable value for each of the four points is obtained as follows:

\[
D_1^* = \frac{(3D_1 + D_2 - D_3 + D_4)}{4} \\
D_2^* = \frac{(D_1 + 3D_2 + D_3 - D_4)}{4} \\
D_3^* = \frac{(-D_1 + D_2 + 3D_3 + D_4)}{4} \\
D_4^* = \frac{(D_1 - D_2 + D_3 + 3D_4)}{4}
\]

<3> Calculation of the maximum gradient direction and the direction of curvature vector

The direction to provide the maximum angle between the two cross sections (cross sections A and B) separated from each other by \( S_0 \) (the maximum gradient direction) is obtained using the three points including the maximum and minimum values \( (D_a > D_b > D_c) \) of the most probable values. Figure 2.3.3 shows the state where the displacement measuring section after being deformed is cut along the direction of maximum gradient composed with two cross sections A and B; Figure 2.3.4 shows the relationships between the measurement points and the maximum gradient directions. It is enough to obtain the direction on which \( B'a, B'b \), and \( B'c \) are on the same line based on the geometric similarity condition, so the maximum gradient direction (direction of radius vector) \( \theta_R \) can be obtained by the following formula:

\[
\tan \theta_R = -\frac{\left( d_{cb}A_{ba} - d_{ba}A_{cb} \right)}{\left( d_{cb}B_{ba} - d_{ba}B_{cb} \right)}
\]

Here,
\[
d_{ij} = D_i - D_j \\
A_{ij} = r_i \cos \theta_i - r_j \cos \theta_j \\
B_{ij} = r_i \sin \theta_i - r_j \sin \theta_j \\
r_i = \sqrt{x_i^2 + y_i^2} \\
\theta_i = \tan^{-1}(y_i/x_i)
\]

Also, the direction of curvature vector \( \theta_\phi \) (see Figure 2.3.4) can be obtained by the formula of \( \theta_\phi = \theta_R + \pi/2 \).

<4> Calculation of the average curvature along the gauge length

The average curvature \( \phi \) can be obtained by the following formula using the maximum and minimum values among the most probable values

\[
\phi = (D_a - D_c) / (\Delta R(S_0 + (D_a + D_c)/2))
\]

Here,
\[
\Delta R = (A_{ac} \cos \theta_R - B_{ac} \cos \theta_R) / 2
\]
3. Analysis

3.1 Fiber Model

Concept and algorithm are described in the BICOLA user's manual. Following description is extraction from the BICOLA user's manual.

(1) Section Discretization and Background Theory

The column section is arbitrary in shape. It is idealized as an assemblage of steel and concrete fibers defined in an orthonormal right hand coordinate system, normal to the reference axis of the generic member. All fibers are assumed to be in a uniaxial stress state. The Bernouilli-Navier kinematic assumption is assumed valid, namely that plane sections remain plane. Effects of shear on the kinematics as well as on the stress state of the materials are ignored. Similarly, any bond deterioration and strain incompatibilities between neighboring steel and concrete are disregarded. However, initial prestressing (stresses or strains) is allowed in the formulation.

Under the plane sections assumption, the section kinematic degrees of freedom that fully define the section state consist of the reference strain at the section origin and two orthogonal curvatures about the section axes,

\[ v = < \varphi_x, \varphi_y, \epsilon_o > \]

The corresponding section resistance degrees of freedom are

\[ S = < M_x, M_y, N > \]

namely axial load and orthogonal section moments defined the plastic centroid.

The section location about which the moment is reported is user defined. Usually this is the plastic centroid of the section; if not specified, the program will calculate internally the plastic centroid coordinates (about which all moments will be calculated) during the input state. For estimation of this point, the section is assumed to be under no bending, while the section stress state follows the ACI recommendation, the tension steel is yielded and the concrete stress block is assumed to have the equivalent rectangular shape. The intensity of the stress block is also user defined (typically 85% of the maximum compressive stress \( f_{\text{c,cm}} \) of the concrete is also used, this also being the default value).

(2) Fiber Strains

Under the assumption of plane sections remaining plane, the \( i \)th fiber strain is evaluated by the linear transformation

\[ \epsilon_i = -\epsilon_0 + \varphi_y (x_i - x_o) + \varphi_x (y_i - y_o) \]

\[ \epsilon_i = a_i \nu_i \]
where $Q_3$ is the displacement transformation matrix. Coordinates $x_i, y_i$ are the fiber centroid coordinates in the orthonormal system, defined during generation and stored internally. Coordinates $(x_0, y_0)$ are the coordinates of the point at which the strain is $\varepsilon_0$ (the reference strain).

Following the variable excitation feature of the uniaxial programs, the structure of BICOLA also allows for different controls in the analysis stage. The user has the option of specifying at any stage of the analysis session, which type of procedure should be activated, through the Main Analysis Menu. Depending on the type of analysis option activated, the controlled (known) section degrees of freedom differ. Due to the larger amount of section parameters involved in the biaxial case, the analyst has a wider set of options to choose from. Possible candidates are the two curvatures ($\phi_x$, $\phi_y$), the two neutral axis intercepts with the coordinate axes passing through the section origin $(x_{na}, y_{na})$ and the reference strain ($\varepsilon_0$ at $x_0, y_0$), or combinations of the above in groups of three.

Additional flexibility is given to the program for calculating section states given that the strain at a certain location other than the reference origin is controlled (namely $\varepsilon_x$, at $x", y"$). Clearly, in this case the number of deformation control combinations becomes too large, therefore only a limited set has been implemented, those being most likely to be used in analysis procedures. In addition, further combined force-deformation control options are also available, and discussed in subsequent sections. These options are necessarily iterative in nature, making use of a certain deformation degree of freedom (above) as the iteration unknown.

Among others, cases of section state definition currently considered (subroutine strain) are as follows:

a) strain at a point and the two neutral axis intercepts with the two orthonormal axes through the origin known:

$$\varnothing_x = \frac{\varepsilon \cdot x_{na}}{(y^* \cdot x_{na} + x^* \cdot y_{na} - x_{na} \cdot y_{na})}$$  

$$\varnothing_y = \frac{\varepsilon \cdot y_{na}}{(y^* \cdot x_{na} + x^* \cdot y_{na} - x_{na} \cdot y_{na})}$$

b) strain at a point, neutral axis intercept with one of the axes and (in plane) curvature vectorially collinear with this same axis known:

$$\varnothing_x = (\varepsilon^* - x^* \cdot \varnothing_y) / (y^* - y_{na})$$

$$x_{na} = y_{na} \varnothing_x / \varnothing_y$$

or (other direction)

$$\varnothing_y = (\varepsilon^* - y^* \cdot \varnothing_x) / (x^* - x_{na})$$

$$y_{na} = x_{na} \varnothing_y / \varnothing_x$$
c) strain at a point and the two out-of-plane curvatures known:

\[ x_{na} = x' + (y' \partial_x - \epsilon')/\partial_y \]  
\[ y_{na} = y' + (x' \partial_y - \epsilon')/\partial_x \]  

Once the section deformation degrees of freedom are fully defined, the ith fiber strain is calculated with a modification of Eq. 3.3, using neutral axis and curvatures instead of reference strain values:

\[ \epsilon_i = (y_i - y_{na}) \partial_x + x_i \partial_y \]  

\[ \partial_y = \epsilon'/(x' - x_{na}) \]  

e) uniaxial bending only. Neutral axis and curvature known (again, either direction is permissible); the strain is calculated in both uniaxial cases (d) and (e) by

\[ \epsilon_i = (y_i - y_{na}) \partial_x \]  

The program at any stage will internally switch to the appropriate uniaxial search if the out-of-plane bending curvature is smaller than a default minimum in absolute value. Such a minimum curvature is usually selected equal to \(10^{-5}\) divided by the maximum section dimension.

Numerically ill-defined situations, such as for instance the denominator of Eqs. 3.5, 3.6 or 3.14 being equal to zero (implying alignment of the neutral axis and the monitored strain location in the spatial case), are detected and a warning diagnostic given. This forces modification of the input, if a relevant deformation control analysis option is still active.

Calculated strains are subsequently fed to the material model routines in order to establish corresponding stress. The mathematical material models are discussed in next section. Following the sign convention of UNCOLA, fiber stresses and strains are positive in tension. However, reference strain and axial loads are positive in compression.

