SPCFRAME-Computer Program for Nonlinear Segmental Analysis of Planar Prestressed Concrete Frames

Kang, Young-Jin

1989-02-01

REPORT NO.
UCB/SEMM-89/07

STRUCTURAL ENGINEERING,
MECHANICS AND MATERIALS

SPCFRAME - COMPUTER PROGRAM FOR NONLINEAR SEGMENTAL ANALYSIS OF PLANAR PRESTRESSED CONCRETE FRAMES

by
YOUNG-JIN KANG
Visiting Associate Research Engineer

Faculty Investigator: A. C. SCORDELIS

FEBRUARY 1989

DEPARTMENT OF CIVIL ENGINEERING
UNIVERSITY OF CALIFORNIA AT BERKELEY
BERKELEY, CALIFORNIA

Department of Civil Engineering
Structural Engineering Mechanics and Materials

Report No. UCB/SEMM-89/07

SPCFRAME - COMPUTER PROGRAM FOR NONLINEAR SEGMENTAL ANALYSIS OF PLANAR PRESTRESSED CONCRETE FRAMES

by
Young-Jin Kang
Visiting Associate Research Engineer

Faculty Investigator : A. C. Scordelis

College of Engineering
Office of Research Services
University of California
Berkeley, California

February 1989

BIBLIOGRAPHIC DATA SHEET	1. Report No.	2.	3. Recipient's Accession No.

4. Title and Subtitle

SPCFRAME - Computer Program for Nonlinear Segmental Analysis of Planar Prestressed Concrete Frames

5. Report Date
February, 1989

6.

7. Author(s)
Young-Jin Kang

8. Performing Organization Rept. No. UCB/SEMM-89/07

9. Performing Organization Name and Address

Department of Civil Engineering
760 Davis Hall
University of California
Berkeley, CA 94720

10. Project/Task/Work Unit No.

11. Contract/Grant No.

12. Sponsoring Organization Name and Address

Korean Ministry of Education, and
Byron L. and Elvira E. Nishkian Professorship,
University of California at Berkeley

13. Type of Report & Period Covered

14.

15. Supplementary Notes

This study was conducted while the author was visiting Berkeley in 1988 as Visiting Associate Research Engineer from Seoul National University, Seoul, Korea.

16. Abstracts

Computer program SPCFRAME, for the material and geometric nonlinear analysis of segmentally erected planar prestressed concrete frames, including the time-dependent effects due to load history, temperature history, creep, shrinkage, aging of concrete and relaxation of prestress, is developed. The program, based on the finite element method, is capable of predicting the response of these structures throughout their various stages of construction and service load history, as well as through the elastic, cracking, inelastic, and ultimate ranges. Changes in boundary conditions and loads, installation and removal of frame elements and prestressing tendons can be incorporated for modeling segmental construction operations. User's guide, illustrative examples, and programmer's guide are included.

17. Key Words and Document Analysis. 17a. Descriptors

Structural engineering; Computer program; Prestressed concrete; Nonlinear analysis; Material nonlinearities; Geometric nonlinearities; Creep; Shrinkage; Temperature; Load history; Cracking; Segmental erection; Finite elements; Frames; Bridges.

17b. Identifiers/Open-Ended Terms

17c. COSATI Field/Group

18. Availability Statement Release unlimited	19. Security Class (This Report) UNCLASSIFIED	21. No. of Pages 120
	20. Security Class (This Page UNCLASSIFIED	22. Price

Abstract

Computer program SPCFRAME, for the material and geometric nonlinear analysis of segmentally erected planar prestressed concrete frames including the time-dependent effects due to load history, temperature history, creep, shrinkage, aging of concrete and relaxation of prestress, is developed. The program, based on the finite element method, is capable of predicting the response of these structures throughout their various stages of construction and service load history as well as through elastic, cracking, inelastic and ultimate ranges. Changes in boundary conditions and loads, installation and removal of frame elements and prestressing tendons can be incorporated for modeling segmental construction operations. User's guide, illustrative examples and programmer's guide are included.

Acknowledgements

The author wishes to express his sincere gratitude to Professor A. C. Scordelis for his constant guidance, encouragement and support during the author's visit to Berkeley to conduct this research. Helpful discussions and assistance given by Chul Ho Kim, Chong Yoon and Fouad Kasti are appreciated. He also wishes to extend his appreciation to his wife and three children for their love and support.

Financial aid provided by the Korean Ministry of Education and Byron L. and Elvira E. Nishkian Professorship of the University of California are gratefully acknowledged.

Table of Contents

1. Introduction

1.1 Background

Research efforts on the nonlinear analysis of prestressed concrete structures have been going on since early 1970's at the University of California, Berkeley under the direction of Professor A. C. Scordelis. The results of these efforts have been summarized well by Professor Scordelis [1,2,3]. Several computer programs have emerged from these ongoing research efforts.

The program PCFRAME [4,5] was written by Kang in 1977. It has been used for the nonlinear material, geometric, and time-dependent analysis of reinforced and prestressed concrete plane frame structures. The program is capable of tracing the behavior of these structures at any time and under any load or environmental history through the elastic, cracking, inelastic, and ultimate ranges. One-dimensional frame elements, consisting of concrete and reinforcing steel layers and prestressing steel segments, and having three displacement degrees of freedom at each node are used.

In 1978 Van Zyl wrote the program SEGAN [6,7] for the analysis of curved segmentally erected prestressed concrete box girder bridges including time-dependent effects. Linearly elastic material properties are assumed. A skew-ended finite element with eight degrees of freedom at each of its end nodes is used. Operations used in segmental construction such as addition of segments, prestressing, changing of support conditions, application or removal of construction loads and prescribed displacements can be analyzed.

Van Greunen wrote the program NOPARC [8] in 1979 for the nonlinear geometric, material and time-dependent analysis of reinforced and prestressed concrete slabs and panels. Nine DOF triangular plate bending elements consisting of concrete and smeared reinforcing steel layers and straight prestressing tendon segments are used for analyzing the slabs and panels under any load or environmental history through various stages of service load and ultimate load ranges.

Mari wrote the program PCF3D [9] in 1984 for the nonlinear geometric, material and time-dependent analysis of three-dimensional reinforced and prestressed concrete frames. The program has similar capabilities to those of PCFRAME for three dimensional frames. The 3D frame elements have six degrees of freedom at each node, and consist of concrete and reinforcing steel filaments and straight prestressing tendon segments. The displacement control strategy is added to the load control strategy for the solution of nonlinear equilibrium equations.

In 1986 Ketchum developed the program SFRAME [10] which is capable of analyzing segmentally erected prestressed concrete bridges and other plane frame structures for time-dependent effects. Linearly elastic material properties are assumed. One-dimensional frame elements made up of concrete, mild steel and prestressing steel tendons are used. Changes in structural configuration or loading at any time can be incorporated. These include restraining and releasing boundary conditions, installing and removing frame elements, stressing, restressing and removing prestressing tendons, moving form travelers, and applying or removing nodal or element loads. Creep analysis is based on an improved numerical algorithm. The program utilizes the in-core and out-of-core database manager, and the free field input interpreter developed by Wilson and Hoit [11].

Most recently in 1986, Choudhury developed the program NAPBOX [12] for the nonlinear material analysis of curved nonprismatic prestressed concrete box girder bridges. A series of macro-elements each having three nodes on the longitudinal axis of the bridge, and consisting of concrete and reinforcing steel filaments and prestressing tendon segments, are used in the analysis. Each node has eight degrees of freedom including a longitudinal warping mode and a transverse distortional mode of the cross-section.

The programs developed so far are capable of either the nonlinear analysis of prestressed concrete structures erected in one step, or the time-dependent analysis of segmentally erected prestressed concrete structures with linearly elastic material properties. In order to assess the load carrying capacity of segmentally erected prestressed concrete structures in their various stages of construction and service life, it is desirable to have time-dependent nonlinear analysis capability of these structures.

The present program SPCFRAME has been developed combining the nonlinear analysis capabilities of the program PCFRAME and the segmental analysis capabilities of the program SFRAME by incorporating the nonlinear one-dimensional finite element of PCFRAME into SFRAME. A number of improved nonlinear analysis features have also been added. These will be discussed in detail in section 1.2.

Another program, called SPCF3D is being developed by Kasti at the University of California, Berkeley for the nonlinear time-dependent analysis of segmentally erected prestressed concrete three-dimensional frames, by combining the capabilities of the programs PCF3D and SFRAME. The capability for the analysis of composite three dimensional prestressed concrete frames are also included in the program. The program is almost completed.

1.2 Program Capabilities

The program SPCFRAME can be used for the time-dependent nonlinear analysis of segmentally erected reinforced and prestressed concrete planar frames due to any nodal or environmental load history through elastic, cracking, inelastic and ultimate ranges. Changes in support conditions and loads, intallation and removal of frames and tendons can be incorporated for modeling segmental construction operations. Since SPCFRAME is developed combining the two earlier programs PCFRAME and SFRAME, the capabilities of these programs are automatically incorporated. The capabilities of these three programs are listed and compared in Table 1.1. A number of improved nonlinear analysis features are added to those available in the earlier program PCFRAME. These include the following:

(1) The concrete stress-srain curve is modeled by a parabolic-linear-linear curve so that the confined concrete behavior can be properly modeled (see Fig. 2.2(a)).

(2) Tension stiffening effect can be included in the analysis. The effect is modeled by an unloading branch in the tensile stress-strain curve of the concrete (see Fig. 2.2(a)).

(3) Improved creep analysis model developed by Ketchum [10] is incorporated. The creep data can be generated automatically by the program utilizing ACI or CEB-FIP recommendations, or laboratory test data.

(4) Linear axial strain variation along the frame reference axis is assumed, instead of contant strain variation assumed in the program PCFRAME, by utilizing additional internal degree of freedom. This gives more accurate results for the nonlinear analysis in ultimate load ranges in which neutral axis of the cross-section can be shifted by the cracking and yielding of the material.

(5) Displacement control strategy for the solution of nonlinear equilibrium equations are added to the load control strategy so that either method can be selected by the user. The double step method developed by Powell and Simons [13] is incorporated. This method can be particularly useful for the post-buckling analysis and for the ultimate load analysis near the failure load where the slope of the load-deflection curve is extremely flat.

(6) The symmetric band solver SYMSOL in PCFRAME is replaced by the symmetric active column solver COLSOL. This should result in better performance in the iterative solution of nonlinear equilibrium equations.

(7) Most of the storage-consuming data such as the frame and tendon element data, and the concrete and reinforcing steel layer data are stored in a peripheral storage device. They are brought into the computer's memory only when necessary. These out-of-core files and dynamically allocated in-core arrays are efficiently managed by a database manager. As a result there are virtually no limits on the size of the problems that can be handled. Frames having more than a hundred nodes and elements can be solved even on a personal computer.

(8) Restart option has been added. This time-saving feature can be used conveniently in nonlinear analyses in which it is often necessary to examine intermediate results before deciding on the control values for the final analysis.

(9) Some convenience features have been added: free field input format; extensive data generation options; use of any consistent set of units.

Table 1.1 Comparison of Program Capabilities (1/4)

Category	PCFRAME	SFRAME	SPCFRAME
1. Types of structures			
Concrete structures	o Plain, reinforced and prestressed concrete frames in plane	o Plain, reinforced and prestressed concrete frames in plane	o Plain, reinforced and prestressed concrete frames in plane
Prestressed concrete structures	o Pretensioned frames o Post-tensioneed bonded frames o Post-tensioned unbonded frames	o Post-tensioneed bonded frames	o Pretensioned frames o Post-tensioneed bonded frames
2. Types of analysis			
Nonlinearities considered	o Material Geometry Linear Linear Linear Nonlinear Nonlinear Linear Nonlinear Nonlinear	o Material Geometry Linear Linear	o Material Geometry Linear Linear Linear Nonlinear Nonlinear Linear Nonlinear Nonlinear
Time dependent analysis	o Instantaneous analysis o Time step analysis	o Instantaneous analysis o Time step analysis	o Instantaneous analysis o Time step analysis
3. Construction operations			
Installation and removal		o Frames, tendons, traveling formworks	o Frames, tendons, traveling formworks
Boundary condition change		o Restraining and releasing	o Restraining and releasing
Types of operation		o Cantilever construction o Span-by-span construction o Incremental launching	o Cantilever construction o Span-by-span construction o Incremental launching
4. Types of loading			
External loading	o Joint loads o Imposed displacements	o Joint loads o Uniformly distributed loads o Imposed displacements	o Joint loads o Uniformly distributed loads o Imposed displacements
Internal loading	o Creep,shrinkage	o Creep, Shrinkage	o Creep, Shrinkage
Environmental loading	o Temperature change	o Temperature change	o Temperature change
Cross-sectional variation of shrinkage and temperature	o Arbitrary	o Linear	o Arbitrary

Table 1.1 Comparison of Program Capabilities (2/4)

Category	PCFRAME	SFRAME	SPCFRAME
5. Material property model			
Stress-strain curve model			
Concrete	o Parabolic-linear	o Linear	o Linear, Parabolic-linear, Parabolic-linear-linear
Reinforcing steel	o Linear, Bilinear	o Linear	o Linear, Bilinear
Prestressing steel	o Linear, Multilinear	o Linear	o Linear, Multilinear
Nonlinear effects			
Concrete	o Cracking, Yielding, Crushing o Load reversal		o Cracking, Yielding, Crushing o Load reversal, Tension stiffening
Reinforcing steel	o Yielding, Failure, Load reversal		o Yielding, Failure, Load reversal
Prestressing steel	o Yielding, Failure, Load reversal		o Yielding, Failure, Load reversal
Time dependent effects			
Concrete	o Creep, Shrinkage, Aging	o Creep, Shrinkage, Aging	o Creep, Shrinkage, Aging
Prestressing steel	o Relaxation	o Relaxation	o Relaxation
6. Concrete creep model			
Analysis model	o Stress Material Parameters Constant Constant	o Stress Material Parameters Constant Constant Linear Constant Linear Linear	o Stress Material Parameters Constant Constant Linear Constant Linear Linear
Creep data	o Laboratory, ACI	o Laboratory, ACI, CEB	o Laboratory, ACI, CEB
Nonlinear creep effects	o Creep increase in high stress levels o Temperature dependent creep compliance function		o Creep increase in high stress levels

Table 1.1 Comparison of Program Capabilities (3/4)

Category	PCFRAME	SFRAME	SPCFRAME
7. Element formulation			
Cross-section idealization	o Discrete concrete and steel layers	o Concrete and smeared reinforcing steel	o Discrete concrete and steel layers
Tendon idealization	o Linear segments within frame elements	o Linear segments within frame elements	o Linear segments within frame elements
Displacement functions	o Cubic deflection and linear curvature variation o Constant axial strain on reference axis	o Cubic deflection and linear curvature variation o Constant axial strain on reference axis	o Cubic deflection and linear curvature variation o Linear axial strain variation on ref. axis
Element degree of freedom	o End Rotation Axial Displacement 2 1	o End Rotation Axial Displacement 2 1	o End Rotation Axial Displacement 2 2 condensed into 1
Integration for stiffness and internal resisting loads			
Over cross-section	o Layer integration	o Closed form	o Layer integration
Over element length	o 3-point Gaussian quadrature	o Closed form	o 3-point Gaussian quadrature
State determination	o Path independent	o Closed form	o Path independent
8. Nonlinear analysis strategy			
Instantaneous analysis	o Load control with multiple load steps	o Load control with single load step	o Load control with multiple load steps o Displ. control with multiple displ. steps
Time step analysis	o Load control with multiple load steps and multiple time steps	o Load control with single load step and multiple time steps	o Load control with single load step and multiple time steps
Iteration scheme	o Constant stiffness iteration o Variable stiffness iteration	o Constant stiffness iteration	o Constant stiffness iteration o Variable stiffness iteration
Convergence criteria	o Displacement ratio o Ceiling on maximum unbalanced load	o Unbalanced load norms o Stress norms o Stress difference	o Displacement ratio o Ceiling on maximum unbalanced load o Stress ratio
9. Equation solver	o SYMSOL Symmetric band solver	o COLSOL Symmetric active column solver	o COLSOL Symmetric active column solver

Table 1.1 Comparison of Program Capabilities (4/4)

Category	PCFRAME	SFRAME	SPCFRAME
10. Input format			
Data input format	o Fixed format	o Free field format	o Free field format
Execution sequence	o Fixed	o Controlled by input Command lines	o Controlled by input Command lines
11. Automatic data generation			
Nodes, elements & loading		o Generated	o Generated
Tendon geometry		o Parabolic tendons generated	o Parabolic tendons generated
12. Units	o Any consistent units for lab creep data o Pounds-Inches for ACI creep data	o Pounds-Inches	o SI, Metric, U.S. conventional unit o Any other consistent unit
13. Program organization			
Modularity	o Low	o High	o High
No. of subprograms	o 9	o 123	o 130
No. of instruction lines	o 2,000	o 7,700	o 9,600
Core storage allocation	o Static	o Dynamic	o Dynamic
Data base management			
Core memory		o Manages incore arrays	o Manages incore arrays
Direct access files		o Frame, Tendon	o Frame, Tendon, Concrete & Steel layer
Sequencial files		o Camber, Incore arrays	o Camber, Incore arrays
14. Program execution mode	o Batch o Terminal	o Batch o Terminal o Restart	o Batch o Terminal o Restart

1.3 Objective and Scope

The purpose of this report is the description of the computer program SPCFRAME for the time-dependent nonlinear analysis of segmentally erected prestressed concrete planar frames, for the users and programmers who intend to modify the program to suit their own requirements.

Chapter 1 describes the background to the development of the program, and the comparison of program capabilities.

The user's guide, which constitutes the main body of this report, is given in chapter 2. It consists of the descriptions of the outline of the program, structural modeling, preparation of input data, command summary, and the detailed description of each command syntax.

An illustrative example is given in chapter 3. The structural modeling, preparation of input data, and the interpretation of output for a three-span continuous prestressed concrete box girder bridge are described in detail. Input data listings of this bridge and two more examples are given. The last example simulates the segmental erection sequence of a continuous post-tensioned box girder bridge built by the cantilever construction method. An edited output of the first example is printed.

The programmer's guide constitutes chapter 4. System implementation notes, program organization, detailed description of incore arrays, out-of-core file arrays and labeled common variables are given. This guide is provided for those who intend to modify the program to suit their own requirements.

This report constitutes the first part of the study for the time-dependent nonlinear analysis of segmentally erected prestressed concrete planar frames. The second report, which will describe the theoretical developments of this study along with detailed descriptions of illustrative examples, will be published before the end of this year.

1.4 Disclaimer

Considerable effort has gone into the development and testing of this program. It can analyze particular classes of structures based on the conditions and assumptions described in this report. The program should be used only under these conditions and assumptions. The input preparation and output interpretation must be handled by an appropriately qualified person.

Although the program has been extensively tested, no warranty is made regarding the accuracy and reliability of the program, and the author assumes no responsibility in this respect.

2. User's Guide

2.1 Outline of the Program

The computer program SPCFRAME has been developed as a tool for the linear or nonlinear material, geometric and time-dependent analysis of plain, reinforced and prestressed concrete plane frame structures which are erected in one step or in multiple steps segmentally. The program can trace the time-dependent response of these structures through various stages of construction and under any load or environmental history through the elastic, cracking, inelastic and ultimate ranges.

For the quasi-static time-dependent analysis, the time domain is divided into a discrete number of intervals, and a step-forward integration is performed in which increments of displacements, strains and stresses are successively added to the previous totals as the solution progresses in the time domain. Changes in the plane frame structural configuration, loading, prestressing or support conditions are accounted for during each time step. At each time step, a direct stiffness analysis based on the displacement method is performed in the space domain in which the equilibrium equations to be solved are necessarily nonlinear due to various material and geometric nonlinearities considered. The nonlinear equilibrium equations are solved by the Newton-Raphson iteration with unbalanced load corrections for load or displacement increments.

The plane frame is modeled as an assemblage of straight one-dimensional frame elements having three degrees of displacement freedoms at each node, and consisting of concrete, reinforcing steel and prestressing tendon segments. The cross-section which is prismatic along element axis and has an axis of symmetry, consists of a discrete number of concrete and reinforcing steel layers. Prestressing steel tendons are modeled as an assemblage of multilinear prestressing steel segments each of which spans a frame element. Each concrete or reinforcing steel layer or prestressing steel segment is assumed to be in a state of uniaxial stress defined by given nonlinear stress-strain relationships, including load reversal. In addition, time-dependent relationships for for creep, shrinkage, aging and temperature history for the concrete and relaxation in the prestressing steel are given.