(3) Section Stiffness

The section tangent stiffness matrix \(k_s\) is calculated at any converged state of the analysis and stored in file name "stf". The stiffness can be determined by the standard virtual work integration

\[ k_s = \sum_{i=1}^{n} a_i^T E a_i \, dA \]  

- 10 -
over all the section fibers. Following a trapezoidal rule numerical integration, the formulation can be recast to the summation idealization for the entire section stiffness, coupling section deformations with forces,

\[ S_i = k_i \nu_i \quad 3.17 \]

For the three-dimensional problem this stiffness is equal to:

\[
k_i = \begin{bmatrix}
\sum_{j=1}^{n} A_j E_j x_j y_j^2 & \sum_{j=1}^{n} A_j E_j x_j y_j & -\sum_{j=1}^{n} A_j E_j y_j \\
\sum_{j=1}^{n} A_j E_j x_j y_j & \sum_{j=1}^{n} A_j E_j x_j^2 & -\sum_{j=1}^{n} A_j E_j x_j \\
-\sum_{j=1}^{n} A_j E_j y_j & -\sum_{j=1}^{n} A_j E_j x_j & \sum_{j=1}^{n} A_j E_j \\
\end{bmatrix} \quad 3.18
\]

where, \( E_j \) is the current tangent modulus of the \( j \)'th fiber returned by the material model routines and \( A_j, x_j, y_j \) the area and coordinates of the fiber. In the actual implementation, the evaluation of the stiffness terms is more efficient by making use of the fact that different groups of fibers can be defined having equal area. The stiffness is determined about the plastic centroid, the point about which section resisting forces are also reported. The off diagonal coupling terms of the section stiffness (Eq. 3.18) vanish in the case of a biaxially symmetric homogeneous elastic section with the origin of the section axes set at the transformed section centroid.

(4) Initial Strains

Specification of initial strains is possible for selected material fibers during the input and edit options. Alternatively, stresses can also be specified, which are internally converted into strains assuming monotonic loading from zero strain and no previous cycling history (this is possible for all steel models; the original UNCOLA version allowed initial stress only for the bilinear model). At the fiber strain level, any initial stress \( (\varepsilon_{pstr}) \) is added on the calculated values as

\[ \varepsilon_i = \varepsilon_i + \varepsilon_{pstr} \quad 3.19 \]

Residual stress in the section is also handled in the same manner, being converted into initial strains.

(5) Internal Forces

Axial load and bending moments about the two orthonormal axes are estimated by stress integration, also following a trapezoidal rule. Stresses
are assumed to be constant within the fiber, equal to the value at the fiber centroid, the location monitored within the program. Following the group program organization, the section load vector is evaluated from the fiber stress \( f_j \) of the \( i \)th group through

\[
N = - \left( \sum_{i=1}^{\text{nc}} A_i \sum_{j=1}^{\text{sc}} f_{ij} + \sum_{i=1}^{\text{nsc}} A_i \sum_{j=1}^{\text{ss}} f_{ij} \right) 
\]

\[
M_x = \sum_{i=1}^{\text{nc}} A_i \sum_{j=1}^{\text{sc}} f_{ij} y_{ij} + \sum_{i=1}^{\text{nsc}} A_i \sum_{j=1}^{\text{ss}} f_{ij} y_{ij} 
\]

\[
M_y = \sum_{i=1}^{\text{nc}} A_i \sum_{j=1}^{\text{sc}} f_{ij} x_{ij} + \sum_{i=1}^{\text{nsc}} A_i \sum_{j=1}^{\text{ss}} f_{ij} x_{ij} 
\]

where \( c, s \) denote concrete and steel, respectively. Furthermore, \( \text{ngc} \) is the number of concrete and \( \text{nsc} \) the number of steel groups, \( \text{nc} \) the number of concrete fibers and \( \text{ns} \) the number of steel fibers per group (similar double summations are followed in the estimation of the section stiffness, following Eq. 3.18).

Moments are subsequently converted to values at the plastic centroid, (subroutine mplcnt) through

\[
M_x^p = M_x - N y_c , \quad M_y^p = M_y - N x_c 
\]
3.2 Forming of Analytical Cross Section

The analytical cross section simulates the cross section of the specimen (see Figure 2.2.1) accurately, and for the basic specimen, it was formed into a model with 432 concrete fibers and 12 reinforcing fibers as shown in Figure 3.2.1. In this case, different concrete fiber models were used for the inside and outside of the hoops.

For the U3 and U4 specimens, 24 and 36 reinforcing fibers were used respectively, according to the number of reinforcing bars in the specimen.
3.3 Concrete Models

(1) Stress-strain Envelope

The model which can be dealt with in BICOLA program is general enough to represent several uniaxial stress-strain laws proposed for reinforced concrete. Such models consist in general of an initial quadratic curve, defined by the equation

\[ \frac{f}{f_c} = \frac{\varepsilon}{\varepsilon_0} \left[ 2 - \frac{\varepsilon}{\varepsilon_0} \right] \]

with an initial tangent modulus given by the expression

\[ E' = 2 \frac{f_c}{\varepsilon_0} \]

and zero tangent modulus at \( \varepsilon_0 \); beyond the initial quadratic the behavior is expressed by a constant resistance portion (optional) and a linearly decaying branch to zero or nonzero strength

\[ f = Z f_c (\varepsilon_c - \varepsilon_{20\%}) \]

which remains constant beyond that point and is reduced to 0. beyond a specified strain.

\[ f = f_{20\%} \quad \text{for} \quad \varepsilon_{20\%} < \varepsilon < \varepsilon_c \]

\[ f = 0 \quad \text{for} \quad \varepsilon_c < \varepsilon \]

The key parameters of the above family of models, as used in Eqs. 3.24 through 3.27 above, are as follows:

- \( f_c \), the maximum concrete compressive stress
- \( \varepsilon_0 \), the corresponding strain
- \( \varepsilon_t \), the ending strain of the flat portion
- \( f_{20\%} \), the concrete compressive stress denoting the end of the decay portion of the model (such as 20% \( f_c \)) in stress units
- \( \varepsilon_{20\%} \), the corresponding strain
- \( Zf_c \), the linear decay slope
- \( \varepsilon_{cr} \), the crushing concrete strain
- \( f_t \), the maximum tensile concrete stress
In this study, the core concrete confined by hoops (confined concrete) and the cover concrete outside the hoops (unconfined concrete) were respectively formed into models, and then according to the type of concrete the input data for the above mentioned general analytical models were determined.

a) Models of unconfined concrete

Analytical models of unconfined concrete were determined based on the test results of concrete cylinder. The stress-strain envelope consists of two parts, i.e., a parabolic curve within the range until the stress reaches a peak, and a straight line within the range after the peak. These results are shown in Figure 3.3.2 together with the test results.

The tensile strength was taken as being 1/13 lower than the compressive strength based on the test results of concrete cylinder.

b) Models of confined concrete

Since no test was performed on confined concrete, models for analysis were decided based on the Kent & Park method, which has so far been published.

The stress-strain relationships of confined concrete can be obtained by the following formula, based on unconfined concrete (Figure 3.3.2), and are shown in Figure 3.3.3.

region AB: \( \varepsilon_c \leq 0.002 \)

\[
f_c = f' \left( \frac{2 \varepsilon_c}{0.002} - \left( \frac{\varepsilon_c}{0.002} \right)^2 \right)
\]

3.29

region BC: \( 0.002 \leq \varepsilon_c \leq \varepsilon_20c \)

\[
f_c = f' \left[ 1 - Z(\varepsilon_c - 0.002) \right]
\]

3.30

where

\[
Z = \frac{0.5}{\varepsilon_{50u} + \varepsilon_{50h} - 0.002}
\]

3.31

\[
\varepsilon_{50u} = \frac{3 + 0.002f'}{f' - 1000}
\]

3.32

\[
\varepsilon_{50h} = \frac{3}{4} \rho \sqrt{\frac{b''}{S_h}}
\]

3.33

\[
\varepsilon_{50c} = \varepsilon_{50u} + \varepsilon_{50h}
\]

3.34
\[ \varepsilon_{20c} = \frac{8}{5} (\varepsilon_{\infty} - 0.002) + 0.002 \quad 3.35 \]

region CD: \( \varepsilon_{20c} \leq \varepsilon_c \)

\[ f_c = 0.2 f_{c'} \quad 3.36 \]

where \( f_{c'} \) = concrete cylinder strength (6300psi). \( \beta_s \) = ratio of volume of transverse reinforcement to volume of concrete core measured to outside of hoops (0.0048), \( b' \) = width of confined core measured to outside of hoops (134mm), and \( sh \) = spacing of hoops (35mm).