Segmental construction methods are implemented by providing the capability to change the configuration of the structure at any time step of the solution. Possible changes in structuiral configuration and loading at any time include restraining and releasing boundary conditions, installing and removing frame elements, stressing, restressing and removing prestressing tendons, moving form travelers, and applying and removing nodal or element loads. Any statically feasible time-dependent construction sequence can be considered. Common segmental bridge construction methods such as free cantilever, span-by-span and incremental launching construction methods can be analyzed.

Ultimate load analysis at any particular stage of construction and time can be carried out by either the load control or the displacement control strategy. The displacement control strategy can be particularly useful in the post-buckling analysis or the ultimate load analysis near the failure load where the slope of the load-deflection curve is extremely flat.

Concrete creep analysis is based on an integral formulation in which concrete is assumed to be an aging viscoelastic material and a special exponential form of specific creep compliance function is used as the degenerate kernel of the superposition integral for creep. The special exponential form enables one to avoid the storage of entire stress history except for the stress history at the last time step to compute the creep strain increment. In the analysis three options are given regarding the assumption in the variations of the stress and the material parameters over the length of a time step: (1) constant stress and constant material parameters; (2) linear stress variation and constant material parameters; (3) linear variations of stress and material parameters. Options (2) and (3) yield more accurate results in general, especially for relaxation type problems. Empirical creep data based on ACI recommendations, CEB-FIP recommendations, or laboratory creep data can be incorporated in the analysis.

The solution at a particular stage of loading and time consists of nodal displacements, corresponding total nodal loads, reactions at restrained nodes and unbalanced loads, frame element total forces and moments with contributions from concrete layers, reinforcing steel layers, prestressing tendons and travelers, and stresses, strains, material state code numbers of concrete and reinforcing steel layers and prestressing tendon segments.

2.2 Structural Modeling

A plane frame structure is modeled as an assemblage of plane frame elements in the global X- Y- plane interconnected at nodal points (Fig. 2.1(a)). Global displacement degrees of freedom of the nodes consist of X- and Y- translations and Z-axis rotations. The boundary condition at each nodal degree of freedom may be specified as unrestrained or rigidly restrained to zero displacement, and may be changed at any solution step.

A frame element in its local coordinate system with resultant end forces and moments are shown in Fig. 2.1(b). Local x-axis is defined by its two end nodes, i and j. The cross-section has an axis of symmetry and is prismatic over its element length. Local y-axis coincides with the axis of symmetry. The origin of the y-axis does not have to coincide with the centroid of the cross-section although it is preferrable. The cross-section consists of a discrete number of concrete and mild steel layers as shown in Fig. 2.1(c). The layers are defined by their areas and local y-coordinates. Each element may be installed into and susequently removed from the structure at any solution step.

Pre-tensioned and post-tensioned prestressing steel tendons with arbitrary tendon profiles are modeled with a number of piecewise linear tendon segments defined by tendon points (Fig. 2.3(a)). Each tendon point is associated with a node, to which its displacements are rigidly constrained. Tendon point global coordinates may be input directly or may be generated automatically using a parametric generation scheme (Fig. 2.3(b)). Initial tendon force computation includes the effects of instantaneous prestressing force losses due to friction and anchorage slip. The prestressing tendons may be stressed, restressed or removed at any stage of the solution.

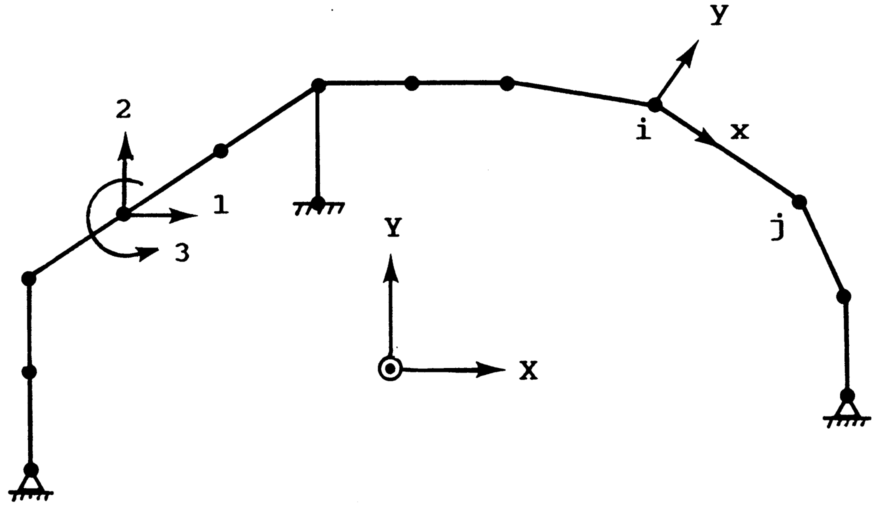
Traveling formworks are modeled with special frame elements which may be moved around the structure. They consist of linearly elastic material and are not subject to time-dependent strains.

Instantaneous stress-strain relations assumed in the analysis are shown in Fig. 2.2(a), (b) and (c), for the concrete, reinforcing steel and prestressing steel, respectively. Material state code numbers used in the analysis are also shown in the figure. The code numbers are explained in detail in section 3.2 (page 76). The effects of load reversal can be analyzed with these models. Both confined and unconfined concrete can be modeled, and the tension stiffening effect can be included in the analysis.

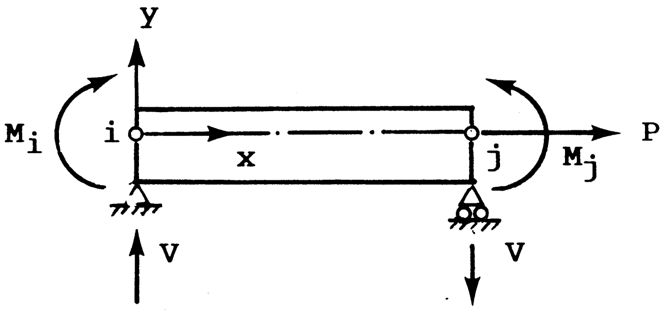
Time-dependent concrete properties are computed in two stages. The normalized values of the compressive strength, tensile strength, intial elastic moduli, shrinkage strain, and creep coefficients at each solution time step are first computed assuming the unit values of the cylinder strenth, the ultimate creep and the ultimate shrinkage. The actual values for the particular concrete properties at a specific solution time are computed later at each time step.

Possible structure loadings include internally computed frame element dead loads, concentrated loads on the nodes, imposed displacements on restrained degrees of freedom of the nodes, uniformly distributed global X- and Y- direction loads and moments along the axis on frame elements, and temperature changes in the elements which can vary arbitrarily over the depth of the cross-section. Arbitrary variation of shrinkage over the depth of the cross-section can also be incorporated.

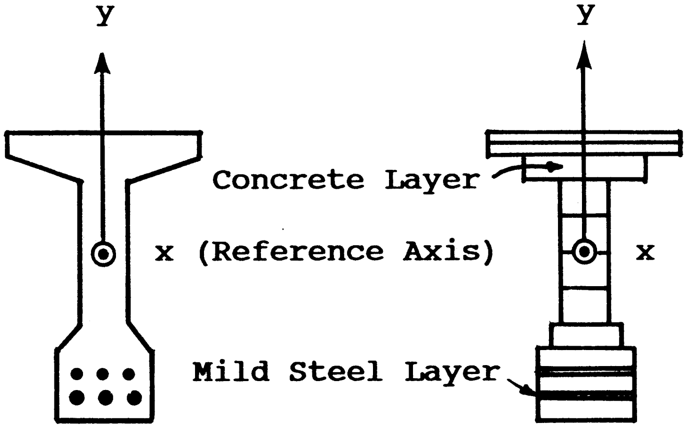
All nodes, frame elements and prestressing tendons which will ever exist in the structure are input to the program in a mesh input phase prior to the time-dependent analysis. These elements plus additional travelers, are tagged for installation or removal during additional structure configuration change phase of the program execution. At the beginning of the analysis, no elements are considered to be installed. Only when a node, element or tendon is "defined" as described below is it included in the analysis. An element is defined if it has been installed into the structure and has not yet been removed, otherwise the element is not defined. A node is defined if at least one defined frame or traveler element is connected to it, otherwise the node is undefined. The structure configuration change commands affecting these changes may be issued prior to any solution step.



(a) Structure and Coordinates

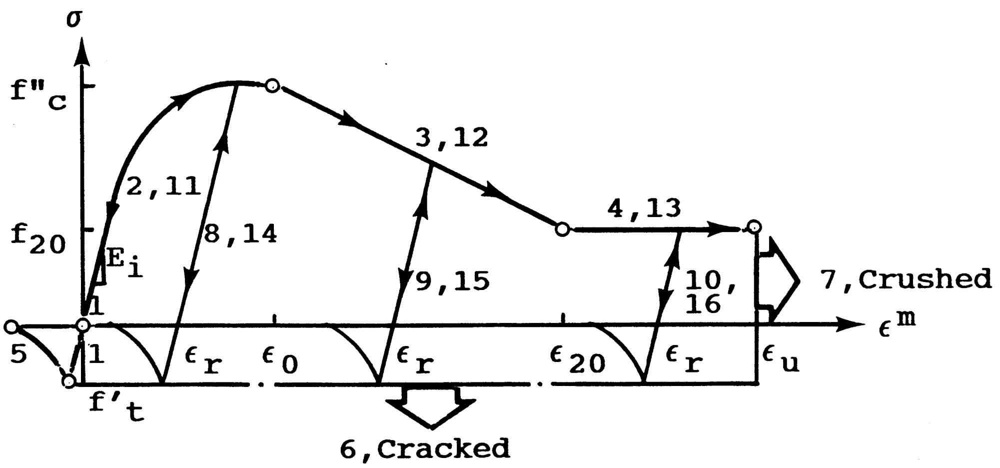


(b) Element

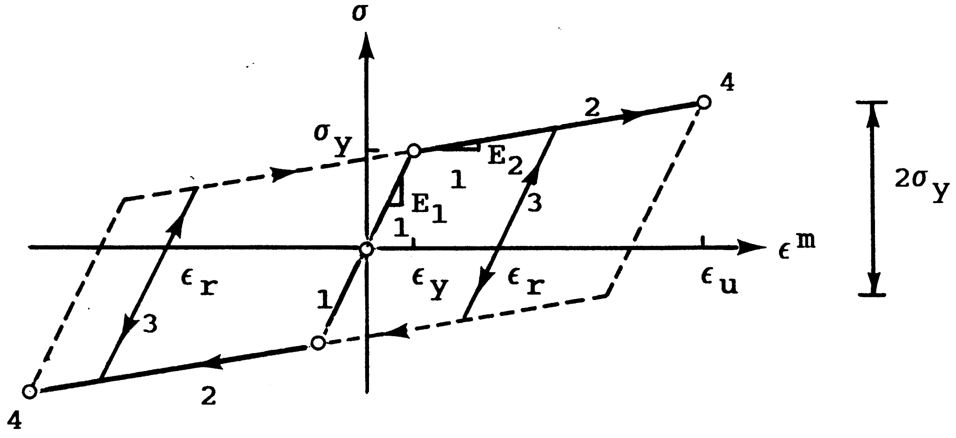


(c) Cross Section Idealization

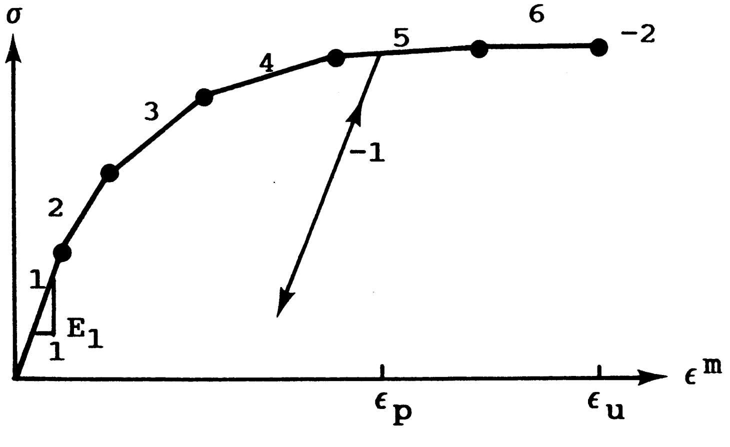
Fig. 2.1 Geometry of Concrete Plane Frames



(a) Concrete

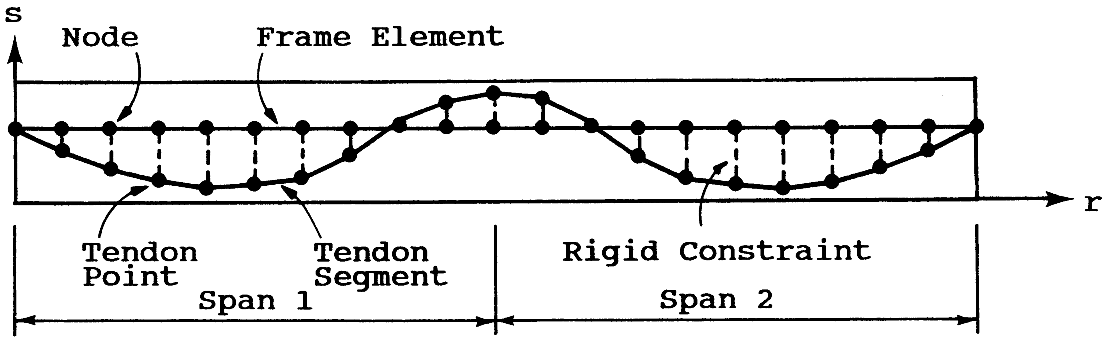


(b) Reinforcing Steel

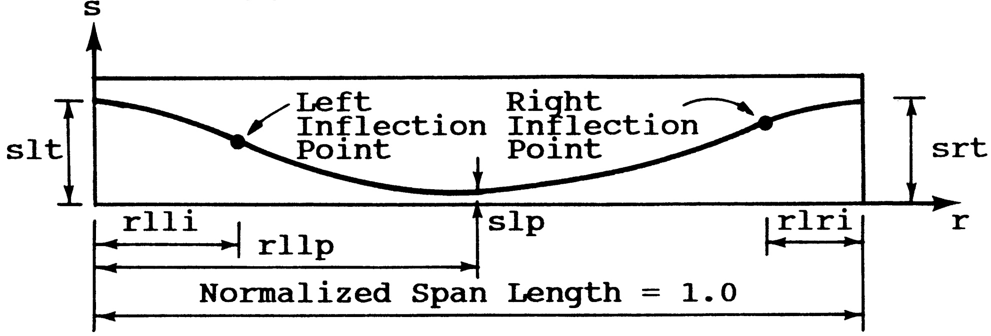


(c) Prestressing Steel

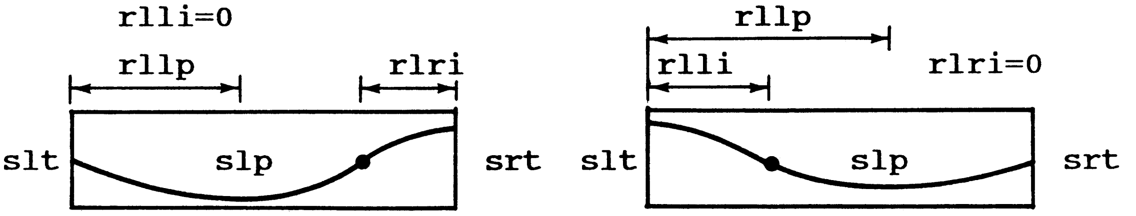
Fig. 2.2 Stress-strain Curves and Material State Code Numbers



(a) Tendon Discretization

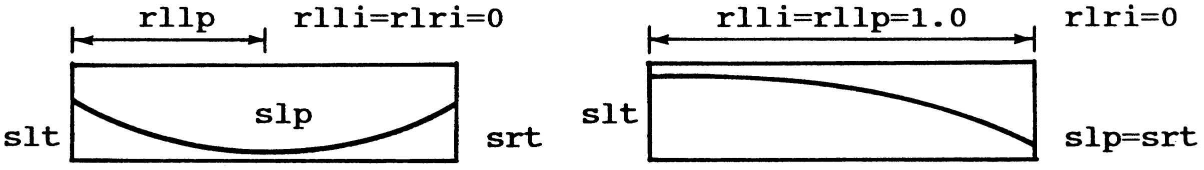


(b) Tendon Geometry Generation Parameters



Left Span of Two Span Bridge

Right Span of Two Span Bridge



Typical Simple Span

Typical Cantilever Span

(c) Tendon Geometry Generation Examples

Fig. 2.3 Tendon Discretization and Tendon Geometry Generation

2.3 Preparation of Input Data

Input data are provided to the program from a terminal or from a deck of cards or from an input file of card images, depending on the computer system used. All input takes the form of Commands consisting of one or more key-words, followed by optional numerical data required for the execution of the command. The actions and the data requirements of the commands are summarized in section 2.4 and described in detail in section 2.5 of this chapter. Only the first six characters of the key-words are recognized by the program. Character data can be input in either lowercase or uppercase letters. All letters are translated to capital letters in the program before they are interpreted. All numerical data are entered in the following free field form:

```
n1,n2,n3,...   A=a1,a2,a3,...   B=b1,b2,b3,...
```

where ni, ai and bi represent input data and the character pairs A= and B= are identifiers, specified in the input manual, for the data list which follows. Items in a data list must be separated by a single comma or one or more blanks. If a numerical data list without identification, such as n1,n2,n3,... above, is required it must be located as the first data list on the line. A data list of the form B=b1,b2,b3,... may be in any order or location on the line.

Simple arithmetical statements are possible when entering floating point real numbers. This can clarify the meanings of some data. For example the following forms of data can be entered:

```
E=29600*144  C=200*12+35, 400/12, 10+20/5-2
```

These arithmetical statements are evaluated from left to right without operator hierarchy. The statement 10+20/5-2 is evaluated as ((10+20)/5)-2. Blank spaces within arithmetical statements are not allowed.

The command interpreter in the program recognizes several special characters used to delimit multiple command lines on one physical line, continuation of a command line on the next physical line, and comments to be ignored by the command interpreter.

The exclamation point "!" is a special character used to delimit several command lines provided on one physical line. The data to the right of an exclamation point are not considered to be part of the current command line, and are instead considered as the next command line. Any number of command lines may be input on one physical line by separating them with this character.

The backslash "\" is a special character used for continuation of a command on the next physical line. All data to the right of the backslash are ignored, and the following input line is interpreted as a continuation of the first line. This option allows a maximum of 160 characters to be entered as one line of data.

The semicolon ";" is a special character used to delimit comments in the input stream. All characters to the right of the semicolon up to the end of the line are ignored by the program. If the semicolon is located in the first column in a command line the entire command line is ignored. An exclamation point "!" to the right of a semicolon will still be interpreted as a new line delimiter, and the data to the right of the exclamation point will be interpreted as the next data line. Two semicolons ";;" in the first two columns of a physical line will delimit the entire line, including exclamation points, as a comment.