Consequently, the following values are obtained:

\[ \varepsilon_{50u} = 0.003 \]
\[ \varepsilon_{50h} = 0.007 \]
\[ \varepsilon_{50c} = 0.010 \]
\[ \varepsilon_{20u} = 0.015 \]

If the stress-strain relationships are to be determined by the Sheikh & Uzumeri method, the magnitudes of strain at the beginning of the range where the stress on the model is constant, becomes larger than that at the end due to a large strength of concrete (\( \varepsilon_o > \varepsilon_f \)) and causes a contradiction in the case of application. However the values themselves are almost identical with each other and almost the same as those obtained by the Kent & Park method. (See Figure 3.3.1.)

If the above results are expressed by the input form for analytical models (Figure 3.3.1), each input value is as shown in Table 3.3.1.

(2) Hysteresis Modeling

As for the hysteresis of the above concrete models, the initial elastic coefficient (Ec) was used for the gradient at the time of stress reducing, as shown in Figure 3.3.4.
3.4 Reinforcing Bar Models

In the biaxial M-Ø analysis program, BICOLA, Bilinear, Cubic, and Ramberg-Osgood (R/O) models are provided as reinforcing bar models.

(1) Stress-strain Envelope

The form of the envelope of reinforcing bar models is shown in Figure 3.4.1, which was based on the tensile test results of reinforcing bars (Figure 2.2.2). However, the strain in the test was measured by strain gauges which were attached on the surface between knots, so the measured value is slightly larger than the average strain of reinforcing bars. Therefore, the strain which was compensated with reference to existing experiment results, was used for the analytical models.

- Yield strength ($f_y: 3030 \text{ kgf/cm}^2 = 43.1 \text{ ksi}$)
- Yield strain ($\varepsilon_y: 0.00158$)
- Initial elastic modulus ($E_c: (1.918 \times 10^6 \text{ kgf/cm}^2 = 27280 \text{ ksi})$)
- Tensile strength ($f_{\text{max}}: 4785 \text{ kgf/cm}^2 = 68.06 \text{ ksi}$)
- Strain at strain hardening ($\varepsilon_{sh}: 0.01516$)
- Strain at maximum stress ($\varepsilon_{\text{max}}: 0.095$)
- Elastic modulus at strain hardening ($E_{sh}: (0.0633 \times 10^6 \text{ kgf/cm}^2 = 900 \text{ ksi})$)

In the case of the Bilinear model, $f_y$, $E_s$, and $E_{sh}$ only were given and coefficient $\alpha$ for the Cubic model was set to 0.

(2) Hysteresis Models

a) Bilinear model

For the Bilinear model, the envelope is as shown in Figure 3.4.1(a) and the hysteresis is as shown in Figure 3.4.2. In this hysteresis, a linear relationship between stress and strain applies, in which the initial elastic modulus is used within the range until yield of the reinforcing bar occurs, but after yielding, the elastic modulus at strain hardening applies. When the strain decrease, the stress decreases linearly with strain, where the initial elastic modulus applies.

b) Cubic model

(Most of description in this section is extracted from BICOLA User's Manual)

For the Cubic model, the envelope is as shown in Figure 3.4.1(b) and is an equation of the third degree.
Typical regions are

(1) the monotonic region at onset of hardening (virgin curve) and
(2) any stress reversal post yield curve exhibiting the Bauschinger
effect;

The cubic equation is defined by translation of the origin on the
zero stress axis, the initial slope at the origin and the stress and
strain at ultimate tensile strength. For continuity, this initial slope
is equal to the hardening slope on the monotonic curve or equal to the
unloading slope on reversal. At the ultimate, a zero tangent modulus is
assumed. The defining equations for the stress and the tangent modulus
$E_t$ are:

$$f = \alpha \varepsilon^3_t + \beta \varepsilon^2_t + E_0 \varepsilon \quad , \quad \varepsilon < \varepsilon_{\text{max}}$$  \hspace{1cm} 3.37

$$f = f_{\text{max}} \quad , \quad \varepsilon > \varepsilon_{\text{max}}$$

$$E_t^{\text{max}} = \frac{df}{d\varepsilon} = 3\alpha \varepsilon^2_t + 2\beta \varepsilon_t + E_0$$  \hspace{1cm} 3.38

where, in order to satisfy the boundary conditions, $\alpha$ and $\beta$ are defined
by

$$\alpha = \frac{(E_0 \varepsilon_{\text{max}} - 2f_{\text{max}})}{\varepsilon_{\text{max}}}$$  \hspace{1cm} 3.39a

$$\beta = \frac{(3f_{\text{max}} - 2E_0 \varepsilon_{\text{max}})}{\varepsilon_{\text{max}}^2}$$  \hspace{1cm} 3.39b

where $E_0$ is the initial tangent modulus and $f_{\text{max}}, \varepsilon_{\text{max}}$ are the maximum
stress and strain, respectively at which the zero slope condition is
enforced. A constant stress equal to the ultimate strength is imposed
beyond the ultimate strain $E_{\text{ult}}$, since the cubic expression cannot model
a softening path. All of the above are user defined.

The initial monotonic bounding curve (also used in the remaining
nonlinear models) is defined by $f_y$ and four additional parameters. These
are

- the onset of hardening strain $\varepsilon_h$ as a (false) origin,
- the hardening slope $E_h$ for the initial slope of the cubic and
- $f_{\text{ult}}$ the maximum steel resistance,
- $E_{\text{ult}}$ the corresponding strain (namely $E_{\text{max}}, f_{\text{max}}$ of equ. 3.37).

Any reversal curve is defined with a (false) origin at the strain
intercept of the elastic unloading line with the zero stress axis. The
new curve uses for initial slope continuity the elastic tangent modulus
of the material. Usually this corresponds to $E_0$, though it is possible
to define a degradation parameter $\chi$ that forces the unloading modulus
(from the point of strain reversal) to be a fraction of $E_o$ depending on the accumulated plastic strain, following

$$E_{\text{ult}} = E_o \left( \frac{\varepsilon_p}{\varepsilon_{\text{pl}}} \right)^a$$

3.40

The exponent $\alpha$ can be 0. For the ultimate stress value the same assumptions as for the monotonic curve are used by default, thus any isotropic hardening is excluded.

Problems in the cubic expression may arise in the implementation of the model due to the nature of Eq. 3.38. Such problems were observed in earlier versions of the section programs, and have been corrected by constraining constants $\alpha$ and $\beta$. Due to the possibility that the cubic defined over the interval $[E_o, E_{\text{max}}]$ has a mathematical point of inflection within this interval (a point of zero slope derivative, depicting the physically unrealistic behavior of an additional constraint is further imposed.

$$\frac{1}{3} \leq \frac{f_{\text{max}}}{E_{\text{ult}}} \leq \frac{2}{3}$$

3.41

This constraint ensures that a single turning point occurs within the interval over which the cubic is defined.

In order to satisfy this condition, a shifting of the user defined $E_{\text{max}}$ may be required sufficiently near (or far) from the current origin $E_o$ such that physically unrealistic transitions of the stress to the ultimate value will be avoided. In the case of the monotonic post-hardening envelope curve, the error is not significant for the range of values of $E_o$, $f_y$, $E_{\text{max}}$ and $f_{\text{max}}$ of mild steel. Generally the curve is not recommended for use in cyclic modeling of reinforcing steels which exhibit a pronounced Bauschinger curvature which is larger than that of the shifted cubic. This is usually the case for small amplitude reversals, where inadequate modeling may be obtained.

Here, coefficient $\alpha$ is a value related to the form of loop in the case where the stress decreases and takes a value between 0 and 1; the larger this value is, the thinner the form of loop becomes. Coefficient $\alpha$ is assumed to be "0" in the analysis at this time.

Figure 3.4.3 shows an example of the stress-strain history of reinforcing bar using the Cubic model.

c) Remberg-Osgood model

For the Remberg-Osgood model, the envelope is as shown in Figure 3.4.1 (b), as in the case of the Cubic model, and its hysteresis was determined by the following formula.
The basic equation is

\[ \epsilon_s - \epsilon_o = f_y / E_s (1 + | f_y / f_{ch}|^{-1}) \]  
\[ E_s^\text{mag} = df_y / d\epsilon_s = E_s / (1 + r | f_y / f_{ch}|^{-1}) \]

where \( f_{ch} \), and \( r \) are defined by

\[ f_{ch} = f_y (0.744 / \ln(1+1000\epsilon_p)) - (0.071 / (1. - \exp(1000\epsilon_p))) \]  
\[ r = 2.2 / \ln (n+1) - (0.469 / \exp(n) - 1) + 3.04, \, n \text{ even} \]  
\[ r = 4.49 / \ln (n+1) - (6.03 / \exp(n) - 1) + 0.297, \, n \text{ odd} \]

where \( \epsilon_p \) is the accumulated plastic strain and \( n \) is the current cycle number.

Again, a degrading tangent modulus can be used as a function of plastic straining. Because of the implicit formulation of the equation, and iterative solution is needed for the determination of the stress given the strain. The modified Regula Falsi method is used with extremely fast convergence results.

For biaxial bending behavior, such an iterative solution scheme makes it costly for use in some forms of cyclic analysis. Hence an explicit expression with fewer parameters is also provided as proposed by Menegotto and Pinto.

Figure 3.4.4 shows an example of the stress-strain history of reinforcing bar using the Remberg-Osgood model.
4. Verification of Analysis Through Comparison by Experiment

4.1 Uniaxial Bending

(1) Results of Experiment

Relationships between bending moment and curvature of each specimen (M-\( \phi \)) under the condition where a uniaxially reversed bending moment is applied under a constant axial force, are shown in Figures 4.1.1, 2, 3 and 4.

Here, the U1 specimen (steel ratio: 0.95%, compressive axial force: 80 kgf/cm²) is modeling the tower members of cable-stayed bridge. The U2 specimen (steel ratio: 0.95%, compressive axial force: 7 kgf/cm²) is the one assuming conventional bridge piers, which has an axial force far lower than that for the U1 specimen. The same axial force as that for the U1 specimen was applied to the U3 specimen (steel ratio: 1.90%, compressive axial force: 80 kgf/cm²) and the U4 specimen (steel ratio: 2.85%, compressive axial force: 80 kgf/cm²) but their steel ratio is larger than that of the U1 specimen, in particular the U4 specimen assumes the role of heavily reinforced RC columns, such as building columns.

Among these specimens, however, the test on the U3 specimen encountered a problem when the load reached the maximum value, which meant the data thereafter could not be obtained.

Characteristic features in the relationships between the bending moment and curvature of members are as described below.

<1> In the U2 specimen, which has an axial force smaller than the U1 as an object of comparison, the maximum load was about 180 kip-in and no decrease was found in the envelope of the M-\( \phi \) relationships within the range of the test.

The form of hysteresis except that in the small curvature ranges was like a rhomb.

<2> In the U1 specimen, the maximum load was about 400 kip-in and the envelope of the M-\( \phi \) relationships showed a decrease in load at a curvature of about \( 2 \times 10^{-2} \) 1/in. According to observation of the specimen during the test, this point approximately coincided with the time buckling of reinforcing bars and spalling of the cover concrete occurred. Each hysteresis showed a form having a constriction in the vicinity of the coordinate origin.

The cause of the constriction in hysteresis is considered to be that the residual elongation of reinforcing bars is forced to be reduced by the action of compressive axial force when the bending moment is decreased. Further, if the bending moment becomes lower than the decompression moment created by axial force, the gradient of the M-\( \phi \) relationships increases providing a greater rigidity for the member.
In the U4 specimen, which has a steel ratio three times larger than that of the U1 specimen, the maximum load of about 550 kip-in was larger than that of the U1 specimen. The curvature at the point when the bending moment of the M-Ø envelope begins to decrease was almost the same as that of the U1 specimen. In the form of hysteresis, the constriction was slightly mitigated as compared to the case of the U1 specimen.

The reason constriction in the hysteresis was mitigated is considered to be that the amount of reduction of elongation of reinforcing bars caused by the axial force was smaller due to the larger number of reinforcing bars.

In the U3 specimen, which has a steel ratio intermediate between the U1 and U4 specimens, the value of the maximum load and the form of hysteresis within the range of test, also seemed to be intermediate between the U1 and U4 specimens (Fig. 4.1.5).

(2) Comparison Between Analysis and Experiment

The analytically obtained M-Ø relationships of each specimen are shown in Figures 4.1.6 - 8, 13 - 15, 20 - 22 and 4.1.27 - 29.

The analytically MØ relationships and those obtained by experiment are compared for each specimen and their features described as follows:

All analytical simulations were performed by such methods as the curvature histories of test result being given as input data, after which bending moment histories were calculated.

a) U2 specimen (assuming conventional bridge piers)

On either of the Bilinear model, Cubic model, or R/O model, the result of analysis represents the characteristics of the M-Ø relationships in the experiment.

However, there are the following differences when looking into details (Table 4.1.1):

In the case of the Bilinear model, the analytical value of the maximum load is near to the test value. The form of envelope has a steeper gradient than the test result. Hysteresis represents the test results.

In the case of the Cubic model, the magnitude of the maximum load and the form of the envelope represent the test results well. However, the area enclosed with one loop of hysteresis is larger than the test result.

In the case of the R/O model, the analytical value of the maximum load is smaller than the test value. On the envelope, the load decreased in the middle as the curvature increased. Hysteresis expresses the test results well.
In the case of the R/O model, the decrease in load in the middle of the envelope is considered to be due to a feature of the R/O model, whereby a decrease in stress occurs in the middle of the envelope for the stress-strain model of the reinforcing bar when the reversed strain is imposed.

In the case of the R/O model and Cubic model, the result of analysis of hysteresis shows a phenomenon of the bending moment increasing discontinuously as compared with the test result when the curvature "φ" increases. This occurred when the strain of steel crossed zero strain toward the compression side, so it is considered due to the sudden move of the centroid of the compression zone towards the outside when the crack closed. In other words, although an ideal facial touch of concrete is assumed in the analysis, the touched area gradually increases depending on the state of engagement of aggregate in the actual case, so the centroid of the compression zone moves smoothly. This difference seemed to have been reflected in the results. The reason this phenomenon is mitigated in the Bilinear model is considered to be that the stress of reinforcing bars can become large in this model, consequently the bending moment can increase gradually as the curvature increases, while it cannot become larger than a certain limit in the former two models.

In the case of this specimen, the axial force is small, and, moreover, reinforcing bars are allocated symmetrically on the tensile side and compression side, so the resistance component of the concrete against the bending moment is small, as shown in Figure 4.1.12.

b) U1 specimen (assuming tower members)

Generally speaking, the analysis on either model gives a good approximation of the M-φ relationship in the test.

However the maximum bending moment in analysis is almost the same as the experimental value on the Cubic model, but it is smaller than the experimental value on the R/O model (Table 4.1.2).

Further, on either model, the analysis gives a smaller curvature at the maximum bending moment and a slower decrease in load after the maximum bending moment as compared with the experimental value.

Among the analytical results, that of the Cubic model indicates a larger area enclosed within one loop of hysteresis than in the case of other models.

It was observed in the process of loading in the test, that the steep decrease in load after maximum load is due to spalling of the concrete cover and buckling of reinforcing bars. In the analysis at this time, the forming of buckling of compressed...
reinforcing bars into a model has not yet been accomplished, so it is considered that the difference in the envelope after maximum load between the analytical and test results is attributable to this fact.

c) U4 specimen (assuming columns with a large amount of reinforcement)
<1> Generally speaking, the analysis on either model gives a good approximation of the $M-\phi$ relationship in the test model.

<2> As for maximum load, however, although the analytical value on the Bilinear model and R/O model is almost the same as the test result, the analysis on the Cubic model is larger than the test result. Further, as in the case of the U1 specimen, any result of analysis shows a slower decrease in load after the maximum load than that in the test result. On either model, the analytical hysteresis gives a good approximation of the test result. (Table 4.1.3)

d) U3 specimen (intermediate between U1 and U3 specimens)
<1> Within the range where test values have been obtained, characteristics of the relationships between the analytical result and test result seem to be intermediate between those of the U1 and U4 specimens.

e) Summary of uniaxial bending
<1> Envelope and the form of hysteresis

From the result of comparison between the analytical $M-\phi$ relationship and the experimental relationship as described above, the following conclusion can be obtained with the exception of differences from the experimental relationship in the magnitude of maximum load and the degree of decrease in load on the envelope after maximum load (these will be discussed later). That is, each model has both merits and demerits when used for the analysis of RC columns with small axial force such as conventional bridge piers, so it may be necessary to use the models suitably according to the object on which attention must be paid, such as an envelope or hysteresis, to obtain an accurate analytical value.

In the case of tower members and building columns subjected large axial force, the $M-\phi$ relationships seem to be simulated in a good approximation by any model. If analytical simulations are conducted for the purpose of investigating the effect of the magnitude of axial force or steel ratio on the $M-\phi$ relationships, they will offer good information on the characteristics of each specimen as obtained by the experiment on either model.
simple as an analytical model for reinforcing bars, but it enables simulation of the M-\(\phi\) relationships of a member only marginally inferior to that of other models (however, when the load sharing of concrete is relatively small as is the case under a small axial force, the characteristics of the reinforcing bar model largely affects the M-\(\phi\) relationships, so care should be taken.)