2.4 Command Summary

The following list of each command and its input data requirements are intended as a summary of SPCFRAME's command syntax and as a quick reference guide for the experienced user. Detailed descriptions of each command and the meaning of all data are provided in section 2.4 of this User's Guide.

```
START  R=rstrt
TITLE  N=nl
UNIT   U=funit-lunit  C=cfunit
CONCRETE PARAMETERS   N=ncpt   T=na,nt,nr   P=iprn
   n  M=model   T=nat,ntt,nrt   R=rt   G=igen   \
        A=a   B=b   C=c   D=d   E=e   F=f   T=t0   W=w   X=rt  ( for ACI )
        H=humd   A=area   P=prim   X=rt                          ( for CEB )
      A=agel   T=time(1),time(2),...,time(ntt)               ( for times )
      A=agel   E=emod   S=shrn   F=fpc   R=fpt   T=time(1),.,time(ntt)
      C=crep(1),crep(2),...,crep(ntt)                          ( for LAB )

MESH INPUT
NODES  N=nnod
   n   X=xord   Y=yord   C=xord,yord   S=scale   G=n1,n2,inc         !
SEQUENCE
   n1,n2,inc                                                          !
CONCRETE PROPERTIES  N=ncnc
   n  F=fpc   C=ultcrp   S=ultshr   W=weight   M=model   A=alfa   \
      O=r1    P=r2       T=type
      C=rc    U=epsu     E=eps20    R=r20      K=kts          ( for T=2 )
MILD STEEL PROPERTIES  N=nmsl
   n  E=Es1   T=Es2   Y=fsy   U=epsu   A=alfa
PRESTRESSING STEEL  N=npsl   P=npt
   n  C=crvf   W=wblf   F=fpy   R=rlxp   A=alfa
      F=fp(1,n),fp(2,n),...,fp(npt,n)
      E=ep(1,n),ep(2,n),...,ep(npt,n)
```

```
SECTION PROPERTIES  N=nsct  C=mcl  S=msl
   n  N=nc  C=ncl  S=nsl  K=kshr
      A=acl(1,n),acl(2,n),...,acl(ncl,n)
      Y=ycl(1,n),ycl(2,n),...,ycl(ncl,n)
      S=shk(1,n),shk(ncl,n)                           ( for K=2 )
      S=shk(1,n),shk(2,n),...,shk(ncl,n)              ( for K=3 )
      B=asl(1,n),asl(2,n),...,asl(nsl,n)
      Z=ysl(1,n),ysl(2,n),...,ysl(nsl,n)
      M=ssl(1,n),ssl(2,n),...,ssl(nsl,n)
TEMPERATURE PATTERNS  N=ntdp
   n  X=nx  K=ktmp
      T=tfac(1),tfac(ncl)                             ( for K=2 )
      T=tfac(1),tfac(2),...,tfac(ncl)                 ( for K=3 )
FRAME ELEMENTS  N=nfml
   n,ni,nj  X=nx  D=cd  F=if  G=n1,n2,...,n7                   !
TENDON GEOMETRY  N=ntdn  P=iprn
   n  S=nspn  M=matl  A=area
      ns  N=nnis  G=n1,n2,inc  B=x0,y0  E=x1,y1
          L=n(1),n(2),...,n(nnis)                     ( for G=0 )
          R=rlli,rllp,rlri  S=slt,slp,srt       ( TP generation )
          np  R=r(np)  S=s(np)                  ( direct input  )
TRAVELERS  N=ntrv
   n  E=emod  A=area  I=izz  W=wegt  N=nnit
MESH COMPLETE
```

```
SET  D=day  T=temp  G=gx,gy  A=aclr  \
     C=ctol(1),ctol(2),...,ctol(8)  I=mxit(1),mxit(2)

CHANGE STRUCTURE
RESTRAINTS
   n1,n2,inc  R=rx,ry,rzz                                        !
BUILD   N=n1,n2,inc  D=cd
REMOVE  N=n1,n2,inc
STRESS  N=n1,n2,inc  T=ktns  R=ra,rb  S=sa,sb  F=fa,fb  D=da,db
MOVE    N=nt  D=n(1),n(2),...,n(nnit)
CHANGE COMPLETE

LOADING
   N=n1,n2,inc  F=fx,fy,fzz
   N=n1,n2,inc  D=dx,dy,dzz
   L=l1,l2,inc  F=fx,fy,fzz
   L=l1,l2,inc  P=mtdp  T=tmlt                                   !

SOLVE  T=ktim  C=kldc  S=nstp  D=date  L=nlds  G=kgn  X=kgs  \
       N=nodc  O=ndof  A=cdsp  P=iprn  Q=lprn
       F=fload(1),fload(2),...,fload(nlds)
       F=fdisp(1),fdisp(2),...,fdisp(nlds)

CAMBER

STOP
```

2.5 Detailed Description of Command Syntax

The detailed syntax of each command and a description of its use, actions and output are described below. The commands appear in the approximate order in which they will be found in a typical input file. Most commands are optional, and must not be used unless required for the structure under analysis. Default values for input quantities, where implemented in the program, are indicated by [?] in the descriptions. [pv] indicates a default value of the previous value entered. The backslash "\" is used for continuation of a command on the next physical line.

The START Command

```
START  R=rstrt

   rstrt = Data restore flag [0]
         = 0 to start a new problem
         = 1 to restore database
```

The START command specifies the location in the input file at which execution begins. All input lines before the START command are ignored. All input lines after the START command are interpreted as input data.

Under most circumstances, this command is the first command in the input file. It may be located later in the input file when the analysis is a restart of a prior analysis terminated with the STOP command. For a restart case, R=1 should be specified on the command line so that the database will be restored.

Judicious use of the restart option can lead to substantial savings in the execution time, especially in the nonlinear analysis of complex structures.

The TITLE Command

```
TITLE  N=n1
```

n1 = Number of problem identification text lines [1]

The TITLE command prints a program identifier in the output file, and then prints n1 lines of text provided on the n1 input lines immediately following the command line.

The TITLE command is optional, but should be the first command interpreted, in order to clearly identify the output file.

The UNIT Command

```
UNIT  U=funit-lunit  C=cfunit
```

funit = force unit [lb]
lunit = length unit [in]
cfunit = unit conversion factor [1.0]

Any consistent set of units can be used in the analysis. The UNIT command specifies the force and length units used in the analysis. The following combinations of units can be specified.

SI	N ,kN	- mm,cm,m
Metric	kg,ton	- mm,cm,m
US Conventional	lb,kip	- in,ft

If the force unit - length unit combination is chosen within the same unit system listed above the unit conversion factor is computed internally. If the unit combination is mixed among different unit systems (e.g. kip-m) or any unit system which is not listed above is used, the unit conversion factor, cfunit must be input.

The unit conversion factor is defined as the multiplier of the stress unit of the lb-in system to convert it to the current unit system. For example, if kip-m system is used, the conversion factor, cfunit = 1.55 since 1 psi = 1.55 kips/m^2.

Whatever unit system is chosen, consistent units have to be used throughout the input data except for the CONCRETE PARAMETERS command in which specific unit systems to be used are designated.

The UNIT command is optional. If the default unit system lb-in is used this command can be omitted.

The CONCRETE PARAMETERS Command

```
CONCRETE PARAMETERS  N=ncpt  T=na,nt,nr  P=iprn
(                                    )
( Concrete Parameter Specifications )
(                                    )
```

ncpt = Number of concrete parameter types
na = Maximum "loading ages" in parameter tables [32]
nt = Maximum "times after loading" for parameter tables [48]
nr = Maximum "retardation times" in parameter tables [4]
iprn = Output print flag [0]

The CONCRETE PARAMETERS command generates tables of initial elastic modulus, shrinkage strain, compressive strength, tensile strength and creep coefficients versus time. The CONCRETE PARAMETERS command must be used prior to the MESH INPUT command described below.

These tables provide the constitutive constants used in the time dependent nonlinear analysis of the structure. The values in the tables are normalized for ultimate creep coefficient C_u = 1.0, ultimate shrinkage strain = 1.0, and cylinder strength $f'_c(28)$ =

1.0. Prior to use in each solution step, these normalized values are scaled by the ultimate creep coefficient, ultimate shrinkage strain and $f'_c(28)$ values input under the CONCRETE PROPERTIES subcommand of the MESH INPUT command described below. The values in the tables are first generated in lb-in units, and then converted to user specified units internally. Only one concrete parameter type is required under most circumstances.

The quantity of output is controlled using the P= identifier. For P=0, only the echo print of the input data is output. For P=1, a summary of material parameters for each loading age of each parameter type is also output. For P=2, a detailed, very large table of model diagnostics is also output.

Time dependent concrete behavior can be modeled according to the ACI recommendations, the CEB/FIP recommendations, or laboratory test data. Each parameter type can employ a different model.

Concrete Parameter Specification :

The first data line for each parameter type is a control data line which takes the following form. If the internally generated loading ages and observation times are used, this is the only line required for the parameter type.

```
n  M=model  T=nat,ntt,nrt  R=rt  G=igen . . . . .
```

n = Parameter type number
model = Model type used [ACI]
= ACI for ACI recommendations
= CEB for CEB/FIP recommendations
= CEB1 for one component CEB/FIP model
= LAB for laboratory test data
nat = Number of loading ages in this table [na]
ntt = Number of times after loading for this table [nt]
nrt = Number of retardation times in this table [nr]

rt = Smallest retardation time for creep model (days) [5]
igen = Loading ages and times generation flag [0]
= 0 for ages and times generated by the program
= 1 for ages and times to be input on the following lines

Additional data are interpreted from this line, the meaning of which depends on the model specified with the M= identifier.

When M=ACI is specified, the following additional data are interpreted from the control line:

A=a B=b C=c D=d E=e F=f T=t0 W=w X=rt

a [4.0] and b [0.85] are used in the time function for $f'_c(t)$:

$$f'_c(t) = \frac{t}{a + bt} \cdot f'_c(28)$$

c [1.25] and d [0.118] are used in the age function for creep:

$$K_\tau = c \cdot \tau^{-d}$$

e [1.0], f [50.0] and t0 [7.0] are used in the time function for shrinkage:

$$\epsilon^s(t) = \frac{(t-t0)^e}{f + (t-t0)^e} \cdot \epsilon^s_u$$

w [150.0] is the unit weight of concrete (pcf), used in

$$E(t) = 33 \cdot w^{1.5}[f'_c(t)]^{1/2}$$

rt [7.5] is the tensile strength ratio, used in

$$f'_t(t) = rt \cdot [f'_c(t)]^{1/2}$$

When M=CEB is specified, the following additional data are interpreted from the control line:

```
H=humd  A=area  P=prim  X=rt
```

humd = Ambient relative humidity (percent) [70.0]
area = Cross sectional area of the specimen (cm^2) [30.0]
prim = Perimeter of the specimen (cm) [1.0]
rt = Tensile strength ratio as in ACI input [7.5]

When M=ACI or M=CEB is specified along with G=0, loading ages and observation times are not generated and must be input from the following lines. For each loading age 1,2,...,nat, one additional line of input in the following format is required.

```
A=agel  T=time(1),time(2),...,time(ntt)
```

agel = Age at loading (days) for table generation
time(?) = Observation times (days) for table generation

When M=LAB is specified, the loading ages, observation times, initial elastic moduli, shrinkage strains, compressive strengths, tensile strengths and creep strains are not generated by the program and must be input from the following lines. For each loading age 1,2,...,nat, two additional lines of input in the following format are required. The units for this case should be consistent with those which were input under the UNIT command.

```
A=agel  E=emod  S=shrn  F=fpc  R=fpt  T=time(1),...,time(ntt)
C=crep(1),crep(2),...,crep(ntt)
```

agel = Age at loading (days) for table generation
emod = Initial elastic modulus E(agel)
shrn = Shrinkage strain ϵ^s(agel)
fpc = Compressive strength f'_c(agel)
fpt = Tensile strength f'_t(agel)
time(?) = Observation times (days) for creep strains
crep(?) = Creep strains corresponding to agel and time

The MESH INPUT Command

```
MESH INPUT
(                          )
( Mesh Input Subcommands )
(                          )
```

The MESH INPUT command has no arguments of its own, but is followed by a series of mesh input subcommands which specify the node coordinates, material and section properties, and frame and tendon element geometries for the plane frame structure. All nodes, frame elements, tendons and travelers which will ever exist in the analysis history of the structure are defined using the mesh input subcommands. The erection sequence is specified later in the input using the RESTRAINTS, BUILD(frame element), REMOVE(frame element), STRESS(tendon) and MOVE(traveler) subcommands of the CHANGE STRUCTURE command.

The mesh input subcommands include NODES, SEQUENCE, CONCRETE PROPERTIES, MILD STEEL PROPERTIES, PRESTRESSING STEEL, SECTION PROPERTIES, FRAME ELEMENTS, TENDON GEOMETRY, TRAVELERS, and MESH COMPLETE described below.

The MESH INPUT command can be used only once in a given analysis. Each subcommand can also be used only once. The subcommands should be entered in the order in which they are presented below.

The NODES Subcommand

```
NODES N=nnod
   n  X=xord  Y=yord  C=xord,yord  S=scale  G=n1,n2,inc
```

nnod = Total number of nodes in the structure

n = Node number

xord = Global X-coordinate of node n1 [0,pv]

yord = Global Y-coordinate of node n1 [0,pv]
scale = Scale factor for global X- and Y- coordinates [1,pv]
n1,n2,inc = Generation parameters described below

The NODES command is used to specify the total number of nodes and the geometry of the nodes. The node number n must be less than or equal to the total number of nodes input on the NODES command line. Node coordinates may be input or generated in any order with any number of data lines. If a node is input or generated more than once, only the last specification is used.

Node coordinates are input in the global X- Y- coordinate system (Fig. 2.1). The input X- and Y- node coordinates are multiplied by the scale factor immediately after input. Once the scale factor is entered (usually on the first line of node input data) do not enter it again unless it is necessary to reset its value.

Additional node numbers and coordinates may be automatically generated using the G=n1,n2,inc parameters. Nodes and node coordinates are generated at equal intervals along a straight line between two previously specified nodes. The node generation parameters are defined as

n1 = A previously specified node number
n2 = A previously specified node number
inc = Node number increment defining the generated nodes [1]

If node coordinates are defined more than once, only the last definition will be used. The final set of coordinates for all nodes is printed in the output.

This sequence of lines must be terminated by a blank line.

The SEQUENCE Subcommand

SEQUENCE

n1,n2,inc

n1 = First node in a generation sequence

n2 = Last node in a generation sequence

inc = Node number increment [1]

The SEQUENCE command is an optional command used to specify the node number order used for numbering the nodal displacement degrees of freedom of the structure. Skillful use of this command can reduce the profile of the stiffness matrix and increase the program's efficiency in solving the equilibrium equations. If this command is not used, degrees of freedom will be numbered in node number order 1,2,...,nnod.

Data lines must be provided to generate a list of node numbers in the order in which their degrees of freedom are to be numbered. Each node number must appear not more than once in this list. Any nodes which are omitted from this list will be numbered in numerical order after all other nodes.

This sequence of lines must be terminated by a blank line.

The CONCRETE PROPERTIES Subcommand

```
CONCRETE PROPERTIES  N=ncnc
   n  F=fpc  C=ultcrp  S=ultshr  W=weight  M=model  A=alfa  \
      O=r1   P=r2      T=type
      C=rc   U=epsu    E=eps20   R=r20     K=kts    (for T=2 only)
```

ncnc = Total number of concrete types [1]
n = Concrete type number
fpc = 28 day cylinder strength
ultcrp = Ultimate creep coefficient [0.0]
ultshr = Ultimate shrinkage coefficient [0.0]
weight = Unit weight (for computing dead load only) [0.0]
model = Time dependent model (parameter type) number [1]
alfa = Thermal expansion coefficient [0.0]
r1 = Ratio r1 for effective stress calculation [0.35]
r2 = Ratio r2 for effective stress calculation [1.865]
type = Stress-strain curve type
= 1 for linear curve
= 2 for nonlinear curve
rc = Ratio rc for compressive strength calculation [1.0]
epsu = Ultimate compressive strain [0.0031]
eps20 = Strain at the end of descending straight line [0.003]
r20 = Ratio for computing f20 corresponding to eps20 [0.85]
kts = Tension stiffening code [0]
= 0 if tension stiffening is not considered
= 1 if tension stiffening is considered

The CONCRETE PROPERTIES command is used to specify the different basic concrete material properties found in the structure. The concrete types may be supplied in any order; however each type must be specified once and only once. Absolute values for the stresses and strains should be entered even though they represent the compressive region of the stress-strain curve. Proper signs are assigned in the program. Consistent units as specified in the UNIT command should be used.

Only the simple descriptive properties listed above are input under this command. The complete numerical description of the concrete at different times is found by the program by combining this data with the time dependent constitutive constants which were input or generated using the CONCRETE PARAMETERS command.

Nonlinear creep effects at high stress levels are accounted for by the effective stress concept. The effective stress f_e is defined by the following equations:

$$
\begin{array}{lll}
f_e = f & \text{if} & f < r1*f''_c \\
f_e = c1*f + c2*f''_c & \text{if} & r1*f''_c < f < f''_c \\
f_e = r2*f & \text{if} & f = f''_c
\end{array}
$$

In the above equations r1 is the stress-strength ratio up to which creep strain is proportional to stress intensity, and r2 is the magnifying factor when the stress equals the maximum compressive stress f''_c. c1 and c2 are computed in the program by linear interpolation. Nonlinear creep effects can be neglected in the analysis by specifying r1 = r2 = 1.0.

Both linear and nonlinear stress-strain curve types can be used in the analysis. When the nonlinear type is used, an additional line of data characterizing the stress-strain curve should be input. See Fig. 2.2(a) for graphic representation of these data. rc is used in $f''_c = rc*f'_c$ for computing f''_c, the maximum cpmpressive stress. The horizontal straigt line branch of the stress-strain curve is used to model the behavior of the confined concrete. The stress f20 corresponding to eps20 is computed by $f20 = r20*f''_c$. The default values are set for normal unconfined concrete.

A table of concrete properties is printed in the output.

The MILD STEEL PROPERTIES Subcommand

```
MILD STEEL PROPERTIES  N=nmsl
   n  E=Es1  T=Es2  Y=fsy  U=epsu  A=alfa
```

nmsl = Total number of mild (reinforcing) steel types [1]
n = Mild steel type number
Es1 = First elastic modulus
Es2 = Second elastic modulus
fsy = Yield stress
epsu = Ultimate strain
alfa = Thermal expansion coefficient

The MILD STEEL PROPERTIES command is used to specify the different mild (reinforcing) steel material properties found in the structure. Mild steel types may be supplied in any order; however each mild steel type must be specified once and only once.

Bilinear stress-strain relationship assumed in the analysis is shown in Fig. 2.2(b). Both linear and nonlinear steel properties can be modeled.

A table of mild steel properties for all mild steel types is printed in the output.

The PRESTRESSING STEEL Subcommand

```
PRESTRESSING STEEL  N=npsl  P=npt
   n  C=crvf  W=wblf  F=fpy  R=rlxp  A=alfa
      F=fp(1,n),fp(2,n),...,fp(npt,n)
      E=ep(1,n),ep(2,n),...,ep(npt,n)
```

npsl = Total number of prestressing steel types [1]

npt = Number of points on the multilinear stress-strain curve excluding the origin (maximum 10)

n = Prestressing steel type number

crvf = Curvature friction coefficient [0.0]

wblf = Wobble friction coefficient per unit length [0.0]

fpy = 0.1 % offset yield stress for relaxation calculation

rlxp = Relaxation coefficient [10.0]

alfa = Thermal expansion coefficient [0.0]

fp(?,n) = stress value at point "?" for steel type n

ep(?,n) = strain value at point "?" for steel type n

The PRESTRESSING STEEL PROPERTIES command is used to specify the different prestressing steel material properties found in the structure. Prestressing steel types may be supplied in any order; however each prestressing steel type must be specified once and only once.

Multilinear stress-strain curve is assumed in the analysis. See Fig. 2.2(c) for graphical representation. The discrete stress and strain values at npt points are input on the third and fourth lines, respectively. Both linear and nonlinear stress-strain relations can be modeled.

Prestressing steel is considered as a relaxing material. The yield stress is used only in the relaxation calculation. The friction coefficients are used in computing the initial tendon forces under the STRESS subcommand of the CHANGE STRUCTURE command.

The SECTION PROPERTIES Subcommand

```
SECTION PROPERTIES  N=nsct  C=mcl  S=msl
   n  N=nc  C=ncl  S=nsl  K=kshr
      A=acl(1,n),acl(2,n),...,acl(ncl,n)
      Y=ycl(1,n),ycl(2,n),...,ycl(ncl,n)
      S=shk(1,n),shk(ncl,n)               (This line for K=2 only)
      S=shk(1,n),shk(2,n),...,shk(ncl,n)  (This line for K=3 only)
      B=asl(1,n),asl(2,n),...,asl(nsl,n)  (Skip if nsl=0)
      Z=ysl(1,n),ysl(2,n),...,ysl(nsl,n)  (Skip if nsl=0)
      M=ssl(1,n),ssl(2,n),...,ssl(nsl,n)  (Skip if nsl=0)
```

nsct = Total number of section types [1]

mcl = Maximum number of concrete layers in any section

msl = Maximum number of mild steel layers in any section

n = Section type number

nc = Concrete type number [1]

ncl = Number of concrete layers [mcl]

nsl = Number of mild steel layers [msl]

kshr = Shrinkage distribution code [0]
= 0 for no shrinkage
= 1 for uniform distribution
= 2 for linear distribution
= 3 for nonlinear distribution

acl(?,n) = Area of concrete layer No. "?" in section No. n

ycl(?,n) = Local y-coordinate of concrete layer No. "?" in section No. n

shk(?,n) = Shrinkage factor of concrete layer No. "?" in section No. n

asl(?,n) = Area of mild steel layer No. "?" in section No. n

ysl(?,n) = Local y-coordinate of mild steel layer No. "?" in section No. n

ssl(?,n) = Mild steel type number of mild steel layer No. "?" in section No. n

The SECTION PROPERTIES command is used to specify the different section properties found in the frame elements making up the structure. The section types may be supplied in any order; however each section type must be specified once and only once.

The cross section is divided into a number of concrete and mild steel layers as shown in Fig. 2.1(c). Principal properties of these layers are their areas and local y-coordinates (distances from the reference axis). Note that the concrete type number is an attribute of a section type while the mild steel type number is an attribute of a mild steel layer so that each mild steel layer can be assigned a different steel type.

Arbitrary distribution of shrinkage strain over the depth of the cross section can be modeled due to the layer discretization. The shrinkage factor input can be skipped if the shrinkage is not considered in the analysis or if the shrinkage distribution is uniform over the cross section. The shrinkage factors entered here are fractions of shrinkage strain present in the structure at a particular time.