<2> Maximum load

Even when the same analytical model is used, the value of maximum load may be larger than, smaller than, or approximately equal to the experimental value according to the above discussion. To clarify this phenomenon, the relationship between the ratio of analytical maximum load to test result and magnitude of axial force or steel ratio are plotted for each models as shown in Figure 4.1.30. Although no specific trend can be found as to the magnitude of axial force, the effect of the steel ratio is clearly indicated, where the ratio of the analytical to empirical value of maximum load increases as the steel ratio increases.

Therefore, it is considered that the Cubic model provides a good approximation to the case of tower members which have a smaller steel ratio, while the R/0 model provides a good approximation to the case of columns with large amount of reinforcement.

<3> Decrease in load on the envelope after maximum load

The degree of decrease in load on the envelope after maximum load has not been simulated by the analysis, as described above. This is because the buckling of reinforcing bars as observed during the experiment is not considered in the analysis. In the application of analytical simulation, judgement will be necessary as to what range of analytical M-\(\phi\) relationship is reliable and what range is not reliable. Therefore, the moment when reinforcing bars buckle needs to be estimated. Generally speaking, however, estimation of the buckling stress of reinforcing bars based on the stiffness (tangential modules of elasticity) of the reinforcing bar and the distance between supports of the reinforcing bar is extremely difficult at the present stage. Therefore, buckling conditions for reinforcing bars for analysis are assumed as follows:

1. The strain of cover concrete exceeds the strain value corresponding to the maximum compressive stress, and
2. The tangential stiffness is near to "0" in the stress-strain relationship on reinforcing bars.
In consequence, this assumption is not applicable to the Bilinear model.
The analytical results of stress-strain relationships on each type of specimen are shown in the following figures:

Figure 4.1.9 - 11 (U2 specimen)
Figure 4.1.16 - 18 (U1 specimen)
Figure 4.1.23 - 25 (U4 specimen)

The point that satisfies the above conditions is marked in the figure, and the curvature on that position is quite similar to the curvature where the load decreased on the M-\( \phi \) envelope in the experiment. From this fact, it is possible to know the range of high accuracy on the M-\( \phi \) relationships obtained in the analysis, if the curvature where the reinforcing bar buckles is used as an index.
4.2 Biaxial Bending

(1) Results of Experiment

Relationships between bending moments or curvatures of two directions and the relationships between the bending moment and curvature on each specimen, on which a reversed bending moment was applied under a constant compressive axial force, are shown in the following figures:

B1 specimen: \( M_x - M_y \) (Figure 4.2.1), \( \phi_x - \phi_y \) (Figure 4.2.2)
\( M_x - \phi_x \) (Figure 4.2.3), \( M_y - \phi_y \) (Figure 4.2.4)

B2 specimen: \( M_x - M_y \) (Figure 4.2.5), \( \phi_x - \phi_y \) (Figure 4.2.6)
\( M_x - \phi_x \) (Figure 4.2.7), \( M_y - \phi_y \) (Figure 4.2.8)

B3 specimen: \( M_x - M_y \) (Figure 4.2.9), \( \phi_x - \phi_y \) (Figure 4.2.10)
\( M_x - \phi_x \) (Figure 4.2.11), \( M_y - \phi_y \) (Figure 4.2.12)

Here, the details of specimens and the magnitude of axial force applied are the same as those of the U1 specimen. On the B1 specimen, a reversed bending moment was applied at the 45° direction from the principal axis of the cross section. On the B2 specimen, a reversed bending moment was applied around the stronger principal axis of the cross section, while a constant bending moment was applied around the weaker principal axis. This simulates the case where a seismic load is applied in the longitudinal direction of the bridge to the A-shaped tower member which is subjected to a static load. On the B3 specimen, a random bending moment was applied simultaneously around both the stronger and weaker principal axis. Curvature hysteresis was applied as random loads whose magnitudes are proportional to the accelerations in two horizontal directions of the Elcentro earthquake and two types of waves (smaller and larger) were applied.

Characteristic features in the relationships between two moment components, two curvature components or the bending moment and curvature of members are as described below.

<1> B1 specimen (\( \theta_M = 45° \))

The \( \phi_x-\phi_y \) relationships are generally linear, the gradient (the ratio between \( \phi_x \) and \( \phi_y \)) increases as the reversed bending moment increases. As for the \( M-\phi \) relationships in each direction, although the envelope and hysteresis show the same forms as in the case of uniaxial bending (on the U1 specimen) in both the \( x \) and \( y \) directions, the magnitude of the maximum load around the stronger principal axis is as low as about 220 kip-in as compared with about 400 kip-in in the case of uniaxial bending. Further, the constriction as seen in the hysteresis on the U1 specimen seems to be mitigated.
The reason for the smaller maximum load is that it was significantly affected by the bending strength capacity around the weaker principal axis because the same magnitude of bending moment was applied in both directions.

Further, it is considered that the reason why the constriction on hysteresis was mitigated is as follows: because the cross section was subjected to biaxial bending, the distance between the neutral axis and the centroid of the cross section was smaller than that in the case of uniaxial bending. Therefore, the restoring force given by the compressive axial force acting on the centroid was also smaller and thus it mitigated the constriction.

<2> B2 specimen (My = constant)

When looking into the $\phi_x$-$\phi_y$ relationships, $\phi_y$ value increases in spite of the constant value of My. The envelope and the form of hysteresis in the $M_x$-$\phi_x$ relationships are similar to those of the U1 specimen. When looking into details, however, the maximum load is slightly smaller and the curvature corresponding to the maximum load is also smaller. Further, the constriction as seen in the hysteresis on the U1 specimen seems to be mitigated. It can be seen in the My-$\phi_y$ relationships that $\phi_y$ is increased in spite of the constant value of My.

The reason for the smaller maximum load and the curvature corresponding to the maximum load in the $M_x$-$\phi_x$ relationships, is that they are affected by My. Further, the reason for the mitigated constriction in the hysteresis is the same as that for the B1 specimen.

Incidentally, the reason why the My value in the My-$\phi_y$ relationships decreased at the final stage of loading is that the maintenance of the desired My value became difficult because of the failure progress in the specimen.

<3> B3 specimen (random)

The $M_x$-$My$ relationships show a form expanding to the whole areas of the X and Y axes with no clear convex portion, while the $\phi_x$-$\phi_y$ relationships show an envelope having clear convex portions. The $M_x$-$\phi_x$ relationships are moving within a relatively narrow range on the diagonal line on the x and y axes, and the decrease in load is observed in the third quadrant. The My-$\phi_y$ relationships have a moving range wider than that of the $M_x$-$\phi_x$ relationships, and a clear decrease in load is observed in the third quadrant.

The reason the convex and concave portions on the envelope of the $M_x$-$My$ relationships were mitigated as compared with the $\phi_x$-$\phi_y$ relationships, is that no load exceeding a certain value of bending moment i.e., yielding moment, could occur.

(2) Comparison Between Analysis and Experiment

Relationships between bending moments or curvatures of two directions, and the relationships between bending moment and curvature on each specimen obtained from analysis, are shown in the following figures. Here the analysis
was performed by the method where the bending moments are calculated on given curvatures.

B1 specimen: \( M_x - M_y \) (Figure 4.2.13,14), \( \phi x - \phi y \) (Figure 4.2.15,16)
\( M_x - \phi x \) (Figure 4.2.17,18), \( M_y - \phi y \) (Figure 4.2.19,20)

B2 specimen: \( M_x - M_y \) (Figure 4.2.21,22), \( \phi x - \phi y \) (Figure 4.2.23,24)
\( M_x - \phi x \) (Figure 4.2.25,26), \( M_y - \phi y \) (Figure 4.2.27,28)

B3 specimen: \( M_x - M_y \) (Figure 4.2.29,30), \( \phi x - \phi y \) (Figure 4.2.31,32)
\( M_x - \phi x \) (Figure 4.2.33,34), \( M_y - \phi y \) (Figure 4.2.35,36)

Analytical relationships and empirical relationships are compared on each specimen and the features are also described in the following.

a) B1 specimen (\( \theta_m = 45^\circ \))

<1> Although analytical, \( M_x - M_y \) relationship has a wider dispersion than the experimental one, analytical results represent experimental \( M-\phi \) relationships generally well on either model and show the features described for experimental ones.

<2> When attention is paid to the maximum value on the envelope of the \( M-\phi \) relationships, although it is almost the same as the experimental value on the Cubic model, it is smaller on the R/O model.