A table of section properties including the total area, the static moment of area and the moment of inertia about the reference axis is printed in the output.

The TEMPERATURE PATTERNS Subcommand

```
TEMPERATURE PATTERNS  N=ntdp
   n  X=nx  K=ktmp
      T=tfac(1),tfac(ncl)             (This line for K=2 only)
      T=tfac(1),tfac(2),...,tfac(ncl) (This line for K=3 only)
```

ntdp = Total number of temperature distribution patterns [1]
n = Temperature distribution pattern number
nx = Section type number
ktmp = Temperature distribution code [1]
= 1 for uniform distribution
= 2 for linear distribution
= 3 for nonlinear distribution
ncl = Number of concrete layers in section type nx
tfac(?) = Temperature factor for concrete layer number "?"

The TEMPERATURE PATTERNS command is used to specify the different temperature distribution patterns found in the the structure. The temperature distribution patterns may be specified in any order; however each temperature distribution pattern must be specified once and only once.

Arbitrary distribution of temperature over the depth of the cross section can be modeled due to the layer discretization. The temperature factors entered here are multiplied by the element temperature multipliers input under the LOADING command to produce the actual temperatures at each part of the structure.

The temperatures of mild steel layers and prestressing tendons are computed in the program based on their locations within the cross section.

The TEMPERATURE PATTERNS command is optional. If the temperature variation is not considered in the analysis this command can be omitted.

The FRAME ELEMENTS Subcommand

FRAME ELEMENTS N=nfml
n,ni,nj X=nx D=cd F=if G=n1,n2,...,n7

nfml = Total number of frame elements [1]
n = Frame element number
ni = Node number for node i
nj = Node number for node j
nx = Section type number
cd = Casting date (days)
if = Creep integration type [3]
n1 - n7 = Generation parameters described below

The FRAME ELEMENTS command is used to define all the frame elements used in modeling the structure. The frame element descriptions may be supplied in any order; however each frame element description must be specified or generated once and only once.

The geometry of frame elements is shown in Fig. 2.1. Frame elements consist of parallel concrete and mild steel layers. The structural properties of a frame element are determined by the specificaion of its two end nodes and the section type. The casting date (in days) may be specified with the D= identifier, but this specification can be overridden under the BUILD subcommand of the CHANGE STRUCTURE command.

The type of creep strain integration over a time step is specified with the F= identifier. With F=1, constant stress and constant material parameters are assumed over a time step. With F=2, linear stress variation and constant material parameters are assumed. With F=3, linear variations of both the stress and the material parameters are assumed.

Additional frame elements may be automatically generated using the G=n1,n2,...,n7 parameters. Frame elements are generated by incrementing the input parameters by their respective increments, which are input using the G= identifier. The generation parameters are defined as

n1 = First element in generation sequence (usually = n)
n2 = Last element in generation sequence
n3 = Frame element number increment
n4 = Node i increment
n5 = Node j increment
n6 = Cross section type increment
n7 = Casting date increment

A table of frame element specifications is printed in the output.

<u>This sequence of lines must be terminated by a blank line.</u>

The TENDON GEOMETRY Subcommand

```
TENDON GEOMETRY   N=ntdn   P=iprn
   (                                )
   ( Tendon Geometry Specifications )
   (                                )
```

ntdn = Total number of prestressing steel tendons [1]
iprn = Output print flag [1]
  = 1 for short form
  = 2 for long form

The TENDON GEOMETRY command is used to specify and generate the geometry of all the prestressing steel tendons.

Each tendon is modeled as a series of segments connected at tendon points (Fig. 2.3(a)). Each tendon point is rigidly constrained to a specified node, which is usually different for each tendon point. Each tendon segment must lie entirely within one frame element. The global X- and Y- coordinates of the tendon points completely define the tendon's geometry. These coordinates may be input directly for each tendon point, or they may be generated using a parametric generation scheme. The input or generation can be in an alternative, translated and rotated local r- s- coordinate system specified by the user. The r- and s- coordinates are converted internally to the global X- Y- system.

For input convenience, each tendon may consist of several spans. These spans are defined by the node numbers corresponding to the tendon points along their length, and need not correspond to the actual spans of the structure. The tendon point geometry of each span may be either directly input or parametrically generated.

Tables of tendon point and segment geometry may be printed in the output. The P= identifier on the TENDON GEOMETRY command line is used to set the amount of output and the number of tables printed. If P=0, only an echo of the input is printed. If P=1, a table of final tendon point coordinates is added. If P=2, detailed tables of tendon point local and global coordinates and tendon segment geometries are added. Use of P=2 produces voluminous output.

Several lines of input, described below, are required to specify the geometry of each span of each tendon. This set of input lines must be provided for each tendon.

Tendon Geometry Specification :

The first line of each tendon is the tendon control data line. Tendon numbers must be supplied in ascending consecutive order starting with tendon 1.

n S=nspn M=matl A=area

n = Tendon identification number
nspn = Number of spans used to define the tendon geometry
matl = Presressing steel type number
area = Cross section area for the tendon

For each span, several lines must be provided to input or generate the tendon point geometry for that span. The first line specifies the number of nodes and the node numbers defining the span, and alternatively the base vectors for the local r- s- coordinate system (Fig. 2.3(a)) in which the tendon geometry is input or generated. Once these base vectors have been entered for a particular span, they need not be entered again for other spans or other tendons unless it is necessary to reset them. The initial default local coordinate system is the global X- Y- coordinate system. Span numbers must be supplied in ascending consecutive order starting with span 1.

ns N=nnis G=n1,n2,inc B=x0,y0 E=x1,y1

ns = Span number
nnis = Number of nodes in the span (including both end nodes)
n1 = Node number at "left" end of the span
n2 = Node number at "right" end of the span
inc = Node number increments for generating nodes
x0,y0 = Global coordinates of the origin of the local r- s- coordinate system [0.0,0.0]
x1,y1 = Global coordinates of a point on the local r- axis [1.0,0.0]

If the node numbers cannot be generated for a particular span, then the G= identifier above should be excluded, and the next line must provide a list of the nnis node numbers included in the span, in order from "left" to "right" in the span.

L=n(1),n(2),...,n(nnis)

n(?) = Node number included in the span
nnis = Number of nodes in the span (input on previous line)

The next line(s) provide data for generation or direct input of tendon point coordinates for the span. If parametric tendon point generation is used for the span, one line with the following data must be provided. See Fig. 2.3(b) for the graphical description of terms to be input.

R=rlli,rllp,rlri S=slt,slp,srt

rlli = Fraction of the total span length of the distance between the "left" end of the span and the "left" inflection point (a positive number)
rllp = Fraction of the total span length of the distance between the "left" end of the span and the point of zero tendon slope ("low point") relative to the r- s- coordinate system (a positive number)
rlri = Fraction of the total span length of the distance between the "right" inflection point and the "right" end of the span (a positive number)
slt = s- coordinate of left end of tendon
slp = s- coordinate of point of zero slope
srt = s- coordinate of right end of tendon

If direct tendon point coordinate input is used for the span, the following nnis lines must be provided in place of the one above. In this case the default r- coordinates are the r- coordinates of the nodes included in the span.

np R=r(np) S=s(np)
np = Tendon point number
r(np) = r- coordinate of tendon point np
s(np) = s- coordinate of tendon point np

The TRAVELERS Subcommand

```
TRAVELERS  N=ntrv
   n  E=emod  A=area  I=izz  W=wegt  N=nnit
```

ntrv = Total number of travelers
n = Traveler number
emod = Elastic modulus of material
area = Cross section area
izz = Moment of inertia
wegt = Total weight of traveler
nnit = Number of nodes in traveler

The TRAVELERS command is used to describe all the traveling formworks used in modeling the construction sequence. Traveler numbers must be supplied in ascending consecutive order starting with traveler 1.

Travelers are modeled as linear elastic frame elements linearly connecting several nodes. Only the traveler properties are input under this command. The locations of the travelers may change at any time are input under the MOVE subcommand of the CHANGE STRUCTURE command.

The MESH COMPLETE Subcommand

```
MESH COMPLETE
```

The MESH COMPLETE command has no arguments. It signals the program that the mesh input phase of the analysis is complete. No mesh input commands are allowed after this command has been interpreted.

The SET Command

```
SET  D=day  T=temp  G=gx,gy  A=aclr  \
   C=ctol(1),ctol(2),...,ctol(8)  I=mxit(1),mxit(2)
```

day = Current date (days)
temp = Current temperature [0.0]
gx = Global X- direction gravity load multiplier [0.0]
gy = Global Y- direction gravity load multiplier [-1.0]
aclr = Time step solution covergenge acceleration factor [.71]
ctol(1) = tols, Stress ratio tolerance for time step solution
ctol(2) = tolf, Displacement ratio tolerance for final load or displacement step
ctol(3) = toli, Displacement ratio tolerance for intermediate load or displacement steps
ctol(4) = tolc, Displacement ratio tolerance for changing stiffness matrix
ctol(5) = toll, Maximum unbalanced force allowed in iteration
ctol(6) = tolm, Maximum unbalanced moment allowed in iteration
ctol(7) = told, Maximum translation allowed in iteration
ctol(8) = tolr, Maximum rotation allowed in iteration
mxit(1) = Maximum number of iterations allowed for final load or displacement step [50]
mxit(2) = Maximum number of iterations allowed for intermediate load or displacement steps [30]

The SET command is used to set and reset the basic environmental factors influencing the solution, as well as solution convergence and acceleration which influence the accuracy and cost of the analysis. The command may be issued any number of times in order to change these factors as required.

The D= identifier is used to set the date at the end of the next solution step. This date can also be set under the SOLVE command.

The T= identifier is used to set the element temperatures for succeeding solution steps. This temperature is used for the calculation of temperature strains only. This temperature specification overrides any previously specified temperature gradients entered using the LOADING command.

The G= identifier is used to set the load multipliers for gravity loads in the global X- and Y- directions. The gravity load multipliers gx and gy, however, influence only the gravity load increment and should be set only once, prior to any construction operations performed under the CHANGE STRUCTURE command.

The A= identifier is used to set the convergence acceleration factor for the creep solution. The default value has proven satisfactory in most cases.

The C= identifier is used to set various convergence tolerances required to control the flow of the nonlinear analysis solution scheme. Since these tolerances are the most important parameters which control the nonlinear analysis procedure, each term will be described in detail. All the convergence tolerance values should be input in absolute values.

ctol(1) = tols. The stress ratio tolerance is used to control the time step solution. The stress ratio is defined as the ratio of the stress increment for the current iteration to the total stress increment of the time step up to previous iteration. If the stress ratio is greater than tols, then the iteration continues. The stress component to be checked is the maximum stress occurring on the first iteration of the time step. Specifically, the location of the maximum stress is identified by the Gauss integration point number, concrete layer number and the frame element number. This location identification data along with the stress ratio value is printed in the output. The tols value on the order of 0.01 has yielded satisfactory results.

ctol(2) = tolf. The displacement ratio tolerance is used to control the nonlinear load or displacement step solution. The displacement ratio is defined as the ratio of the displacement increment for the current iteration to the total displacement increment up to previous iteration of the current load or displacement step. In the final load or displacement step, if the displacement ratio is greater than tolf, then the iteration continues. The displacement component to be checked is the maximum translation or rotation occurring on the first iteration of the current load or displacement step. Between the translational and rotational components, the one which yields the greater ratio is the controlling displacement component. In the displacement step solution, the controlled displacement component cannot be selected as the controlling displacement component. The controlling displacement component and the displacement ratio is printed in the output. The tolf value between 0.001 and 0.05 has been used successfully.

ctol(3) = toli. In intermediate load or displacement steps, if the displacement ratio is greater than toli, then the iteration continues. The toli value between 0.001 and 0.05 has been used successfully.

ctol(4) = tolc. Either constant stiffness or variable stiffness iteration can be selected by this tolerance. If the displacement ratio is greater than tolc, then new structure tangent stiffness matrix is formed and used for the next iteration. Otherwise, previously formed and reduced stiffness matrix is used. The number of iterations required to arrive at the solution is affected by the value of tolc. In a typical nonlinear analysis problem the state determination phase takes up more computer time than the stiffness formation and reduction phase. Thus the use of small values for tolc, which results in the smaller number of iterations, is recommended, especially for ultimate load analyses in which structure tangent stiffness varies steeply. However, for creep analyses in which the structure tangent stiffness stays essentially the same, larger value for tolc is recommended. The tolc value between 0.0001 and 0.1 has yielded satisfactory results.

ctol(5) = toll, ctol(6) = tolm. These tolerances set the ceiling on the maximum unbalanced load allowed in an iteration. If the maximum unbalanced force component is greater than toll or the maximum unbalanced moment component is greater than tolm, then the iteration continues. Note that for an iteration to be terminated, the conditions for the unbalanced load as well as the displacement ratio or the stress ratio have to be satisfied. The values for toll and tolm obviously depend on the units used in the analysis. Approximately 1/1000 to 1/100 of the maximum expected nodal force or moment is recommended.

ctol(7) = told, ctol(8) = tolr. These tolerances set the ceiling on the maximum allowed values of translational and rotational displacement increments in an iteration. These are provided in order to guard against the displacement overshoot which may occur in a geometrically nonlinear analysis or a load reversal analysis with material nonlinearities. Approximately 10 times the maximum expected displacement or rotation increment value is recommended.

mxit(1) and mxit(2). In addition to the convergence tolerances described above, a ceiling is provided to limit the number of iterations performed for each load or displacement step in case the tolerances provided are too stringent. mxit(1) is for the final load or displacement step and mxit(2) is for intermediate steps. Default values can be used for ordinary problems.

The CHANGE STRUCTURE Command

```
CHANGE STRUCTURE
(                          )
( Construction Subcommands )
(                          )
```

The CHANGE STRUCTURE command has no arguments of its own, but is followed by a series of construction operation subcommands which specify the current boundary conditions, the installation and removal of frame elements and prestressing steel tendons, and the movement of travelers in the plane frame structure. The structure to be analyzed at a particular time is completely defined by the cumulative effects of CHANGE STRUCTURE commands.

The construction operation subcommands include RESTRAINTS (to change nodal boundary conditions), BUILD (to install frame elements), REMOVE (to remove frame elements), STRESS (to stress, restress, or remove prestressing steel tendons), MOVE (to move traveling formworks), and CHANGE COMPLETE (to terminate the command) described below.

The CHANGE STRUCTURE command can be used any number of times in a given analysis in order to model the construction sequence. The effects of the change on the displacements and internal forces in the structure are found by using the SOLVE command after the CHANGE STRUCTURE command and any optional loadings commands.

The RESTRAINTS Subcommand

```
RESTRAINTS
   n1,n2,inc  R=rx,ry,rzz
```

n1 = Node number of first node in a series of nodes with identical restraint specifications
n2 = Node number of last node in the series [n1]
inc = Node number increment used to define nodes in the series
rx = X- displcement restraint specification
ry = Y- displcement restraint specification
rzz = Z- rotation restraint specification

The RESTRAINTS command is used to specify the bundary conditions on the structure. The data line is repeated as many times as required to specify the desired boundary condition changes.

Each node has three displacement degrees of freedom (Fig. 2.1(a)) each of which may be specified with one of three restraint types:

r? = 0 : Free to displace
r? = 1 : Fixed at current total displacement
r? = 2 : Fixed with zero displacement

If the restraint specification is "0" the degree of freedom is unrestrained, any existing reaction is applied as a load, and the displacement or rotation will be evaluated by the program. If the restraint specification is "1" the total displacement or rotation is restrained to its current value. If the restraint specification is "2" the total displacement or rotation is restrained to zero. Restraint types 1 and 2 are identical except that for type 2, an imposed displacement equal and opposite to the total current displacement is applied in the next solution step.

Any restraint change remains in effect until changed again by a subsequent RESTRAINTS subcommand. All unspecified nodes are assumed to have a "0" (free to displace) boundary condition in all three degrees of freedom.

The program automatically determines which nodes are defined in the current structure and includes only their unrestrained degrees of freedom in the global equilibrium equations. Thus the user need not restrain unused nodes with this command.

<u>This sequence of lines must be terminated by a blank line.</u>

The BUILD Subcommand

```
BUILD  N=n1,n2,inc  D=cd
```

n1 = Element number of first frame element in a series of elements to be installed in the structure
n2 = Element number of last element in the series [n1]
inc = Element number increment
cd = Casting date of concrete in the specified elements

The BUILD command is used to install new frame elements into the structure. The elements can later be removed using the REMOVE subcommand described below. The sequence of elements generated with the N= identifier must be statically feasible in order for node displacement initialization to work properly. Thus, for the BUILD command, backward generation (i.e. inc less than 0) is allowed. If a casting date is specified under this command, it overrides the value input under the FRAME ELEMENTS subcommand of the MESH INPUT command.

The dead load of the frame element is automatically included as concentrated forces at the nodes based on the length, cross section area and material unit weight of the elements input under the MESH INPUT command, multiplied by the current gravity load multipliers specified with the SET command.

The displacements of any previously unrestrained nodes which as a result of this command are made active, are initialized based on the total displacements of the node at the other end of the element and assumed rigid behavior of the element. This makes it necessary under this subcommand to generate elements in a statically feasible order.

The REMOVE Subcommand

```
REMOVE  N=n1,n2,inc
```

n1 = Element number of first frame element in a series of elements to be removed from the structure
n2 = Element number of last element in the series [n1]
inc = Element number increment

The REMOVE command is used to remove existing frame elements from the structure. The elements must have been installed using the BUILD subcommand described above. Once an element has been removed with this command, it is permanently gone from the structure and may never be installed again.

The program automatically removes the stiffness, dead load and internal forces from the system matrices when an element is removed. Any additional loads applied to the element under the LOADINGS command described below are <u>not</u> automatically removed, and must be removed manually (by applying an equal but opposite force with another LOADINGS command) <u>before</u> removing the element.

The STRESS Subcommand

```
STRESS N=n1,n2,inc T=ktns R=ra,rb  S=sa,sb  F=fa,fb  D=da,db
```

n1 = Tendon number of first tendon in a series of tendons with identical stressing specifications
n2 = Tendon number of last tendon in the series [n1]
inc = Tendon number increment
ktns = Tensioning operation code [0]
= 0 for post-tensioning
= 1 for pre-tensioning
ra = Jacking stress ratio at tendon end "A"
rb = Jacking stress ratio at tendon end "B"
sa = Jacking stress at tendon end "A"
sb = Jacking stress at tendon end "B"
fa = Jacking force at tendon end "A"
fa = Jacking force at tendon end "B"
da = Anchorage slip (draw-in) at tendon end "A"
db = Anchorage slip (draw-in) at tendon end "B"

The STRESS command is used to install, stress, restress and remove prestressing steel tendons. The tendons' geometry and material properties must have been input under the TENDON GEOMETRY subcommand of the MESH INPUT command.

T= identifier is used to specify the tensioning method. For pre-tensioned tendons, the friction coefficients must have been set to zero under the PRESTRESSING STEEL subcommand of the MESH INPUT command and anchorage slip values should be set to zero under this command.

A tendon is initially stressed by specifying its jacking force and anchorage slip values under this command. When the R= identifier is used to specify the jacking force, the jacking stress ratio is the ratio of the jacking stress to the 0.1 % offset yield stress of the prestressing steel. The tendon segment initial forces are calculated based on the jacking force input and the material properties of the prestressing steel. A tendon may be restressed by specifying a new jacking force under a subsequent application of this command. A tendon may be removed entirely by use of this command with a zero jacking force.

In the first solution step after a STRESS command has been applied to a tendon, the prestress is transferred to the structure as equivalent joint loads. The tendon stiffness is not included at transfer for post-tensioned structures while it is included at transfer for pre-tensioned structures. Thus for pre-tensioned structures, the prestress loss at transfer due to elastic shortening is taken into account. In subsequent solution steps the tendon stiffness is included in both cases.

The MOVE Subcommand

```
MOVE  N=nt  D=n(1),n(2),...,n(nnit)
```

nt = Traveler number
n(?) = Destination nodes for the new traveler location
nnit = Number of nodes in traveler

The MOVE command is used to install, move and remove traveling formwork. The traveler description must have been input under the TRAVELER subcommand of the MESH INPUT command. A traveler may be moved to any location on the structure as many times as desired during the analysis. The traveler may be removed entirely by specifying destination node n(1) = 0.

The dead load and stiffness of the traveler are automatically included in the analysis. When the traveler is moved to a new location, the element geometry of the traveler is adjusted to the node geometry at the new location and all nodes are automatically moved.

The CHANGE COMPLETE Subcommand

```
CHANGE COMPLETE
```

The CHANGE COMPLETE command has no arguments. It signals the program that the interpretation of CHANGE STRUCTURE commands should stop. The effects of the changes on the displacements and internal stresses in the structure are found by using the SOLVE command described below.

The LOADING Command

```
LOADING
   (                    )
   ( Loading Data Lines )
   (                    )
```

The loading command is used to apply concentrated nodal loads, imposed nodal displacements, uniformly distributed frame element loads and temperature change loadings to the structure. Any node or element to which a loading is applied must be a part of the current structure. All loadings remain in effect until they are removed by the application of an equal but opposite loading. All external loadings and imposed displacements should be given in global directions (Fig. 2.1(a)).

Each data line takes one of several forms depending on the type of loading. The loading data lines can contain all three forms in any order.

For concentrated nodal loads the following data must be provided. Nodal loads may be applied to any degree of freedom, whether or not it is free to displace. Loads on restrained degrees of freedom are retained for future use in case the degree of freedom is ever unrestrained.

N=n1,n2,inc F=fx,fy,fzz

n1 = Node number of first node in a series of nodes with identical loading
n2 = Node number of last node in the series [n1]
inc = Node number increment used to define nodes in the series
fx = X- force increment
fy = Y- force increment
fzz = Z- moment increment

<u>For imposed nodal displacements</u> the following data must be provided. Displacements may only be applied to restrained (fixed) degrees of freedom. Applied displacements on unrestrained degrees of freedom are neglected.

N=n1,n2,inc D=dx,dy,dzz

n1 = Node number of first node in a series of nodes with identical applied displacements
n2 = Node number of last node in the series [n1]
inc = Node number increment used to define nodes in the series
dx = X- displacement increment
dy = Y- displacement increment
dzz = Z- rotation increment

<u>For uniformly distributed frame element loads</u> the following data must be provided. Frame element loads are in the global directions, and are specified as force per unit projected length. Frame element loads are converted to concentrated nodal forces for the analysis

L=l1,l2,inc F=fx,fy,fzz

l1 = Element number of first node in a series of elements with identical loading
l2 = Element number of last node in the series [l1]

inc = Element number increment
fx = X- force increment per unit Y- projected length
fy = Y- force increment per unit X- projected length
fzz = Z- moment increment per unit element length

<u>For temperature change loadings</u> the following data must be provided. The temperature of each element may be different and may vary arbitrarily over the depth of the element. Uniform, linear and nonlinear temperature distributions can be considered. The actual temperatures at each part of the elements are computed by multiplying the temperature factors entered under the TEMPERATURE PATTERNS subcommand of the MESH INPUT command by the temperature multipliers given under this command. Temperature strains are included in the frame elements generated with the L= identifier, and also in mild steel layers and all prestressing tendon segments within the elements. The temperatures of mild steel layers and prestressing tendons within element are computed in the program based on their locations within the cross section. Temperature strains in the travelers are not included. The reference temperature for all elements is taken as the ambient temperature (see the SET command) on the day the element was installed.

L=l1,l2,inc P=mtdp T=tmlt

l1 = Element number of first node in a series of elements with identical temperatures
l2 = Element number of last node in the series [l1]
inc = Element number increment
mtdp = Temperature distribution pattern number
tmlt = Temperature multiplier

<u>This sequence of lines must be terminated by a blank line.</u>

The SOLVE Command

```
SOLVE  T=ktim  C=kldc  S=nstp  D=date  L=nlds  G=kgn  X=kgs  \
       N=nodc  O=ndof  A=cdsp  P=iprn  Q=lprn
   F=fload(1),fload(2),...,fload(nlds)  ( for nlds>1 and kldc=0 )
   F=fdisp(1),fdisp(2),...,fdisp(nlds)  ( for nlds>1 and kldc=1 )
```

```
ktim = Time step solution mode code [0]
     = 0 for intantaneous solution
     = 1 for time step solution
kldc = Nonlinear solution strategy code [0]
     = 0 for load control
     = 1 for displacement control
nstp = Number of time steps [1]
date = Day number at end of solution
nlds = Number of load or displacement steps [1 for ktim=1]
kgn  = Geometric nonlinearity code [0]
     = 0 if geometric nonlinearity is not considered
     = 1 if geometric nonlinearity is considered
kgs  = Geometric stiffness code [0]
     = 0 if geometric stiffness is not included
     = 1 if geometric stiffness is included
nodc = Node number with controlled displacement component
ndof = DOF number of controlled displacement component
     = 1 for global X- displacement
     = 2 for global Y- displacement
     = 3 for global Z- rotation
cdsp = Magnitude of controlled displacement
iprn = Output print flag [0]
     = -1 for no output
     = 0 for output at end of solution
     = 1 for output at end of each time step
     = 2 for output at end of each load or displacement step
     = 3 for output at end of each iteration
     = 4 for output at end of each iteration
         and structure stiffness matrix printout
```

lprn = Layer output print flag [0]
= 0 for no layer output
= 1 for layer output at center of element (Gauss point 1)
= 2 for layer output at all 3 Gauss points
fload(?) = multiplier for load step "?"
fdisp(?) = multiplier for displacement step "?"

The SOLVE command is used to solve the current structure for its displacements and internal stresses under current loadings at the specified time. This command performs the majority of the numerical operations required in the analysis.

All command line data are optional, and need not be entered unless they are different from the default values. The default value for the "date" is initially set as the "day" entered under the SET command. However, if the "date" is changed to a new value in a previous SOLVE command, then that value becomes the default value for the "date".

The SOLVE command steps the solution over the time interval from the time at the end of the previous solution to the day number specified with the D= identifier on the command line. All loadings are assumed to be applied gradually over the length of the time step. Thus any instantaneously applied loads require a zero length time step. A zero length time step is also required whenever the structure's configuration has been changed with the CHANGE STRUCTURE command. A zero length time step is specified by omitting the D= identifier on the command line.

In the instantaneous solution mode two options are available for the nonlinear analysis strategy; load control and displacement control. Displacement control strategy is useful in the post-buckling analysis or in the ultimate load analysis near failure load. Two strategies can be mixed in an analysis. The load or controlled displacement values are multiplied by the load or displacement multipliers for each load or displacement step for solving in nlds steps. Geometric nonlinearity and geometric stiffness can be included in the analysis.

In the time step solution mode the solution interval is divided into nstp time steps. For each time step nonlinear analysis is performed with single load step using load control strategy. The times for the intermediate solutions are set up for equal step lengths on a logarithmic scale. Thus if the solution interval is from day 100 to day 1000 and nstp = 4, intermediate solutions are performed for days 177.8, 316.2, 562.3 and 1000. This is a convenience option to simplify input when several time steps are required. The entire external loading increment, however, is applied in the first time step.

The convergence tolerances and the maximum number of iterations allowed for both load/displacement and time step solution modes must have been input under the SET command. These values can be reset by using another SET command if necessary.

Standard output at a particular stage of solution step consists of the convergence information, nodal displacements, current total external nodal loads, unbalanced loads and reactions, frame element forces and moments, prestressing tendon segment forces, material states, stresses and strains. Frame element forces and moments are output in local element coordinate system (see Fig. 2.1(b)). Contributions from concrete layers, mild steel layers, prestressing steel tendons and travelers to total frame element forces and moments are also shown. The standard output can be printed at various stages of solution step: after each iteration, after each load/displacement step, after each time step and at the end of solution depending on the value of the P= identifier.

Concrete and mild steel layer data can be printed by the Q= identifier. The layer data consist of the material state codes, stresses, mechanical and non-mechanical strains of concrete and mild steel layers at Gauss integration points. The distances of

Gauss points 1,2,3 from node i of the frame element on local x-axis are 0.5L, 0.11L and 0.89L, respectively, where L is the element length. See Fig. 2.2 for the definition of material state code numbers for the concrete, mild steel and prestressing steel.

The CAMBER Command

```
CAMBER  C=icmp
```

icmp = Displacement component to be printed [2]
= 1 for X- displacement
= 2 for Y- displacement
= 3 for Z- rotation

The CAMBER cammand is used to print a history of adjusted nodal displacements for all nodes in the structure. The displacements output under this command represent the displacements which must exist at the given times in order for the current total displacements to equal zero.

The STOP Command

```
STOP
```

The STOP command saves the entire structure database and terminates program execution. The analysis can be restarted to analyze for more time steps or load/displacement steps by providing the saved database files and an appropriate input file for the additional steps. See the description of the START command.

3. Illustrative Example

In order to demonstrate the use of the program, a three-span continuous post-tensioned box girder bridge designed and analyzed by Choudhury [12] will be analyzed by SPCFRAME.

3.1 Explanation of the Input Data

Input data listings of the Choudhury bridge is given in Fig. 3.3 (page 69). The configuration of the structure, its loading and the analysis model are shown in Fig. 3.1 (page 67). The structure is a post-tensioned three-span continuous box girder bridge having single cell with cantilevered top slabs. The bridge is post-tensioned with two parabolic tendons in each of its two vertical webs.

One half of the structure is modeled with 21 nodes and 20 frame elements. Kip-in units are selected and the ACI material model is used. On CONCRETE PARAMETERS command line the W= and X= identifiers are used to input the concrete weight and the tensile strength ratio for the calculation of the initial elastic modulus and the tensile strength of the concrete, because the values used by Choudhury are different from default values computed by the program.

In specifying nodal coordinates the foot unit is used in the input, but these will be converted to inch units by the scale factor given with the S= identifier.

The 28-day cyliner strength of the concrete is 4.0 ksi. However, a slightly smaller value is input under the CONCRETE PROPERTIES command with the F= identifier. This is due to the use of inexact default values of ACI coefficients for the calculation of concrete strength in the program. In order to get the exact strength value at 28 days after casting, the factor 0.99382 should be multiplied.

The cross-section is divided into 10 concrete layers and 5 mild steel layers. The mild steel, corresponding to 0.3% of the concrete area, is provided for construction purposes.

The prestressing steel tendon point data are generated by the parametric generation option. Two spans are used to specify the tendon configuration.

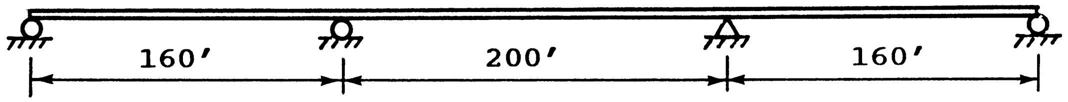
The convergence tolerances are set under the SET command. Relatively small value for changing stiffness is specified so that essentially variable stiffness iteration can be performed. The reason for this is that it is preferrable to keep the number of iterations small since the state determination phase takes much more computer time than the formation and reduction of the structure stiffness matrix for this problem. The maximum unbalanced force and moment values allowed in iteration are set as approximately 1/100 of the expected maximum frame element force and moment values. This relatively crude values are chosen since the finite element mesh layout used in the analysis is relatively crude for the accurate determination of the ultimate load.

Under the CHANGE STRUCTURE command all the frame elements are built into the structure in one step for this example, the boundary conditions are set, and the tensioning operation is performed.

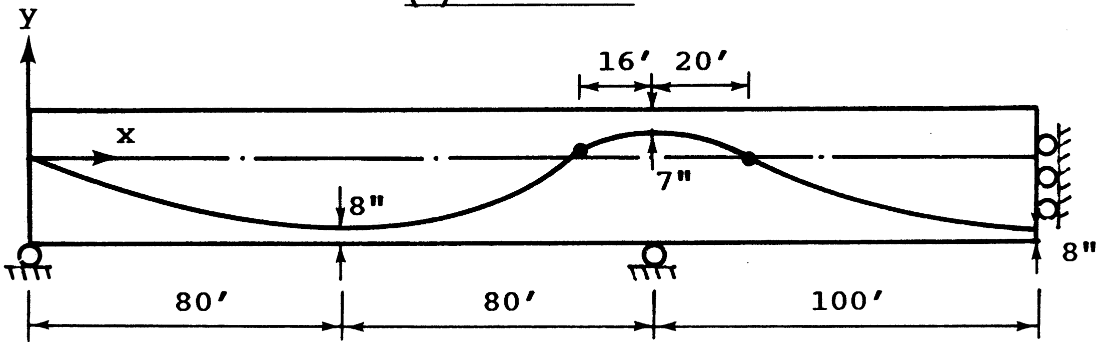
The structure is first analyzed for the dead load and the effects of prestressing under the first SOLVE command. Then the ultimate load analysis is carried out for the factor of P13 truck load which is typical of the heaviest vehicles found on California's highways. The idealized P13 truck load is shown in Fig. 3.1(c). The structure is first analyzed with eight load steps up to 5.5 times the truck load, then the displacement control strategy is used with three displacement steps to determine the ultimate load. The final displacement step is separated to avoid the voluminous printout of layer records during intermediate steps.

In Fig. 3.3 two additional input files for the same structure are shown. The first file shows the input data for analyzing the bridge up to 10000 days (27.4 years) after the completion of construction with 27 time steps. The second file shows the input data for the ultimate load analysis of the bridge at 10000 days after construction utilizing the restart option. Eight files are written after the execution of the first input file EX1.INP; they are EX1.INP, EX1.OUT, EX1.DSP, EX1.FRM, EX1.CNC, EX1.STL, EX1.TDN, and EX1.COR. Among these the last six files, in which all the results of the first analysis are stored, should be made available for the second analysis with the restart option. The second input data file must be named identically with the first input file; that is EX1.INP.

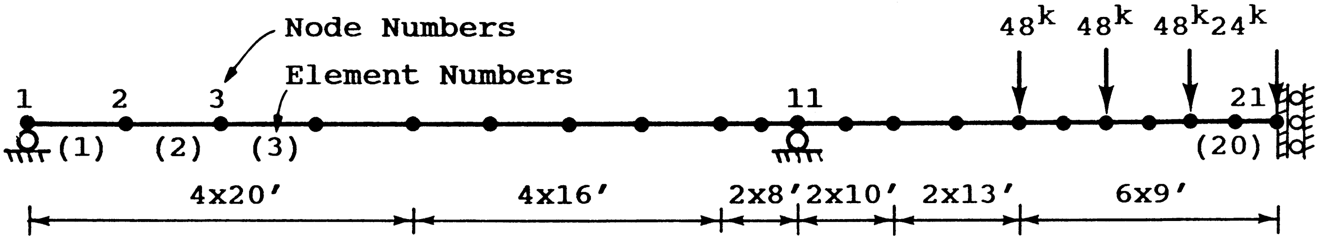
In Fig. 3.3 the input data for two more examples are given. One is for the post-buckling analysis of an eccentrically loaded pre-tensioned column (example 2), the other is for the creep and ultimate load analyses of a three-span cantilever construction post-tensioned box girder bridge (example 3). Example 3 is taken from Ketchum's report [10]. This example shows the capability and applicability of the program SPCFRAME for the time-dependent nonlinear analysis of practical segmentally erected bridges. The results of these and other examples will be presented in more detail in the forthcoming report mentioned earlier.



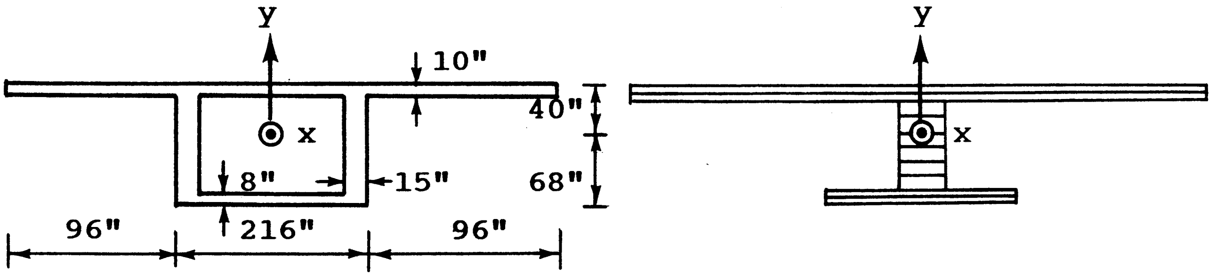
(a) Structure



(b) Prestressing Steel Tendon Profile



(c) Finite Element Mesh Layout and P13 Truck Loading



(d) Cross Section Idealization

Fig. 3.1 Example 1 - Choudhury Bridge
Three Span Post-tensioned Box Girder Bridge

Fig. 3.2 Example 1 - Choudhury Bridge
Comparison of Central Deflections

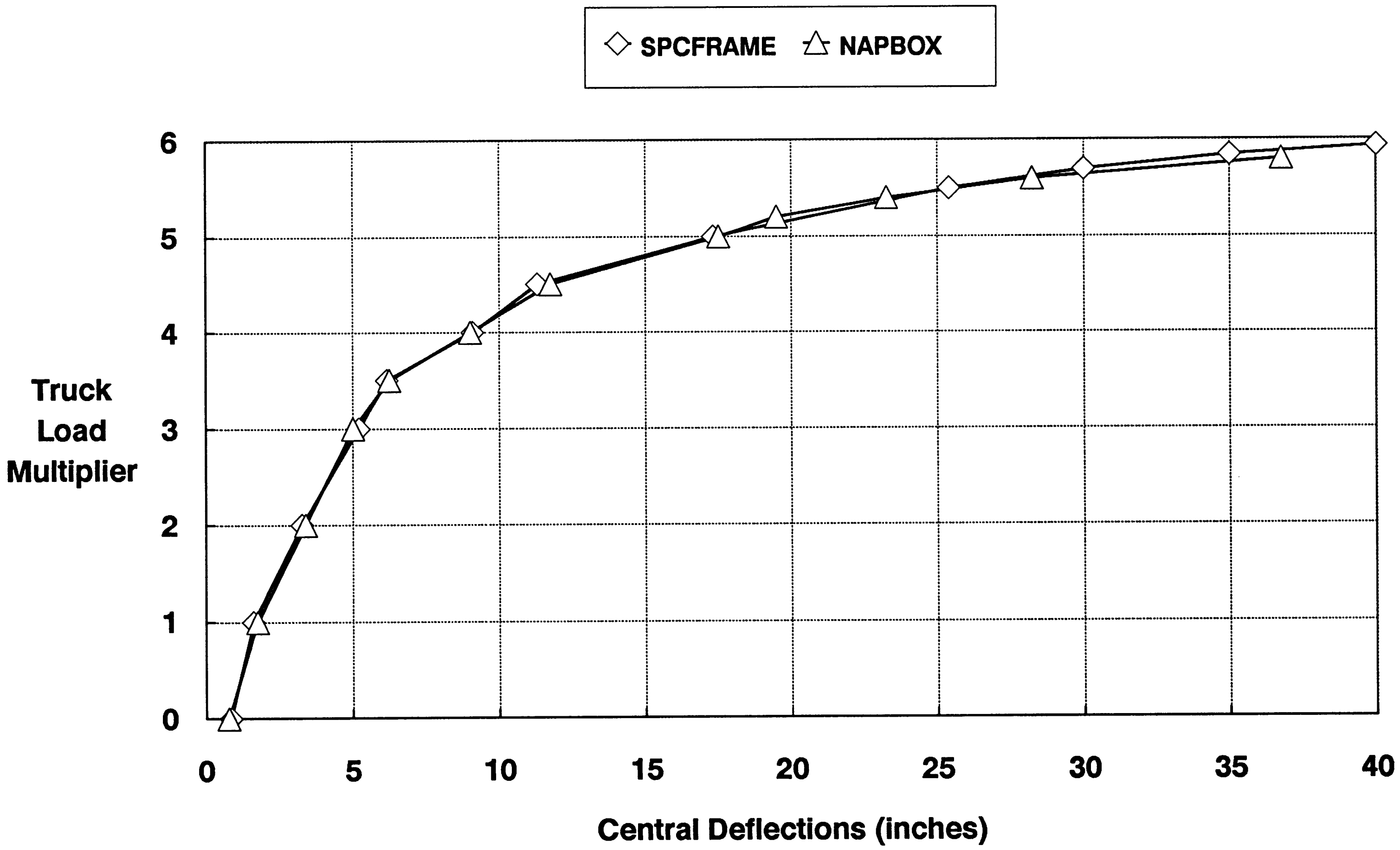


FIG. 3.3 INPUT DATA LISTINGS OF EXAMPLES (1 of 5)