<3> In the analysis, there is a case where the envelope drops once in the middle. Generally speaking, the decrease in load in the analytical \( M-\phi \) envelope after maximum load is slower compared with the experiment.

As for the value of curvature at maximum load, the difference between the analytical value and experimental one is not so clear as compared with the case of uniaxial bending of the U1 specimen.

<4> The form of hysteresis as obtained by analysis is similar to the result of experiment.

<5> In the analysis, there is a point on the moment-curvature envelope where the \( M_x \) or \( M_y \) value once drops while the curvature in the direction of corresponding moment component is increasing. The reason why it occurs is considered as follows: those changes in the specimen such as spalling of concrete and/or buckling of reinforcing bars occurred during experiment and, consequently, the balance of bending moment and curvature on the X axis and Y axis changed. On the other hand, \( \phi x \) and \( \phi y \) as obtained from experiment were used in analysis as input data.
without simulating the buckling of reinforcing bars, etc.

In the $M$-$\phi$ relationships, the hysteresis loop in analysis becomes concave shape which is opposite to convex shape in experiment as the $\phi$ value increases after above mentioned event, and it is also considered to be caused by the same reason as above.

<6> The reason for a wider dispersion in the analytical $M_x$-$M_y$ curve is considered to be due to the larger curvature step in the analysis. The tendency that analytical maximum load and the degree of decrease in load after maximum load compared with the experiments are the same as those of uniaxial bending of the $U_1$ specimen.

As described in the section on uniaxial bending, the moment the reinforcement buckles was marked in analysis, and the curvature at that time is almost the same as the time when a large decrease in load occurs in the experiment.

b) B2 specimen ($M_y$ = constant)

<1> Although the analytical $M_x$-$M_y$ relationship has a wider dispersion than the experiment, it represents the experimental one fairly well on either model and shows the features as described for the experimental one.

<2> The results of comparison of maximum load, degree of decrease in load after maximum load, and the value of curvature at maximum load in the $M_x$-$\phi_x$ relationships with the experimental values, are almost the same as in the case of the $B_1$ specimen.

<3> When attention is paid to the range in $M_y$-$\phi_y$ relationships where the $\phi_y$ value is large, although the $M_y$ value in experiment was almost constant or in the decreasing tendency as the $\phi_y$ increased, the $M_y$ value in analysis increased gradually as the $\phi_y$ value increased.

<4> The form of hysteresis as obtained by analysis represents the results of the experiment fairly well.

<5> The reason for a wider dispersion in the analytical $M_x$-$M_y$ curve is considered to be the same as for the $B_1$ specimen. In addition, it is because $M_y$ was in the range of absolutely small values compared with the range of $M_x$. Further, the results of comparison with the experiment of maximum load and the degree of decrease in load after maximum load are almost the same as the case of the $B_1$ specimen.
The reason why the My value in analysis became large in the range of large \( \phi y \) values in the My-\( \phi y \) relationships is that the fracture progressing in analysis occurs behind the actual timing and the \( \phi y \) value in experiment was used for analysis as input data.

c) B3 specimen (random)

<1> Generally speaking, analytical Mx-My relationships and M-\( \phi \) relationships represent those in the experiment well, and show the features as described for experimental in the results.

<2> The same phenomena as described for the B1 specimen can be observed in the results of comparison of maximum load, degree of decrease in load after maximum load, and the value of curvature at maximum load with the experimental values in Mx-\( \phi x \) and My-\( \phi y \) relationships.

<3> The form of hysteresis as obtained by analysis represents the results of experiment well.

d) Summary of biaxial bending

<1> The relationships, between Mx and My, Mx and \( \phi x \) and My and \( \phi y \) can be simulated fairly well with either the Cubic model or the R/O model.

<2> When attention is paid to maximum load on the envelope of the M-\( \phi \) relationships, although it is almost the same as the experimental value on the Cubic model, it is smaller on the R/O model. (Figure 4.2.37)

<3> On either the Cubic model or the R/O model, the decrease in load after maximum load is slower than the results of experiment. This is because the behavior after buckling of reinforcement due to compression is not formed into a model in the analysis. The curvature at the time a steep decrease in load occurs in the experiment can also be estimated in the analysis by checking the state of reinforcement in stress-strain history.

<4> The features of analytical models as described above are the same as in the case of uniaxial bending.
5. Conclusion

For the purpose of application of fiber model analysis for simulating the relationships between the bending moment and curvature (M-Θ relationships) of the reinforced concrete tower members (longitudinal steel ratio of 0.95%, compressive axial force of 80 kg/cm²) of cable-stayed bridges under biaxial bending, suitability of fiber models were investigated by paying attention to the reinforcing bar model.

Biaxial bending tests were performed on the RC-column specimens having rectangular cross section and the simulation analysis were performed on these specimens by using fiber model.

As the consequence of the comparisons between analyses and experiments, the following facts have been clarified:

a. Uniaxial bending

<1> Whatever model, the Bilinear model, Cubic model, or Ramberg-Osgood (R/O) model, is used as the reinforcing bar model, can well simulate the M-Θ relationships and can well express the influence of the magnitude of the axial force and steel ratio on the M-Θ relationships.

<2> The degree of decrease in load after the maximum bending moment on the envelope of the M-Θ relationships is slower in analysis than in experiment.

Although the point where the load decreases steeply in experiment can be estimated on the Cubic model and R/O model, it is difficult to estimate on the Bilinear model.

<3> As far as the maximum bending moment is concerned, the Cubic model gives a higher accuracy of estimation as to tower members having a longitudinal steel ratio of 0.95%. On the other hand, the R/O model gives a higher accuracy of estimation in the case of columns of highrized building, which have a higher steel ratio.

b. Biaxial bending

Examination on biaxial bending was performed on the specimens having a longitudinal steel ratio of 0.95% using the Cubic model and R/O model.

<1> As the result, although limited in above range, it has been confirmed that the results of the above mentioned uniaxial bending applies to biaxial bending also.

<2> Influence of biaxial loading hysteresis on the M-Θ relationships can be expressed well on either model. However, the Cubic model gives a higher accuracy as to the magnitude of the maximum bending moment, so it is considered that the Cubic model is more suitable for analysis of the towers for cable-stayed bridges.
Table 2.1.1  Test arrangement

<table>
<thead>
<tr>
<th>No.</th>
<th>Dimens. section (mm)</th>
<th>reinf. ratio</th>
<th>Concrete strength (kgf/cm²)</th>
<th>Axial force (kgf/cm²)</th>
<th>Loading direction (deg)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>longi. (%)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>U 1</td>
<td>rectangle</td>
<td>0.95</td>
<td>443</td>
<td>80</td>
<td>( \theta_m = 0^\circ, \text{revers.} )</td>
</tr>
<tr>
<td>U 2</td>
<td></td>
<td>1.90</td>
<td>473</td>
<td>7</td>
<td></td>
</tr>
<tr>
<td>U 3</td>
<td></td>
<td>2.85</td>
<td>412</td>
<td>80</td>
<td>( \theta_m = 45^\circ, \text{revers.} )</td>
</tr>
<tr>
<td>U 4</td>
<td>160</td>
<td>0.16</td>
<td>414</td>
<td></td>
<td>( M_y = 0.57 \text{tm, revers.} )</td>
</tr>
<tr>
<td>B 1</td>
<td>( \times )</td>
<td></td>
<td>407</td>
<td></td>
<td>( \text{El-centro, 2cycle} )</td>
</tr>
<tr>
<td>B 2</td>
<td>250</td>
<td>0.95</td>
<td>426</td>
<td></td>
<td></td>
</tr>
<tr>
<td>B 3</td>
<td></td>
<td></td>
<td>430</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Fig. 2.2.1 Dimensions and reinforcement details of basic test specimen
Fig. 2.2.2 Mechanical properties obtained from material test (#2, deformed)

- Yield stress: 3030 kgf/cm²
- Break stress: 4785 kgf/cm²
- Elastic modulus: $1.5 \times 10^6$ kgf/cm²
- Yield strain: $2000 \times 10^{-6}$
- Strain hardening point: 1.92%
Table 2.2.1 Mix proportion of concrete