EXAMPLE 1 - ULTIMATE LOAD ANALYSIS AFTER CONSTRUCTION

```
start     ; EX1
title n=3
   Example 1 - Choudhury Bridge                          1/17/89   Kang
               Three Span Continuous Post-tensioned Box Girder Bridge
               Ultimate Load Analysis Immediately after Construction
unit u=kip-in
concrete parameters n=1 ! 1 m=aci w=143.96 x=7.9057

mesh input
nodes n=21 ! 1 x=0 y=0 s=12 ! 5 x=4*20 g=1,5 ! 9 x=4*16+80 g=5,9
   11 x=2*8+144 g=9,11 ! 13 x=2*10+160 g=11,13 ! 15 x=2*13+180 g=13,15
   21 x=6*9+206 g=15,21 !
concrete properties n=1 ! 1 f=3.975297 w=0.155/1728 t=2 ! u=0.0041 e=0.004
mild steel properties n=1 ! 1 e=29.e3 y=60. u=0.03
prestressing steel n=1 p=5 ! 1 c=0.25 w=0.0002/12
   f=196.6,220,240,245,270 ! e=.00715,0.009,0.0115,0.0135,0.058
section properties n=1 c=10 s=5 ! 1 k=1
   a=2040,2040,450,450,450,450,450,450,804,804
   y=37.5,32.5,22.5,7.5,-7.5,-22.5,-37.5,-52.5,-62.0,-66.0
   b=12.24,2.70,2.70,2.70,4.824
   z=35,15,-15,-45,-64 ! m=1,1,1,1,1
frame elements n=20 ! 1,1,2 x=1 d=0 g=1,20,1,1,1,0,0 !
tendon geometry n=1 ! 1 s=2 m=1 a=26.32
   1 n=11 g= 1,11 ! r=0.0,0.5,0.1 s= 0,-55, 29
   2 n=11 g=11,21 ! r=0.2,1.0,1.0 s=29,-55,-55
mesh complete

set d=28 c=0.01,0.05,0.05,0.0001,10.0,1000.0,10.0,0.5

change structure
build n=1,20,1 d=0
restraints ! 1,11,10 r=0,1,0 ! 21 r=1,0,1 !
stress n=1 f=5320 d=0.25
change complete

solve q=2
loading ! n=15,19,2 f=0,-48 ! n=21 f=0,-24 !
solve l=8 p=2 ! f=1,1,1,0.5,0.5,0.5,0.5,0.5
loading ! n=15,19,2 f=0,-48 ! n=21 f=0,-24 !
solve c=1 n=21 o=2 a=-1. l=2 p=2 ! f=4.6,5
loading ! n=15,19,2 f=0,-48 ! n=21 f=0,-24 !
solve c=1 n=21 o=2 a=-5. p=2 q=2

stop
```

EXAMPLE 1 - CREEP ANALYSIS UP TO 10000 DAYS

```
start     ; EX1
title n=3
   Example 1 - Choudhury Bridge                          1/17/89   Kang
               Three Span Continuous Post-tensioned Box Girder Bridge
               Creep Analysis up to 10000 Days (27.4 Years)
unit u=kip-in
concrete parameters n=1 ! 1 m=aci w=143.96 x=7.9057

mesh input
nodes n=21 ! 1 x=0 y=0 s=12 ! 5 x=4*20 g=1,5 ! 9 x=4*16+80 g=5,9
   11 x=2*8+144 g=9,11 ! 13 x=2*10+160 g=11,13 ! 15 x=2*13+180 g=13,15
   21 x=6*9+206 g=15,21 !
concrete properties n=1 ! 1 f=3.975297 c=2.35 s=0.0008 w=0.155/1728 t=2
   u=0.0041 e=0.004
mild steel properties n=1 ! 1 e=29.e3 y=60. u=0.03
prestressing steel n=1 p=5 ! 1 c=0.25 w=0.0002/12 f=210
   f=196.6,220,240,245,270 ! e=.00715,0.009,0.0115,0.0135,0.058
section properties n=1 c=10 s=5 ! 1 k=1
   a=2040,2040,450,450,450,450,450,450,804,804
   y=37.5,32.5,22.5,7.5,-7.5,-22.5,-37.5,-52.5,-62.0,-66.0
   b=12.24,2.70,2.70,2.70,4.824
   z=35,15,-15,-45,-64 ! m=1,1,1,1,1
frame elements n=20 ! 1,1,2 x=1 d=0 g=1,20,1,1,1,0,0 !
tendon geometry n=1 ! 1 s=2 m=1 a=26.32
   1 n=11 g= 1,11 ! r=0.0,0.5,0.1 s=0,-55,29
   2 n=11 g=11,21 ! r=0.2,1.0,1.0 s=29,-55,-55
mesh complete

set d=28 c=0.01,0.05,0.05,0.01,10.0,1000.0,10.0,0.5 i=30,20

change structure
build n=1,20,1 d=0
restraints ! 1,11,10 r=0,1,0 ! 21 r=1,0,1 !
stress n=1 f=5320 d=0.25
change complete

solve
solve t=1 d=100 s=10 p=-1
solve t=1 d=200 s= 7 p=-1
solve t=1 d=10000 s=10 p=0 q=2

stop
```

EXAMPLE 1 - ULTIMATE LOAD ANALYSIS AFTER 10000 DAYS

```
start r=1    ; EX1
title n=3
   Example 1 - Choudhury Bridge                          1/17/89  Kang
               Ultimate Load Analysis at 10000 Days after Construction
               Execution Restarted at 10000 Days after Construction

loading ! n=15,19,2 f=0,-48 ! n=21 f=0,-24 !
solve l=8 p=2 ! f=1,1,1,0.5,0.5,0.5,0.5,0.1
loading ! n=15,19,2 f=0,-4.8 ! n=21 f=0,-2.4 !
solve q=2

stop
```

FIG. 3.3 INPUT DATA LISTINGS OF EXAMPLES (2 of 5)

EXAMPLE 2 - POST-BUCKLING ANALYSIS BY DISPLACEMENT CONTROL

```
start     ; EX2
title n=3
   Example 2 - Aroni Column              12/13/88  Kang
               Eccentrically Loaded Pre-tensioned Column
               Post-buckling Analysis by Displacement Control
unit u=N-mm
concrete parameters n=1 ! 1 m=aci w=158.9

mesh input
nodes n=11 ! 1 c=0,0 s=25.4 ! 2 c=4.,4.75e-2 ! 3 c=8.,9.e-2 ! 4 c=12.,0.1275
   5 c=16.,0.16 !  6 c=20.,0.1875 !  7 c=24.,0.21 ! 8 c=28.,0.2275
   9 c=32.,0.24 ! 10 c=36.,0.2475 ! 11 c=40.,0.25 !
concrete properties n=1
   1 f=38.27 t=2 ! u=0.0061 e=0.006
prestressing steel n=1 p=5 ! 1
   f=1007,1351,1503,1655,1731
   e=4.976e-3,6.9e-3,8.63e-3,3.0e-2,5.6e-2
section properties n=1 c=10 ! 1 k=1
   a=390.97,390.97,390.97,390.97,390.97,390.97,390.97,390.97,390.97,390.97
   y=23.09,17.96,12.83,7.70,2.57,-2.57,-7.70,-12.83,-17.96,-23.09
frame elements n=10
   1,1,2 x=1 d=0 g=1,10,1,1,1,0,0,0,0 !
tendon geometry n=2
   1 s=1 m=1 a=39.742 ! 1 n=11 g=1,11,1 ! r=0,1.,0 s= 12.7,19.05,19.05
   2 s=1 m=1 a=39.742 ! 1 n=11 g=1,11,1 ! r=0,1.,0 s=-12.7,-6.35,-6.35
mesh complete

set d=28 c=0.01,0.02,0.02,1.e-4,1.e2,1.e4,20.0,0.2

change structure
build n=1,10,1 d=0
restraints ! 1 r=0,1,0 ! 11 r=1,0,1 !
stress n=1,2,1 t=1 f=21237
change complete

solve g=1 x=1 p=2 q=2
loading ! n=1 f=1000,0,38100 !
solve c=1 l=12 g=1 x=1 n=11 o=2 a=5.0 p=2
    f=1,1,1,1,1,1,1,1,1,1,1,1
loading ! n=1 f=1000,0,38100 !
solve c=1 g=1 x=1 n=11 o=2 a=5.0 q=2

stop
```

EXAMPLE 3 - ANALYSIS UP TO COMPLETION OF CONSTRUCTION

```
START       ; EX3
TITLE N=4
   EXAMPLE 3 - KETCHUM BRIDGE                                 1/21/89  KANG
   THREE SPAN CANTILEVER CONSTRUCTION POST-TENSIONED BOX GIRDER BRIDGE
   CAST IN PLACE CONSTRUCTION WITH TRAVELERS, 7 DAY CYCLE
   ANALYSIS UP TO COMPLETION OF CONSTRUCTION
CONCRETE PARAMETERS N=1 ! 1

MESH INPUT
NODES N=43
   1  X=0     Y=-46.74  !   2  X=180   Y=-40.75  !   3  X=360   Y=-34.50
   4  X=540   Y=-34.50  !   5  X=720   Y=-34.50  !   6  X=788   Y=-34.61
   7  X=948   Y=-35.37  !   8  X=1108  Y=-36.65  !   9  X=1268  Y=-38.53
  10  X=1428  Y=-43.70  !  11  X=1588  Y=-49.21  !  12  X=1748  Y=-55.10
  13  X=1908  Y=-61.40  !  14  X=2068  Y=-68.12  !  15  X=2228  Y=-75.29
  16  X=2388  Y=-82.90  !  17  X=2548  Y=-90.94  !  18  X=2708  Y=-99.41
  19  X=2868  Y=-108.29 !  20  X=3028  Y=-117.58 !  21  X=3188  Y=-127.25
  22  X=3348  Y=-137.29 !  23  X=3420  Y=-141.92 !  24  X=3492  Y=-137.29
  25  X=3652  Y=-127.25 !  26  X=3812  Y=-117.58 !  27  X=3972  Y=-108.29
  28  X=4132  Y=-99.41  !  29  X=4292  Y=-90.94  !  30  X=4452  Y=-82.90
  31  X=4612  Y=-75.29  !  32  X=4772  Y=-68.12  !  33  X=4932  Y=-61.40
  34  X=5092  Y=-55.10  !  35  X=5252  Y=-49.21  !  36  X=5412  Y=-43.70
  37  X=5572  Y=-38.53  !  38  X=5732  Y=-36.65  !  39  X=5892  Y=-35.37
  40  X=6052  Y=-34.61  !  41  X=6120  Y=-34.50  !  42  X=3420  Y=-300.0
  43  X=3420  Y=-900.0  !
SEQUENCE ! G=1,23,1 ! G=42,43 ! G=24,41,1 !
MILD STEEL N=1 ! 1 E=29E6 T=144E3 Y=50E3 U=0.2
CONCRETE N=1 ! 1 F=5000 C=2.35 S=0.0008 W=155/1728 T=2 ! U=0.0041 E=0.004 K=0
SECTION PROPERTIES N=24 C=16 S=8
  1 c=10 s=5 k=1     CANTILEVER SEGMENTS
    a=5347.500,5347.500, 564.867, 564.867, 564.867, 564.867, 564.867, 564.867\
      1572.570,1572.570
    y= 30.946,  23.759,  13.104,  -1.018, -15.139, -29.261, -43.383, -57.504\
      -66.815, -71.315
    b=2.14,0.23,0.23,0.23,0.63
    z=  27.35,    6.04, -22.20, -50.44, -69.07 !   m=1,1,1,1,1
  2 c=10 s=5 k=1
    a=5347.500,5347.500, 572.800, 572.800, 572.800, 572.800, 572.800, 572.800\
      1570.815,1570.815
    y= 31.326,  24.139,  13.385,   -.935, -15.255, -29.575, -43.895, -58.215\
      -67.625, -72.125
    b=2.14,0.23,0.23,0.23,0.63
    z=  27.73,    6.22, -22.42, -51.06, -69.88 !   m=1,1,1,1,1
  3 c=10 s=5 k=1
    a=5347.500,5347.500, 594.133, 594.133, 594.133, 594.133, 594.133, 594.133\
      1566.000,1566.000
    y= 32.356,  25.169,  14.148,   -.705, -15.558, -30.412, -45.265, -60.118\
      -69.795, -74.295
    b=2.14,0.24,0.24,0.24,0.63
    z=  28.76,    6.72, -22.98, -52.69, -72.05 !   m=1,1,1,1,1
  4 c=10 s=5 k=1
    a=5347.500,5347.500, 625.133, 625.133, 625.133, 625.133, 625.133, 625.133\
      1559.025,1559.025
    y= 33.866,  26.679,  15.271,   -.358, -15.986, -31.614, -47.242, -62.871\
      -72.935, -77.435
    b=2.14,0.25,0.25,0.25,0.62
    z=  30.27,    7.46, -23.80, -55.06, -75.19 !   m=1,1,1,1,1
  5 c=10 s=5 k=1
    a=5347.500,5347.500, 655.733, 655.733, 655.733, 655.733, 655.733, 655.733\
      1782.099,1782.099
    y= 37.476,  30.289,  18.498,   2.105, -14.288, -30.682, -47.075, -63.468\
```

FIG. 3.3 INPUT DATA LISTINGS OF EXAMPLES (3 of 5)

```
      -74.250, -79.420
    b=2.14,0.26,0.26,0.26,0.71
    z=  33.88,  10.30, -22.49, -55.27, -76.84 !   m=1,1,1,1,1
 6 c=10 s=5 k=1
    a=5347.500,5347.500, 686.800, 686.800, 686.800, 686.800, 686.800, 686.800\
      2191.822,2191.822
    y=  42.816,  35.629,  23.450,   6.280, -10.890, -28.060, -45.230, -62.400\
      -74.183, -80.578
    b=2.14,0.27,0.27,0.27,0.88
    z=  39.22,  14.86, -19.48, -53.82, -77.38 !   m=1,1,1,1,1
 7 c=12 s=6 k=1
    a=5347.500,5347.500, 621.543, 621.543, 621.543, 621.543, 621.543, 621.543\
       621.543,1728.148,1728.148,1728.148
    y=  48.506,  41.319,  29.956,  14.417,  -1.121, -16.660, -32.199, -47.737\
      -63.276, -73.583, -78.660, -83.737
    b=2.14,0.29,0.29,0.29,0.52,0.52
    z=  44.91,  14.42, -16.66, -47.74, -76.12, -81.20 !   m=1,1,1,1,1,1
 8 c=12 s=6 k=1
    a=5347.500,5347.500, 660.286, 660.286, 660.286, 660.286, 660.286, 660.286\
       660.286,1990.473,1990.473,1990.473
    y=  54.606,  47.419,  35.571,  19.064,   2.557, -13.950, -30.457, -46.964\
      -63.471, -74.672, -80.565, -86.458
    b=2.14,0.31,0.31,0.31,0.60,0.60
    z=  51.01,  19.06, -13.95, -46.96, -77.62, -83.51 !   m=1,1,1,1,1,1
 9 c=12 s=6 k=1
    a=5347.500,5347.500, 704.629, 704.629, 704.629, 704.629, 704.629, 704.629\
       704.629,2245.057,2245.057,2245.057
    y=  61.106,  53.919,  41.517,  23.901,   6.286, -11.330, -28.946, -46.561\
      -64.177, -76.338, -83.045, -89.752
    b=2.14,0.33,0.33,0.33,0.67,0.67
    z=  57.51,  23.90, -11.33, -46.56, -79.69, -86.40 !   m=1,1,1,1,1,1
10 c=12 s=6 k=1
    a=5347.500,5347.500, 754.343, 754.343, 754.343, 754.343, 754.343, 754.343\
       754.343,2493.534,2493.534,2493.534
    y=  68.066,  60.879,  47.856,  28.997,  10.139,  -8.720, -27.579, -46.437\
      -65.296, -78.487, -86.010, -93.533
    b=2.14,0.35,0.35,0.35,0.75,0.75
    z=  64.47,  29.00,  -8.72, -46.44, -82.25, -89.77 !   m=1,1,1,1,1,1
11 c=14 s=6 k=1
    a=5347.500,5347.500, 708.150, 708.150, 708.150, 708.150, 708.150, 708.150\
       708.150, 708.150,2049.757,2049.757,2049.757,2049.757
    y=  75.446,  68.259,  55.813,  38.109,  20.406,   2.702, -15.002, -32.706\
      -50.409, -68.113, -80.091, -86.344, -92.596, -98.849
    b=2.14,0.38,0.38,0.38,0.82,0.82
    z=  71.85,  38.11,   2.70, -32.71, -80.09, -92.60 !   m=1,1,1,1,1,1
12 c=14 s=6 k=1
    a=5347.500,5347.500, 760.650, 760.650, 760.650, 760.650, 760.650, 760.650\
       760.650, 760.650,2223.711,2223.711,2223.711,2223.711
    y=  83.276,  76.089,  62.987,  43.971,  24.954,   5.938, -13.078, -32.094\
      -51.111, -70.127, -83.068, -89.933, -96.798,-103.663
    b=2.14,0.41,0.41,0.41,0.89,0.89
    z=  79.68,  43.97,   5.94, -32.09, -83.07, -96.80 !   m=1,1,1,1,1,1
13 c=14 s=6 k=1
    a=5347.500,5347.500, 817.400, 817.400, 817.400, 817.400, 817.400, 817.400\
       817.400, 817.400,2389.982,2389.982,2389.982,2389.982
    y=  91.526,  84.339,  70.527,  50.092,  29.657,   9.222, -11.213, -31.648\
      -52.083, -72.518, -86.473, -93.948,-101.423,-108.898
    b=2.14,0.44,0.44,0.44,0.96,0.96
    z=  87.93,  50.09,   9.22, -31.65, -86.47,-101.42 !   m=1,1,1,1,1,1
14 c=14 s=6 k=1
    a=5347.500,5347.500, 878.350, 878.350, 878.350, 878.350, 878.350, 878.350\
       878.350, 878.350,2548.877,2548.877,2548.877,2548.877
    y= 100.206,  93.019,  78.446,  56.487,  34.528,  12.569,  -9.389, -31.348\
      -53.307, -75.266, -90.288, -98.373,-106.458,-114.543
    b=2.14,0.47,0.47,0.47,1.02,1.02
    z=  96.61,  56.49,  12.57, -31.35, -90.29,-106.46 !   m=1,1,1,1,1,1
```

```
15 c=14 s=6 k=1
    a=5347.500,5347.500, 943.400, 943.400, 943.400, 943.400, 943.400, 943.400\
       943.400, 943.400,2700.748,2700.748,2700.748,2700.748
    y= 109.306, 102.119,  86.732,  63.147,  39.562,  15.977,  -7.608, -31.193\
      -54.778, -78.363, -94.504,-103.201,-111.899,-120.596
    b=2.14,0.50,0.50,0.50,1.08,1.08
    z= 105.71,  63.15,  15.98, -31.19, -94.50,-111.90 !   m=1,1,1,1,1,1
16 c=14 s=6 k=1
    a=5347.500,5347.500,1012.400,1012.400,1012.400,1012.400,1012.400,1012.400\
      1012.400,1012.400,2843.534,2843.534,2843.534,2843.534
    y= 118.806, 111.619,  95.370,  70.060,  44.750,  19.440,  -5.870, -31.180\
      -56.490, -81.800, -99.109,-108.416,-117.724,-127.031
    b=2.14,0.54,0.54,0.54,1.14,1.14
    z= 115.21,  70.06,  19.50, 122.40, 275.98, 343.91 !   m=1,1,1,1,1,1
17 c=16 s=8 k=1
    a=5347.500,5347.500, 964.711, 964.711, 964.711, 964.711, 964.711, 964.711\
       964.711, 964.711, 964.711,2382.705,2382.705,2382.705,2382.705,2382.705
    y= 128.606, 121.419, 105.766,  81.648,  57.531,  33.413,   9.295, -14.823\
      -38.941, -63.058, -87.176,-103.203,-111.139,-119.075,-127.011,-134.947
    b=2.14,0.43,0.43,0.43,0.43,0.79,0.79,0.79
    z= 125.01,  81.65,  33.41, -14.82, -63.06,-103.20,-119.08,-134.95
    m=1,1,1,1,1,1,1,1
18 c=16 s=8 k=1     PIER TABLE
    a=5347.500,5347.500,1013.822,1013.822,1013.822,1013.822,1013.822,1013.822\
      1013.822,1013.822,1013.822,2455.995,2455.995,2455.995,2455.995,2455.995
    y= 136.006, 128.819, 112.552,  87.207,  61.861,  36.516,  11.170, -14.176\
      -39.521, -64.867, -90.212,-107.030,-115.320,-123.610,-131.900,-140.190
    b=2.14,0.46,0.46,0.46,0.46,0.82,0.82,0.82
    z= 132.41,  87.21,  36.52, -14.18, -64.87,-107.03,-123.61,-140.19
    m=1,1,1,1,1,1,1,1
19 c=16 s=8 k=1     RIGID ZONE
    a=5347.500,5347.500,1029.422,1029.422,1029.422,1029.422,1029.422,1029.422\
      1029.422,1029.422,1029.422,2478.000,2478.000,2478.000,2478.000,2478.000
    y= 138.306, 131.119, 114.657,  88.922,  63.186,  37.451,  11.715, -14.021\
      -39.756, -65.492, -91.227,-108.295,-116.695,-125.095,-133.495,-141.895
    b=10.70,2.32,2.32,2.32,2.32,4.13,4.13,4.1
    z= 134.71,  88.92,  37.45, -14.02, -65.49,-108.30,-125.10,-141.90
    m=1,1,1,1,1,1,1,1
20 c=11 s=5 k=1     NEAR ABUTMENT
    a=5347.500,5347.500, 493.333, 493.333, 493.333, 493.333, 493.333, 493.333\
      2298.477,2298.477,2298.477
    y=  40.496,  33.309,  23.548,  11.215,  -1.118, -13.452, -25.785, -38.118\
      -47.557, -54.100, -60.643
    b=2.14,0.20,0.20,0.20,1.38
    z=  36.90,  17.38,  -7.29, -31.95, -54.10 !   m=1,1,1,1,1
21 c=10 s=5 k=1
    a=5347.500,5347.500, 551.667, 551.667, 551.667, 551.667, 551.667, 551.667\
      1902.966,1902.966
    y=  32.996,  25.809,  15.319,   1.527, -12.264, -26.056, -39.848, -53.639\
      -63.255, -68.695
    b=2.14,0.22,0.22,0.22,0.76
    z=  29.40,   8.42, -19.16, -46.74, -65.98 !   m=1,1,1,1,1
22 c=10 s=5 k=1
    a=5347.500,5347.500, 564.200, 564.200, 564.200, 564.200, 564.200, 564.200\
      1572.750,1572.750
    y=  30.906,  23.719,  13.072,  -1.033, -15.138, -29.243, -43.348, -57.452\
      -66.755, -71.255
    b=2.14,0.23,0.23,0.23,0.63
    z=  27.31,   6.02, -22.19, -50.40, -69.01 !   m=1,1,1,1,1
23 c=10 s=5 k=1
    a=5347.500,5347.500, 564.200, 564.200, 564.200, 564.200, 564.200, 564.200\
      1572.750,1572.750
    y=  30.906,  23.719,  13.072,  -1.033, -15.138, -29.243, -43.348, -57.452\
      -66.755, -71.255
    b=2.14,0.23,0.23,0.23,0.63
    z=  27.31,   6.02, -22.19, -50.40, -69.01 !   m=1,1,1,1,1
```

FIG. 3.3 INPUT DATA LISTINGS OF EXAMPLES (4 of 5)

```
24 c=10 s=5 k=1     PIER
   a=3340.8,3340.8,3340.8,3340.8,3340.8,3340.8,3340.8,3340.8,3340.8,3340.8
   y=43.2,33.6,24.0,14.4,4.8,-4.8,-14.4,-24.0,-33.6,-43.2
   b=6.68,6.68,6.68,6.68,6.68
   z=38.4,19.2,0.0,-19.2,-38.4 !   m=1,1,1,1,1
FRAME ELEMENTS N=42
 1,1,2    X=20 D=100 G=1,4,1,1,1,1,0
 5,5,6    X=1  D=168 G=5,22,1,1,1,1,-7
 23,23,24 X=18 D=49  G=23,40,1,1,1,-1,7
 41,23,42 X=19 D=0 ! 42,42,43 X=24 D=0      !
PRESTRESSING STEEL N=1 P=5 ! 1 C=.25 W=.0004/12 F=21E4
 F=1.96E5,2.2E5,2.4E5,2.45E5,2.7E5 ! E=0.007,0.009,0.0115,0.0135,0.058
TENDONS N=30
; CANTILEVER TENDONS - EACH REPRESENTS 4, 21-STRAND, 1/2" DIAM TENDONS
  1 S=2 M=1 A=12.852 ! 1 N=3  G=21,23,1 ! R=0,.6638,0 S=-20,-7,-7 \
                     ! 2 N=3  G=23,25,1 ! R=0,.3362,0 S=-7,-7,-20
  2 S=2 M=1 A=12.852 ! 1 N=4  G=20,23,1 ! R=0,.3929,0 S=-20,-7,-7 \
                     ! 2 N=4  G=23,26,1 ! R=0,.6071,0 S=-7,-7,-20
  3 S=2 M=1 A=12.852 ! 1 N=5  G=19,23,1 ! R=0,.2790,0 S=-20,-7,-7 \
                     ! 2 N=5  G=23,27,1 ! R=0,.7210,0 S=-7,-7,-20
  4 S=2 M=1 A=12.852 ! 1 N=6  G=18,23,1 ! R=0,.2163,0 S=-20,-7,-7 \
                     ! 2 N=6  G=23,28,1 ! R=0,.7837,0 S=-7,-7,-20
  5 S=2 M=1 A=12.852 ! 1 N=7  G=17,23,1 ! R=0,.1766,0 S=-20,-7,-7 \
                     ! 2 N=7  G=23,29,1 ! R=0,.8234,0 S=-7,-7,-20
  6 S=2 M=1 A=12.852 ! 1 N=8  G=16,23,1 ! R=0,.1492,0 S=-20,-7,-7 \
                     ! 2 N=8  G=23,30,1 ! R=0,.8508,0 S=-7,-7,-20
  7 S=2 M=1 A=12.852 ! 1 N=9  G=15,23,1 ! R=0,.1292,0 S=-20,-7,-7 \
                     ! 2 N=9  G=23,31,1 ! R=0,.8708,0 S=-7,-7,-20
  8 S=2 M=1 A=12.852 ! 1 N=10 G=14,23,1 ! R=0,.1139,0 S=-20,-7,-7 \
                     ! 2 N=10 G=23,32,1 ! R=0,.8861,0 S=-7,-7,-20
  9 S=2 M=1 A=12.852 ! 1 N=11 G=13,23,1 ! R=0,.1019,0 S=-20,-7,-7 \
                     ! 2 N=11 G=23,33,1 ! R=0,.8981,0 S=-7,-7,-20
 10 S=2 M=1 A=12.852 ! 1 N=12 G=12,23,1 ! R=0,.0921,0 S=-20,-7,-7 \
                     ! 2 N=12 G=23,34,1 ! R=0,.9079,0 S=-7,-7,-20
 11 S=2 M=1 A=12.852 ! 1 N=13 G=11,23,1 ! R=0,.0841,0 S=-20,-7,-7 \
                     ! 2 N=13 G=23,35,1 ! R=0,.9159,0 S=-7,-7,-20
 12 S=2 M=1 A=12.852 ! 1 N=14 G=10,23,1 ! R=0,.0773,0 S=-20,-7,-7 \
                     ! 2 N=14 G=23,36,1 ! R=0,.9277,0 S=-7,-7,-20
 13 S=2 M=1 A=12.852 ! 1 N=15 G= 9,23,1 ! R=0,.0716,0 S=-20,-7,-7 \
                     ! 2 N=15 G=23,37,1 ! R=0,.9284,0 S=-7,-7,-20
 14 S=2 M=1 A=12.852 ! 1 N=16 G= 8,23,1 ! R=0,.0666,0 S=-20,-7,-7 \
                     ! 2 N=16 G=23,38,1 ! R=0,.9334,0 S=-7,-7,-20
 15 S=2 M=1 A=12.852 ! 1 N=17 G= 7,23,1 ! R=0,.0623,0 S=-20,-7,-7 \
                     ! 2 N=17 G=23,39,1 ! R=0,.9377,0 S=-7,-7,-20
 16 S=2 M=1 A=12.852 ! 1 N=18 G= 6,23,1 ! R=0,.0585,0 S=-20,-7,-7 \
                     ! 2 N=18 G=23,40,1 ! R=0,.9415,0 S=-7,-7,-20
; CONTINUITY TENDONS - EACH REPRESENTS 8, 21-STRAND, 1/2" DIAM TENDONS
 17 S=2 M=1 A=25.704 ! 1 N=18 G= 1,18,1 ! R=0,.4,.2    S=-46.74,-100,-12 \
                     ! 2 N=6  G=18,23,1 ! R=0,.5,0     S=-12,-12,-12
 18 S=2 M=1 A=25.704 ! 1 N=6  G=23,28,1 ! R=0,.5,0     S=-12,-12,-12 \
                     ! 2 N=14 G=28,41,1 ! R=.3,1,0     S=-12,-100,-100
; LOCAL TENDONS - EACH REPRESENTS 2 OR 4, 21-STRAND, 1/2" DIAM TENDONS
 19 S=1 M=1 A=6.426  ! 1 N=9  G= 1, 9,1 ! R=0,.874,0  S=-46.74,-98.15,-103.34
 20 S=1 M=1 A=6.426  ! 1 N=10 G= 1,10,1 ! R=0,.776,0  S=-46.74,-98.15,-107.43
 21 S=1 M=1 A=6.426  ! 1 N=11 G= 1,11,1 ! R=0,.698,0  S=-46.74,-98.15,-112.64
 22 S=1 M=1 A=6.426  ! 1 N=12 G= 1,12,1 ! R=0,.634,0  S=-46.74,-98.15,-118.91
 23 S=1 M=1 A=6.426  ! 1 N=12 G=30,41,1 ! R=0,1,0     S=-153.64,-100,-100
 24 S=1 M=1 A=6.426  ! 1 N=11 G=31,41,1 ! R=0,1,0     S=-143.58,-100,-100
 25 S=1 M=1 A=12.852 ! 1 N=10 G=32,41,1 ! R=0,1,0     S=-134.42,-100,-100
 26 S=1 M=1 A=12.852 ! 1 N= 9 G=33,41,1 ! R=0,1,0     S=-126.18,-100,-100
 27 S=1 M=1 A=12.852 ! 1 N= 8 G=34,41,1 ! R=0,1,0     S=-118.91,-100,-100
 28 S=1 M=1 A=12.852 ! 1 N= 7 G=35,41,1 ! R=0,1,0     S=-112.64,-100,-100
 29 S=1 M=1 A=6.426  ! 1 N= 6 G=36,41,1 ! R=0,1,0     S=-107.43,-100,-100
 30 S=1 M=1 A=6.426  ! 1 N= 5 G=37,41,1 ! R=0,1,0     S=-103.34,-100,-100
TRAVELERS N=3 ! 1 A=696 I=1.081E6 E=29E6 W=15E4 N=3
 2 A=696 I=1.081E6 E=29E6 W=15E4 N=3 ! 3 A=696 I=1.081E6 E=29E6 W=1E4 N=2
```

```
MESH COMPLETE

SET D=56 C=0.01,0.02,0.02,0.001,1.E5,1.E7,10.0,0.5 I=30,20

; BUILD PIER AND STARTING SEGMENT
CHANGE STRUCTURE !RESTRAINTS ! 1,5 R=0,1,0 !41 R=1,0,1 !43 R=1,1,1 !
BUILD N=41,42 !BUILD N=21,24 D=35
CHANGE COMPLETE !SOLVE T=1 P=-1 !SOLVE A=0 D=63 T=1 P=-1

; STRESS TENDON 1, BUILD TRAVELERS, BUILD SEGMENTS 20 AND 25
CHANGE STRUCTURE !STRESS N=1  S=198E3,198E3 D=.25,.25
MOVE N=1 D=20,21,22 !MOVE N=2 D=24,25,26 !CHANGE COMPLETE !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=20,25,5 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=70 T=1 P=-1
; STRESS TENDON 2, MOVE TRAVELERS, BUILD SEGMENTS 19 AND 26
CHANGE STRUCTURE !STRESS N=2  S=198E3,198E3 D=.25,.25
MOVE N=1 D=19,20,21 !MOVE N=2 D=25,26,27 !CHANGE COMPLETE !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=19,26,7 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=77 T=1 P=-1
; STRESS TENDON 3, MOVE TRAVELERS, BUILD SEGMENTS 18 AND 27
CHANGE STRUCTURE !STRESS N=3  S=198E3,198E3 D=.25,.25
MOVE N=1 D=18,19,20 !MOVE N=2 D=26,27,28 !CHANGE COMPLETE !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=18,27,9 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=84 T=1 P=-1
; STRESS TENDON 4, MOVE TRAVELERS, BUILD SEGMENTS 17 AND 28
CHANGE STRUCTURE !STRESS N=4  S=198E3,198E3 D=.25,.25
MOVE N=1 D=17,18,19 !MOVE N=2 D=27,28,29 !CHANGE COMPLETE  !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=17,28,11 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=91 T=1 P=-1
; STRESS TENDON 5, MOVE TRAVELERS, BUILD SEGMENTS 16 AND 29
CHANGE STRUCTURE !STRESS N=5  S=198E3,198E3 D=.25,.25
MOVE N=1 D=16,17,18 !MOVE N=2 D=28,29,30 !CHANGE COMPLETE  !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=16,29,13 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=98 T=1 P=-1
; STRESS TENDON 6, MOVE TRAVELERS, BUILD SEGMENTS 15 AND 30
CHANGE STRUCTURE !STRESS N=6  S=198E3,198E3 D=.25,.25
MOVE N=1 D=15,16,17 !MOVE N=2 D=29,30,31 !CHANGE COMPLETE  !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=15,30,15 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=105 T=1 P=-1
; STRESS TENDON 7, MOVE TRAVELERS, BUILD SEGMENTS 14 AND 31
CHANGE STRUCTURE !STRESS N=7  S=198E3,198E3 D=.25,.25
MOVE N=1 D=14,15,16 !MOVE N=2 D=30,31,32 !CHANGE COMPLETE  !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=14,31,17 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=112 T=1 P=-1
; STRESS TENDON 8, MOVE TRAVELERS, BUILD SEGMENTS 13 AND 32
CHANGE STRUCTURE !STRESS N=8  S=198E3,198E3 D=.25,.25
MOVE N=1 D=13,14,15 !MOVE N=2 D=31,32,33 !CHANGE COMPLETE  !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=13,32,19 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=119 T=1 P=-1
; STRESS TENDON 9, MOVE TRAVELERS, BUILD SEGMENTS 12 AND 33
CHANGE STRUCTURE !STRESS N=9  S=198E3,198E3 D=.25,.25
MOVE N=1 D=12,13,14 !MOVE N=2 D=32,33,34 !CHANGE COMPLETE  !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=12,33,21 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=126 T=1 P=-1
; STRESS TENDON 10, MOVE TRAVELERS, BUILD SEGMENTS 11 AND 34
CHANGE STRUCTURE !STRESS N=10 S=198E3,198E3 D=.25,.25
MOVE N=1 D=11,12,13 !MOVE N=2 D=33,34,35 !CHANGE COMPLETE  !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=11,34,23 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=133 T=1 P=-1
; STRESS TENDON 11, MOVE TRAVELERS, BUILD SEGMENTS 10 AND 35
CHANGE STRUCTURE !STRESS N=11 S=198E3,198E3 D=.25,.25
MOVE N=1 D=10,11,12 !MOVE N=2 D=34,35,36 !CHANGE COMPLETE  !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=10,35,25 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=140 T=1 P=-1
; STRESS TENDON 12, MOVE TRAVELERS, BUILD SEGMENTS 9 AND 36
CHANGE STRUCTURE !STRESS N=12 S=198E3,198E3 D=.25,.25
```

FIG. 3.3 INPUT DATA LISTINGS OF EXAMPLES (5 of 5)

```
MOVE N=1 D=9,10,11 !MOVE N=2 D=35,36,37 !CHANGE COMPLETE  !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=9,36,27 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=147 T=1 P=-1
; STRESS TENDON 13, MOVE TRAVELERS, BUILD SEGMENTS 8 AND 37
CHANGE STRUCTURE !STRESS N=13 S=198E3,198E3 D=.25,.25
MOVE N=1 D=8,9,10 !MOVE N=2 D=36,37,38 !CHANGE COMPLETE   !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=8,37,29 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=154 T=1 P=-1
; STRESS TENDON 14, MOVE TRAVELERS, BUILD SEGMENTS 7 AND 38
CHANGE STRUCTURE !STRESS N=14 S=198E3,198E3 D=.25,.25
MOVE N=1 D=7,8,9 !MOVE N=2 D=37,38,39 !CHANGE COMPLETE    !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=7,38,31 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=161 T=1 P=-1
; STRESS TENDON 15, MOVE TRAVELERS, BUILD SEGMENTS 6 AND 39
CHANGE STRUCTURE !STRESS N=15 S=198E3,198E3 D=.25,.25
MOVE N=1 D=6,7,8 !MOVE N=2 D=38,39,40 !CHANGE COMPLETE    !SOLVE P=-1
CHANGE STRUCTURE !BUILD N=6,39,33 !CHANGE COMPLETE !SOLVE P=-1
                                                  SOLVE D=168 T=1 P=-1
; STRESS TENDON 16
CHANGE STRUCTURE !STRESS N=16 S=198E3,198E3 D=.25,.25
CHANGE COMPLETE  !SOLVE A=0.71 P=-1

; BUILD SEGMENTS 1 THRU 4, INSTALL CLOSURE FORMWORK, BUILD CLOSURE SEGMENT 5
CHANGE STRUCTURE !BUILD N=1,4 !MOVE N=3 D=5,6 !BUILD N=5
CHANGE COMPLETE  !SOLVE P=-1 !SOLVE D=175 S=3 T=1 P=-1

; REMOVE SUPPORTS AT NODES 2 THRU 5, STRESS TENDONS 19 THRU 22
CHANGE STRUCTURE !RESTRAINTS ! 2,5 R=0,0,0 !
STRESS N=19,22 S=198E3,198E3 D=.25,.25
CHANGE COMPLETE  !SOLVE P=-1 !SOLVE D=182 S=3 T=1 P=-1

; MOVE CLOSURE FORMWORK TO CENTER SPAN, BUILD CLOSURE SEGMENT 40
CHANGE STRUCTURE !MOVE N=3 D=40,41 !BUILD N=40 D=182
CHANGE COMPLETE  !SOLVE P=-1 !SOLVE D=189 S=3 T=1 P=-1

; STRESS TENDONS 23 THRU 30, REMOVE TRAVELERS AND CLOSURE FORMWORK
CHANGE STRUCTURE !STRESS N=23,30 S=198E3 D=.25 !MOVE N=1 D=0 !MOVE N=2 D=0
MOVE N=3 D=0 !CHANGE COMPLETE  !SOLVE P=-1 !SOLVE D=196 S=3 T=1 P=-1

; STRESS CONTINUITY TENDONS, ADD REMAINING DEAD LOAD
CHANGE STRUCTURE !STRESS N=17,18 S=198E3 D=.25 !CHANGE COMPLETE
LOADING ! L=1,40 F=0,-2500/12 !!SOLVE  P=2 Q=1

STOP
```

```
       EXAMPLE 3 - ULTIMATE LOAD ANALYSIS AT COMPLETION OF CONSTRUCTION


START R=1 ; EX3
TITLE N=5
   EXAMPLE 3 - KETCHUM BRIDGE                               1/21/89  KANG
   THREE SPAN CANTILEVER CONSTRUCTION POST-TENSIONED BOX GIRDER BRIDGE
   CAST IN PLACE CONSTRUCTION WITH TRAVELERS, 7 DAY CYCLE
   ULTIMATE LOAD ANALYSIS AT COMPLETION OF CONSTRUCTION
   ANALYSIS RESTARTED AT COMPLETION OF CONSTRUCTION

LOADING ! N=36 F=0,-30030 ! N=37 F=0,-31190 ! N=38 F=0,-34780
          N=39 F=0,-44400 ! N=40 F=0,-3600  ! N=41 F=0,-24000 !
SOLVE L=10 P=2 ! F=2,2,2,2,2,2,2,1,1,1
LOADING ! N=36 F=0,-30030 ! N=37 F=0,-31190 ! N=38 F=0,-34780
          N=39 F=0,-44400 ! N=40 F=0,-3600  ! N=41 F=0,-24000 !
SOLVE C=1 N=41 O=2 A=-1.34 Q=1

STOP


             EXAMPLE 3 - CREEP ANALYSIS UP TO 10000 DAYS


START R=1 ; EX3
TITLE N=5
   EXAMPLE 3 (KETCHUM BRIDGE)                          1/21/89  KANG
   THREE SPAN CANTILEVER CONSTRUCTION PRESTRESSED CONCRETE BRIDGE
   CAST IN PLACE CONSTRUCTION WITH TRAVELERS, 7 DAY CYCLE
   ANALYSIS RESTARTED AT COMPLETION OF CONSTRUCTION
   CREEP ANALYSIS UP TO 10000 DAYS

; STEP THROUGH TIME UP TO 10000 DAYS (27.4 YEARS)
SOLVE D=300 S=10 T=1 P=-1
SOLVE D=10000 S=10 T=1 P=0 Q=1

STOP
```

3.2 Description and Interpretation of Output

An edited version of the output for example 1 is shown in Fig. 3.4 (page 78). The data for the structural configuration, material properties and loading, control data for the nonlinear solution, which had been directly input or generated by the program, are printed. The summary output of the analysis for the dead load and the prestress is first printed.The summary output for the last analysis step, which represents the ultimate stage, will be explained in some detail.