<table>
<thead>
<tr>
<th>Design strength</th>
<th>( f_{ck} = 400 \text{kgf/cm}^2 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water-cement ratio</td>
<td>( W/C = 49.5% )</td>
</tr>
<tr>
<td>Fine aggregate ratio</td>
<td>( s/a = 46% )</td>
</tr>
<tr>
<td>Air content</td>
<td>( A = 3% )</td>
</tr>
<tr>
<td>Water</td>
<td>158 kg/m³</td>
</tr>
<tr>
<td>Cement</td>
<td>319 kg/m³</td>
</tr>
<tr>
<td>High-early-strength Portland Cement</td>
<td></td>
</tr>
<tr>
<td>Fine aggregate</td>
<td>852 kg/m³</td>
</tr>
<tr>
<td>Specific gravity = 2.61</td>
<td></td>
</tr>
<tr>
<td>Maximum size = 5 mm</td>
<td></td>
</tr>
<tr>
<td>Absorption = 1.01%</td>
<td></td>
</tr>
<tr>
<td>F. M. = 2.41</td>
<td></td>
</tr>
<tr>
<td>Coarse aggregate</td>
<td>989 kg/m³</td>
</tr>
<tr>
<td>Specific gravity = 2.58</td>
<td></td>
</tr>
<tr>
<td>Maximum size = 10 mm</td>
<td></td>
</tr>
<tr>
<td>Admixture</td>
<td>798 cc</td>
</tr>
<tr>
<td>Pozoris No. 70</td>
<td></td>
</tr>
</tbody>
</table>
Table 2.2.2 Results of concrete cylinder tests

<table>
<thead>
<tr>
<th>No.</th>
<th>fc''</th>
<th>fr</th>
<th>fc''/fr</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>447</td>
<td>34.1</td>
<td>—</td>
</tr>
<tr>
<td>2</td>
<td>445</td>
<td>33.4</td>
<td>—</td>
</tr>
<tr>
<td>3</td>
<td>426</td>
<td>32.8</td>
<td>—</td>
</tr>
<tr>
<td>Av.</td>
<td>439</td>
<td>33.4</td>
<td>13.1</td>
</tr>
</tbody>
</table>
Fig. 2.2.3 Stress-strain relation obtained from concrete cylinder tests
Fig. 2.3.2 Schematic drawing of loading and measuring system
Fig. 2.3.4 Vertical section of the direction that an angle formed between section A and B is maximum
Fig. 2.3.5 The direction that an angle formed between section A and B, and the direction of curvature vector.
Fig. 3.2.1 Section idealization for analysis
Fig. 3.3.1 Basic concrete model used (after Sheikh and Uzumeri)
Fig. 3.3.2 Material model for unconfined concrete
Fig. 3.3.3 Material model for confined concrete
Table 3.3.1 Input values for analytical concrete models

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Unconfined</th>
<th>Confined</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum compressive stress : $f_c'$</td>
<td>443 kgf/cm$^2$ = 6.3 ksi</td>
<td></td>
</tr>
<tr>
<td>Corresponding strain : $\varepsilon_0$</td>
<td></td>
<td>0.002</td>
</tr>
<tr>
<td>Strain at end of horizontal plateau : $\varepsilon_1$</td>
<td></td>
<td>0.002</td>
</tr>
<tr>
<td>Degradation strain : $\varepsilon_{20c}$</td>
<td>0.0055</td>
<td>0.015</td>
</tr>
<tr>
<td>Corresponding degradation stress : $P_{tfc}$</td>
<td>0.00 ksi</td>
<td>1.26 ksi</td>
</tr>
<tr>
<td>Crushing strain : $\varepsilon_{cr}$</td>
<td></td>
<td>0.10</td>
</tr>
<tr>
<td>Rupture stress : $fr_p$</td>
<td>0.13$f_c'$ = 0.82 ksi</td>
<td></td>
</tr>
</tbody>
</table>
Fig. 3.3.4 Concrete model: unloading and reloading behaviour
Fig. 3.4.1 Stress-strain skeleton model for reinforcing bars
Fig. 3.4.2 Behaviour of the bilinear steel model
Fig. 3.4.3 Cubic steel model behaviour for various of $\alpha$
Fig. 3.4.4 Behaviour of the Ramberg-Osgood steel model

(a) Small strain cycles

(b) Large strain cycles
Fig. 4.1.1 Moment/curvature diagram for specimen UI (experiment): Rotation about X axis
Fig. 4.1.2 Moment/curvature diagram for specimen U2 (experiment)

: Rotation about X axis
Fig. 4.1.3 Moment/curvature diagram for specimen U3 (experiment).

- Rotation about X axis.
Fig. 4.1.4 Moment/curvature diagram for specimen U4 (experiment)

: Rotation about X axis
Fig. 4.1.5 Moment/curvature diagram for specimen U1, U3 and U4 (experiment): Rotation about X axis
Fig. 4.1.2  Moment/curvature diagram for specimen U2 (experiment) 
: Rotation about X axis 

Fig. 4.1.6  Moment/curvature diagram for specimen U2 (bilinear model) 
: Rotation about X axis
Fig. 4.1.7  Moment/curvature diagram for specimen U2 (cubic model)  
: Rotation about X axis

Fig. 4.1.8  Moment/curvature diagram for specimen U2 (r/o model)  
: Rotation about X axis
Fig. 4.1.9 Stress-strain history of reinforcement in specimen U2 (bilinear model)

Fig. 4.1.10 Stress-strain history of reinforcement in specimen U2 (cubic model)
Fig. 4.1.11 Stress-strain history of reinforcement in specimen U2 (r/o model)

Fig. 4.1.12 Stress-strain history of concrete in specimen U2 (r/o model)
Fig. 4.1.1  Moment/curvature diagram for specimen U1 (experiment)
: Rotation about X axis

Fig. 4.1.13 Moment/curvature diagram for specimen U1 (bilinear model)
: Rotation about X axis
Fig. 4.1.14 Moment/curvature diagram for specimen U1 (cubic model)  
: Rotation about X axis

Fig. 4.1.15 Moment/curvature diagram for specimen U1 (r/o model)  
: Rotation about X axis
Fig. 4.1.16 Stress-strain history of reinforcement in specimen U1 (bilinear model)

Fig. 4.1.17 Stress-strain history of reinforcement in specimen U1 (cubic model)
Fig. 4.1.18 Stress-strain history of reinforcement in specimen U1 (r/o model)

Fig. 4.1.19 Stress-strain history of concrete in specimen U1 (r/o model)
Fig. 4.1.4 Moment/curvature diagram for specimen U4 (experiment) : Rotation about X axis

Fig. 4.1.20 Moment/curvature diagram for specimen U4 (bilinear model) : Rotation about X axis
Fig. 4.1.21 Moment/curvature diagram for specimen U4 (cubic model):
Rotation about X axis

Fig. 4.1.22 Moment/curvature diagram for specimen U4 (r/o model):
Rotation about X axis
Fig. 4.1.23 Stress-strain history of reinforcement in specimen U4 (bilinear model)

Fig. 4.1.24 Stress-strain history of reinforcement in specimen U4 (cubic model)
Fig. 4.1.25 Stress-strain history of reinforcement in specimen U4 (r/o model)

Fig. 4.1.26 Stress-strain history of concrete in specimen U4 (r/o model)
Fig. 4.1.3  Moment/curvature diagram for specimen U3 (experiment)  
: Rotation about X axis

Fig. 4.1.27  Moment/curvature diagram for specimen U3 (bilinear model)  
: Rotation about X axis
Fig. 4.1.28 Moment/curvature diagram for specimen U3 (cubic model)
: Rotation about X axis

Fig. 4.1.29 Moment/curvature diagram for specimen U3 (r/o model)
: Rotation about X axis
Fig. 4.1.30 Ratio of analytical to experimental maximum moment
Table 4.1.1 Characteristics of analytical $M-\phi$ relation compared to experiment (specimen U2)

<table>
<thead>
<tr>
<th></th>
<th>bilinear model</th>
<th>cubic model</th>
<th>r/o model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum moment</td>
<td>Almost the same</td>
<td>Almost the same</td>
<td>Smaller</td>
</tr>
<tr>
<td>Envelope</td>
<td>Slope is a little sharper</td>
<td>Same</td>
<td>Dropping in the way</td>
</tr>
<tr>
<td>Hysteresis</td>
<td>Similar</td>
<td>Fatter</td>
<td>Almost the similar</td>
</tr>
<tr>
<td></td>
<td>Bilinear model</td>
<td>Cubic model</td>
<td>R/o model</td>
</tr>
<tr>
<td>--------------------------</td>
<td>----------------</td>
<td>-------------------</td>
<td>------------------</td>
</tr>
<tr>
<td><strong>Maximum moment</strong></td>
<td>Smaller</td>
<td>Almost the same</td>
<td>Smaller</td>
</tr>
<tr>
<td><strong>Envelope</strong></td>
<td>Curvature at the maximum moment is smaller.</td>
<td>Decrease in bearing force after the maximum moment is slower.</td>
<td></td>
</tr>
<tr>
<td><strong>Hysteresis</strong></td>
<td>Similar</td>
<td>Similar</td>
<td>Similar</td>
</tr>
<tr>
<td></td>
<td>bilinear model</td>
<td>cubic model</td>
<td>r/o model</td>
</tr>
<tr>
<td>---------------------------</td>
<td>----------------</td>
<td>-------------</td>
<td>-------------------</td>
</tr>
<tr>
<td>Maximum moment</td>
<td>Almost the same</td>
<td>Larger</td>
<td>Almost the same</td>
</tr>
<tr>
<td>Envelope</td>
<td>Decrease in bearing force after the maximum moment is slower or nothing.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hysteresis</td>
<td>Similar</td>
<td>Similar</td>
<td>Similar</td>
</tr>
</tbody>
</table>
Fig. 4.2.1 Biaxial hysteretic response of moment for specimen B1 (experiment)
Fig. 4.2.2 Biaxial histeretic response of curvature for specimen B1 (experiment)
Fig. 4.2.3  Moment/curvature diagram for specimen B1 (experiment)
: Rotation about X axis
Fig. 4.2.4 Moment/curvature diagram for specimen B1 (experiment)  
: Rotation about Y axis
Fig. 4.2.5 Biaxial hysteretic response of moment for specimen B2 (experiment)
Fig. 4.2.6 Biaxial hysteretic response of curvature for specimen B2 (experiment)
Fig. 4.2.7  Moment/curvature diagram for specimen B2 (experiment)

: Rotation about X axis
Fig. 4.2.8 Moment/curvature diagram for specimen B2 (experiment): Rotation about Y axis
Fig. 4.2.9 Biaxial hysteretic response of moment for specimen B3 (experiment)
Fig. 4.2.10 Biaxial histeretic response of curvature for specimen B3 (experiment)
Fig. 4.2.11 Moment/curvature diagram for specimen B3 (experiment)
: Rotation about X axis
Fig. 4.2.12 Moment/curvature diagram for specimen B3 (experiment):
Rotation about Y axis
Fig. 4.2.1 Biaxial histeretic response of moment for specimen B1 (experiment)

Fig. 4.2.13 Biaxial histeretic response of moment for specimen B1 (cubic model)
Fig. 4.2.14 Biaxial histeretic response of moment for specimen B1 (r/o model)
Fig. 4.2.2  Biaxial histeretic response of curvature for specimen B1 (experiment)

Fig. 4.2.15  Biaxial histeretic response of curvature for specimen B1 (cubic model)
Fig. 4.2.16 Biaxial histeretic response of curvature for specimen B1 (r/o model)
Fig. 4.2.3  Moment/curvature diagram for specimen B1 (experiment)
   : Rotation about X axis

Fig. 4.2.17  Moment/curvature diagram for specimen B1 (cubic model)
   : Rotation about X axis
Fig. 4.2.18 Moment/curvature diagram for specimen B1 (r/o model)

: Rotation about X axis
Fig. 4.2.4 Moment/curvature diagram for specimen B1 (experiment) : Rotation about Y axis

Fig. 4.2.19 Moment/curvature diagram for specimen B1 (cubic model) : Rotation about Y axis
Fig. 4.2.20 Moment/curvature diagram for specimen B1 (r/o model)
: Rotation about Y axis
Fig. 4.2.5 Biaxial histeretic response of moment for specimen B2 (experiment)

Fig. 4.2.21 Biaxial histeretic response of moment for specimen B2 (cubic model)
Fig. 4.2.22 Biaxial hysteretic response of moment for specimen B2 (r/o model)
Fig. 4.2.6 Biaxial histeretic response of curvature for specimen B2 (experiment)

Fig. 4.2.23 Biaxial histeretic response of curvature for specimen B2 (cubic model)
Fig. 4.2.24 Biaxial histeretic response of curvature for specimen B2 (r/o model)
Fig. 4.2.7 Moment/curvature diagram for specimen B2 (experiment)
: Rotation about X axis

Fig. 4.2.25 Moment/curvature diagram for specimen B2 (cubic model)
: Rotation about X axis
Fig. 4.2.26 Moment/curvature diagram for specimen B2 (r/o model) : Rotation about X axis
Fig. 4.2.8 Moment/curvature diagram for specimen B2 (experiment): Rotation about Y axis

Fig. 4.2.27 Moment/curvature diagram for specimen B2 (cubic model): Rotation about Y axis
Fig. 4.2.28 Moment/curvature diagram for specimen B2 (r/o model)

: Rotation about Y axis
Fig. 4.2.9 Biaxial histeretic response of moment for specimen B3 (experiment)

Fig. 4.2.29 Biaxial histeretic response of moment for specimen B3 (cubic model)
Fig. 4.2.30 Biaxial hysteretic response of moment for specimen B3 (r/o model)
Fig. 4.2.10 Biaxial hysteretic response of curvature for specimen B3 (experiment)

Fig. 4.2.31 Biaxial hysteretic response of curvature for specimen B3 (cubic model)
Fig. 4.2.32 Biaxial hysteretic response of curvature for specimen B3 (r/o model)
Fig. 4.2.11 Moment/curvature diagram for specimen B3 (experiment)  
: Rotation about X axis

Fig. 4.2.33 Moment/curvature diagram for specimen B1 (cubic model)  
: Rotation about X axis
Fig. 4.2.34 Moment/curvature diagram for specimen B1 (r/o model)
: Rotation about X axis
Fig. 4.2.12 Moment/curvature diagram for specimen B3 (experiment)  
: Rotation about Y axis

Fig. 4.2.35 Moment/curvature diagram for specimen B1 (cubic model)  
: Rotation about Y axis

- 110 -
Fig. 4.2.36 Moment/curvature diagram for specimen B1 (r/o model)
: Rotation about Y axis
Fig. 4.2.37 Ratio of analytical to experimental maximum moment
Table 4.2.1 Features of the analytical values as compared with the experimental values in the $M_\phi$ relationships, on the B1 specimen

<table>
<thead>
<tr>
<th>Feature</th>
<th>Cubic Model</th>
<th>R/O Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M_x - M_y$ relationships</td>
<td>Roughly simulates the linear relationships though the width of dispersion is large.</td>
<td></td>
</tr>
<tr>
<td>$M - \phi$ relationships</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Maximum moment</td>
<td>Almost the same</td>
<td>Smaller</td>
</tr>
<tr>
<td>Envelope</td>
<td>Decrease in load after the maximum load is slower.</td>
<td></td>
</tr>
<tr>
<td>Hysteresis</td>
<td>Similar</td>
<td>Similar</td>
</tr>
</tbody>
</table>
Table 4.2.2 Features of the analytical values as compared with the experimental values in the $M - \phi$ relationships, on the B2 specimen

<table>
<thead>
<tr>
<th></th>
<th>cubic model</th>
<th>r/o model</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M_x - M_y$ relationships</td>
<td>Roughly simulates the linear relationships though the width is large.</td>
<td></td>
</tr>
<tr>
<td>$M_x - \phi_x$ relationships</td>
<td>Maximum moment</td>
<td>Almost the same</td>
</tr>
<tr>
<td></td>
<td>Envelope</td>
<td>Decrease in load after the maximum load is slower.</td>
</tr>
<tr>
<td></td>
<td>Hysteresis</td>
<td>Similar</td>
</tr>
<tr>
<td>$M_y - \phi_y$ relationships</td>
<td>$M$ increases according to the increase in $\phi$ at the end.</td>
<td></td>
</tr>
</tbody>
</table>
Table 4.2.3 Features of the analytical values as compared with the experimental values in the $M-\phi$ relationships, on the B3 specimen

<table>
<thead>
<tr>
<th></th>
<th>cubic model</th>
<th>r/o model</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M_x - M_y$, relationships</td>
<td>Simulates</td>
<td>Simulates</td>
</tr>
<tr>
<td>$M - \phi$ relationships</td>
<td></td>
<td></td>
</tr>
<tr>
<td>. Maximum moment</td>
<td>Almost the same</td>
<td>Smaller</td>
</tr>
<tr>
<td>. Envelope</td>
<td>Decrease in load after the maximum load is slower.</td>
<td></td>
</tr>
<tr>
<td>. Hysteresis</td>
<td>Similar</td>
<td>Similar</td>
</tr>
</tbody>
</table>
Publication Plan

1. Proceedings of Japan Society of Civil Engineers (Concrete Engineering and Pavements)


3. Proceedings of Japan Concrete Institute, Vol.14, 1992