The first information concerns the current solution state: these include the solution date, displacement (or load) step number, iteration number, displacement ratio, controlling DOF number for the displacement ratio, multiplier of the nodal load for the current displacement step, maximum unbalanced force, maximum unbalanced moment, and current total number of iterations performed in the analysis. These convergence informations can be noted to be within the limits set by convergence tolerances.

Nodal boundary conditions and current total displacements are printed next. Current total joint loads, unbalanced loads and reactions are given. These values are given in global coordinate system. In the displacement controlled solution current total joint loads are computed by the program. For this example the current total loads represent the truck load multiplier of 5.94. Restrained DOF are marked by the word "RESTRAINED", and the corresponding unbalanced load represents the reaction.

Frame element forces and moments are summarized next. These values are given in local element coordinate system (see Fig. 2.1(b)). Contributions of the concrete layers, mild steel layers and prestressing tendon segments are shown in addition to the total values. Contribution of travelers will be included if they are present. Maximum moment occurs at interior support location (joint number 11) as expected, where the unbalanced moment shows the maximum value also. This unbalanced moment is the difference between the "MOMENT-J" of element 10 and the "MOMENT-I" of element 11. This unbalanced moment could have been reduced had the lower value for the convergence tolerance "TOLM" been specified, in which case larger number of iterations would have been required.

Concrete and mild steel layer data are printed if requested. Only the data for element 11 at Gauss point number 2, and element 20 at Gauss point 3 are shown in the edited output. These are the locations of maximum negative and positive moment, respectively. Gauss integration points 1,2,3 from node i of the frame element on local x-axis are 0.5L, 0.11L and 0.89L, respectively, where L is the element length (Fig. 2.1(b)).

Material state code numbers, total stresses, total strains, total mechanical strains, total non-mechanical strains and total creep strains are given for each concrete and mild steel layer. Material state code numbers represent the current position of the material in the instantaneous stress-strain curve. The code numbers for each material are shown in Fig. 2.2.

Sixteen different material states are defined for concrete:

1 = In primary tension
2 = In primary compression, not yielded
3 = In primary compression, yielded
4 = In primary compression, yielded and in horizontal path
5 = In tension stiffening range
6 = Cracked
7 = Crushed
8 = In load reversal path from state 2
9 = In load reversal path from state 3
10 = In load reversal path from state 4
11 = In state 2, and has been cracked
12 = In state 3, and has been cracked
13 = In state 4, and has been cracked
14 = In state 8, and has been cracked
15 = In state 9, and has been cracked
16 = In state 10, and has been cracked

Four different material states are defined for the mild steel:

1 = In primary linearly elastic range
2 = Yielded
3 = In load reversal path
4 = Failed

Prestressing steel material state code numbers are as follows:

n = Between points (n-1) and n on multilinear stress-strain curve
-1 = In load reversal path
-2 = Failed in tension

Non-mechanical strains consist of creep, shrinkage, aging, temperature strains and strains due to imposed displacements for concrete, and temperature strains and strains due to imposed displacements for mild steel and prestressing steel.

It can be noted from the layer data of this example that in the maximum negative moment section at interior support, extensive cracking and yielding of materials have taken place, signifying the impending failure. The compressive stress in the bottom concrete layer of the section shows the maximum value in the structure. Prestressing tendon forces, strains and material state code numbers are also printed. Extensive yielding of material and the increase of prestressing forces can be noted in maximum moment regions.

Based on the above data it can be concluded that the failure is initiated by the crushing of concrete at interior support, and the corresponding truck load multiplier of 5.94 represents a reasonable estimate of the ultimate load. This can be further confirmed by the failure of convergence on the next displacement step.

The load - central deflection curves of this example for both SPCFRAME analysis and Choudhury's NAPBOX [12] analysis are shown in Fig. 3.2 (page 68). Very good agreement between the two results can be noted.

Theoretical aspects of these and other examples will be discussed in greater detail in the forthcoming report mentioned earlier.

FIG. 3.4 OUTPUT LISTING OF EXAMPLE 1 (1 of 7)

```
=============================================================================
=                                                                           =
=                             S P C F R A M E                               =
=                                                                           =
=    NONLINEAR ANALYSIS OF SEGMENTALLY ERECTED PRESTRESSED CONCRETE FRAMES  =
=                                                                           =
=                        VERSION 1.0,  JANUARY  1989                        =
=                                                                           =
=============================================================================


PROBLEM IDENTIFICATION:
=======================

   EXAMPLE 1 - CHOUDHURY BRIDGE                           1/17/89   KANG
               THREE SPAN CONTINUOUS POST-TENSIONED BOX GIRDER BRIDGE
               ULTIMATE LOAD ANALYSIS IMMEDIATELY AFTER CONSTRUCTION

UNIT U=KIP-IN


=============================================================================
= F R A M E   S T R U C T U R E   M E S H   S P E C I F I C A T I O N S   =
=============================================================================

NODE COORDINATES INPUT
======================

       NODE    X-COORD     Y-COORD          NODE    X-COORD     Y-COORD
       --------------------------          --------------------------
        1        .000        .000           12    2040.000        .000
        2     240.000        .000           13    2160.000        .000
        3     480.000        .000           14    2316.000        .000
        4     720.000        .000           15    2472.000        .000
        5     960.000        .000           16    2580.000        .000
        6    1152.000        .000           17    2688.000        .000
        7    1344.000        .000           18    2796.000        .000
        8    1536.000        .000           19    2904.000        .000
        9    1728.000        .000           20    3012.000        .000
       10    1824.000        .000           21    3120.000        .000
       11    1920.000        .000

CONCRETE MATERIAL PROPERTIES INPUT
==================================

             CONCRETE NUMBER    1
         MATERIAL MODEL NUMBER  1
      STRESS-STRAIN CURVE TYPE  2
       TENSION STIFFENING CODE  0

      28-DAY CYLINDER STRENGTH  3.975E+00
     ULTIMATE CREEP COEFFICIENT 0.000E+00
     ULTIMATE SHRINKAGE STRAIN  0.000E+00
                   UNIT WEIGHT  8.970E-05
       THERMAL EXPANSION COEFF  0.000E+00
             STRENGTH RATIO  RC 1.000E+00
                ULTIMATE STRAIN 4.100E-03
                     STRAIN  E20 4.000E-03
                STRESS RATIO R20 8.500E-01
      EFFECTIVE STRESS COEFF  C1 2.331E+00
      EFFECTIVE STRESS COEFF  C2 -4.658E-01
      EFFECTIVE STRESS RATIO  R1 3.500E-01
```

```
MILD STEEL PROPERTIES INPUT
===========================

 STEEL NO    EMOD1      EMOD2       FSY        ESU       ALPHA

       1  2.900E+04  0.000E+00  6.000E+01  3.000E-02  0.000E+00

PRESTRESSING STEEL PROPERTIES INPUT
===================================

 STEEL   CURVATURE    WOBBLE      YIELD       RELAX      THERMAL
   NO     FRICTION   FRICTION     STRESS      COEFF       COEFF

   1     2.500E-01   1.667E-05   0.000E+00   1.000E+01   0.000E+00

STRESS-STRAIN CURVE POINTS FOR PRESTRESSING STEEL NO.  1
--------------------------------------------------------

   POINT        STRESS      STRAIN      MODULUS

     1        1.966E+02   7.150E-03   2.750E+04
     2        2.200E+02   9.000E-03   1.265E+04
     3        2.400E+02   1.150E-02   8.000E+03
     4        2.450E+02   1.350E-02   2.500E+03
     5        2.700E+02   5.800E-02   5.618E+02

FRAME ELEMENT SECTION PROPERTIES
================================

SECTION NO   CONC NO      NCL        NSL       KSHRNK

    1           1         10          5          1

CONCRETE LAYER DATA FOR SECTION NO.  1
--------------------------------------

 LAYER NO      AREA       Y-COORD   SHRNK FACTOR

     1       2.040E+03   3.750E+01   1.000E+00
     2       2.040E+03   3.250E+01   1.000E+00
     3       4.500E+02   2.250E+01   1.000E+00
     4       4.500E+02   7.500E+00   1.000E+00
     5       4.500E+02  -7.500E+00   1.000E+00
     6       4.500E+02  -2.250E+01   1.000E+00
     7       4.500E+02  -3.750E+01   1.000E+00
     8       4.500E+02  -5.250E+01   1.000E+00
     9       8.040E+02  -6.200E+01   1.000E+00
    10       8.040E+02  -6.600E+01   1.000E+00

         TOTAL AREA  =  8.388E+03
   1ST MOMENT OF AREA  = -6.120E+02
    MOMENT OF INERTIA  =  1.400E+07

STEEL LAYER DATA FOR SECTION NO.  1
-----------------------------------

 LAYER NO  STL NO   CL NO     AREA       Y-COORD

     1       1        1     1.224E+01   3.500E+01
```

FIG. 3.4 OUTPUT LISTING OF EXAMPLE 1 (2 of 7)

```
  2       1     3       2.700E+00   1.500E+01
  3       1     5       2.700E+00  -1.500E+01
  4       1     7       2.700E+00  -4.500E+01
  5       1     9       4.824E+00  -6.400E+01

         TOTAL AREA  =  2.516E+01
   1ST MOMENT OF AREA  = -1.836E+00
    MOMENT OF INERTIA  =  4.144E+04


 FRAME ELEMENT DATA INPUT
 ========================

  FRAME  NODE   NODE CONCR   SECT INTEG CASTING
    NO     I      J  TYPE    TYPE  TYPE   DATE

     1     1      2    1       1     3     0.
     2     2      3    1       1     3     0.
     3     3      4    1       1     3     0.
     4     4      5    1       1     3     0.
     5     5      6    1       1     3     0.
     6     6      7    1       1     3     0.
     7     7      8    1       1     3     0.
     8     8      9    1       1     3     0.
     9     9     10    1       1     3     0.
    10    10     11    1       1     3     0.
    11    11     12    1       1     3     0.
    12    12     13    1       1     3     0.
    13    13     14    1       1     3     0.
    14    14     15    1       1     3     0.
    15    15     16    1       1     3     0.
    16    16     17    1       1     3     0.
    17    17     18    1       1     3     0.
    18    18     19    1       1     3     0.
    19    19     20    1       1     3     0.
    20    20     21    1       1     3     0.

 TENDON GEOMETRY AND MATERIAL SPECIFICATIONS FOR TENDON NO.    1
 ===============================================================

     NUMBER OF SPANS =     2
     MATERIAL TYPE   =     1
     TENDON AREA     =  2.632E+01

 GENERATED TENDON POINT GEOMETRY FOR TENDON    1
 ----------------------------------------------

   TP NO    X-COORD     Y-COORD      ANGLE      NODE    X-OFFSET    Y-OFFSET

      1  0.000E+00  -1.804E-15   0.000E+00      1   0.000E+00  -1.804E-15
      2  2.400E+02  -2.406E+01   2.843E-02      2   0.000E+00  -2.406E+01
      3  4.800E+02  -4.125E+01   2.855E-02      3   0.000E+00  -4.125E+01
      4  7.200E+02  -5.156E+01   2.862E-02      4   0.000E+00  -5.156E+01
      5  9.600E+02  -5.500E+01   3.619E-02      5   0.000E+00  -5.500E+01
      6  1.152E+03  -5.080E+01   4.366E-02      6   0.000E+00  -5.080E+01
      7  1.344E+03  -3.820E+01   4.341E-02      7   0.000E+00  -3.820E+01
      8  1.536E+03  -1.720E+01   4.300E-02      8   0.000E+00  -1.720E+01
      9  1.728E+03   1.220E+01   2.144E-02      9   0.000E+00   1.220E+01
     10  1.824E+03   2.480E+01   8.678E-02     10   0.000E+00   2.480E+01
     11  1.920E+03   2.900E+01   7.871E-02     11   0.000E+00   2.900E+01
     12  2.040E+03   2.480E+01   6.963E-02     12   0.000E+00   2.480E+01
     13  2.160E+03   1.220E+01   2.331E-02     13   0.000E+00   1.220E+01
     14  2.316E+03  -7.866E+00   2.244E-02     14   0.000E+00  -7.866E+00
     15  2.472E+03  -2.438E+01   1.907E-02     15   0.000E+00  -2.438E+01
     16  2.580E+03  -3.374E+01   1.565E-02     16   0.000E+00  -3.374E+01
```

```
     17  2.688E+03  -4.139E+01   1.569E-02     17   0.000E+00  -4.139E+01
     18  2.796E+03  -4.735E+01   1.571E-02     18   0.000E+00  -4.735E+01
     19  2.904E+03  -5.160E+01   1.573E-02     19   0.000E+00  -5.160E+01
     20  3.012E+03  -5.415E+01   1.575E-02     20   0.000E+00  -5.415E+01
     21  3.120E+03  -5.500E+01   0.000E+00     21   0.000E+00  -5.500E+01


 MESH INPUT COMPLETE
 ===================


 ==========================================================================
 = D A T E ,   E N V I R O N M E N T ,   &   T O L E R A N C E S         =
 ==========================================================================


                    CURRENT DAY NUMBER  =  2.800E+01

                   CURRENT TEMPERATURE  =  0.000E+00
                   ACCELERATION FACTOR  =  7.100E-01

              GRAVITY LOAD MULTIPLIERS
                           X DIRECTION  =  0.000E+00
                           Y DIRECTION  = -1.000E+00

                CONVERGENCE TOLERANCES
                   STRESS RATIO (TOLS)  =  1.000E-02
     DISP RATIO FOR FINAL STEP  (TOLF)  =  5.000E-02
     DISP RATIO FOR INTMD STEPS (TOLI)  =  5.000E-02
     DISP RATIO FOR STIFFNESSES (TOLC)  =  1.000E-04
          MAX UNBALANCED FORCE  (TOLL)  =  1.000E+01
          MAX UNBALANCED MOMENT (TOLM)  =  1.000E+03
    MAX DISPLACEMENT INCREMENT  (TOLD)  =  1.000E+01
        MAX ROTATION INCREMENT  (TOLR)  =  5.000E-01

    MAX ITERATIONS / LOAD OR DISP STEP
                            FINAL STEP  =  50
                     INTERMEDIATE STEPS  =  30


 ==========================================================================
 = B R I D G E   C O N S T R U C T I O N   C O M M A N D S               =
 ==========================================================================


 INITIAL CONCRETE PROPERTIES    -   CONC. NO. =  1    AGE =   28.00
 ---------------------------

           FPC        FPT      EMODI   SHRINKAGE
     4.000E+00  5.000E-01  3.605E+03  0.000E+00

        A(1,1)     A(2,1)     A(3,1)     A(4,1)
     0.000E+00  0.000E+00  0.000E+00  0.000E+00

                                     FRM ELEM     1 BUILT, CAST  0.000E+00
                                     FRM ELEM     2 BUILT, CAST  0.000E+00
                                     FRM ELEM     3 BUILT, CAST  0.000E+00
                                     FRM ELEM     4 BUILT, CAST  0.000E+00
                                     FRM ELEM     5 BUILT, CAST  0.000E+00
                                     FRM ELEM     6 BUILT, CAST  0.000E+00
                                     FRM ELEM     7 BUILT, CAST  0.000E+00
                                     FRM ELEM     8 BUILT, CAST  0.000E+00
                                     FRM ELEM     9 BUILT, CAST  0.000E+00
                                     FRM ELEM    10 BUILT, CAST  0.000E+00
                                     FRM ELEM    11 BUILT, CAST  0.000E+00
```

FIG. 3.4 OUTPUT LISTING OF EXAMPLE 1 (3 of 7)

```
                                       FRM ELEM   12 BUILT, CAST  0.000E+00
                                       FRM ELEM   13 BUILT, CAST  0.000E+00
                                       FRM ELEM   14 BUILT, CAST  0.000E+00
                                       FRM ELEM   15 BUILT, CAST  0.000E+00
                                       FRM ELEM   16 BUILT, CAST  0.000E+00
                                       FRM ELEM   17 BUILT, CAST  0.000E+00
                                       FRM ELEM   18 BUILT, CAST  0.000E+00
                                       FRM ELEM   19 BUILT, CAST  0.000E+00
                                       FRM ELEM   20 BUILT, CAST  0.000E+00


                                       NODE    1 RESTRAINT SET TO  0 1 0
                                       NODE   11 RESTRAINT SET TO  0 1 0
                                       NODE   21 RESTRAINT SET TO  1 0 1


INITIAL TENDON DATA AFTER INSTALLATION
--------------------------------------

  TDN      TENSIONING      SEG  FRM  MAT
   NO         TYPE          NO   NO CODE       FORCE       STRESS       STRAIN

    1   POST-TENSIONED      1    1    1     4.952E+03   1.882E+02   6.843E-03
    1   POST-TENSIONED      2    2    1     5.002E+03   1.900E+02   6.911E-03
    1   POST-TENSIONED      3    3    1     5.060E+03   1.922E+02   6.992E-03
    1   POST-TENSIONED      4    4    1     5.120E+03   1.945E+02   7.075E-03
    1   POST-TENSIONED      5    5    1     5.068E+03   1.926E+02   7.003E-03
    1   POST-TENSIONED      6    6    1     4.999E+03   1.899E+02   6.908E-03
    1   POST-TENSIONED      7    7    1     4.930E+03   1.873E+02   6.812E-03
    1   POST-TENSIONED      8    8    1     4.868E+03   1.849E+02   6.726E-03
    1   POST-TENSIONED      9    9    1     4.804E+03   1.825E+02   6.638E-03
    1   POST-TENSIONED     10   10    1     4.715E+03   1.791E+02   6.515E-03
    1   POST-TENSIONED     11   11    1     4.615E+03   1.753E+02   6.377E-03
    1   POST-TENSIONED     12   12    1     4.537E+03   1.724E+02   6.268E-03
    1   POST-TENSIONED     13   13    1     4.487E+03   1.705E+02   6.200E-03
    1   POST-TENSIONED     14   14    1     4.451E+03   1.691E+02   6.150E-03
    1   POST-TENSIONED     15   15    1     4.420E+03   1.679E+02   6.107E-03
    1   POST-TENSIONED     16   16    1     4.394E+03   1.669E+02   6.071E-03
    1   POST-TENSIONED     17   17    1     4.369E+03   1.660E+02   6.036E-03
    1   POST-TENSIONED     18   18    1     4.344E+03   1.650E+02   6.002E-03
    1   POST-TENSIONED     19   19    1     4.319E+03   1.641E+02   5.968E-03
    1   POST-TENSIONED     20   20    1     4.298E+03   1.633E+02   5.939E-03

==============================================================================
= I N C R E M E N T A L   S T R U C T U R E   S O L U T I O N               =
==============================================================================


SOLUTION CONTROL DATA
---------------------

          SOLUTION MODE  =  INSTANTANEOUS SOLUTION
      SOLUTION STRATEGY  =  LOAD CONTROL
  GEOMETRIC NONLINEARITY =  NOT CONSIDERED
     GEOMETRIC STIFFNESS =  NOT INCLUDED
    NUMBER OF LOAD STEPS =  1
   LOAD STEP MULTIPLIERS =  1.00E+00
 DAY AT START OF SOLUTION =  0.00E+00
 DAY AT END    OF SOLUTION =  2.80E+01


NODE RESTRAINTS AND EQUATION NUMBERS
------------------------------------
```

```
   NODE       -RESTRAINTS-          -EQUATIONS-
    NO.        X    Y   ZZ          X    Y   ZZ

     1         0    1    0          1    0    2
     2         0    0    0          3    4    5
     3         0    0    0          6    7    8
     4         0    0    0          9   10   11
     5         0    0    0         12   13   14
     6         0    0    0         15   16   17
     7         0    0    0         18   19   20
     8         0    0    0         21   22   23
     9         0    0    0         24   25   26
    10         0    0    0         27   28   29
    11         0    1    0         30    0   31
    12         0    0    0         32   33   34
    13         0    0    0         35   36   37
    14         0    0    0         38   39   40
    15         0    0    0         41   42   43
    16         0    0    0         44   45   46
    17         0    0    0         47   48   49
    18         0    0    0         50   51   52
    19         0    0    0         53   54   55
    20         0    0    0         56   57   58
    21         1    0    1          0   59    0

                                    ACTIVE FRAME ELEMENTS:
                                         1    2    3    4    5    6    7
                                         8    9   10   11   12   13   14
                                        15   16   17   18   19   20

                                    ACTIVE PRESTRESSING TENDONS:
                                         1

                                    STEP  1     DAY AT END    28.00


==============================================================================
= S U M M A R Y   O U T P U T   O F   T O T A L   R E S P O N S E           =
==============================================================================


        SOLUTION DATE  =  2.800E+01
     LOAD STEP NUMBER  =  1
     ITERATION NUMBER  =  2

   DISPLACEMENT RATIO  =  4.854E-02
   CONTROLLING DOF NO  =  37

MAX UNBALANCED FORCE   =  1.079E+00
MAX UNBALANCED MOMENT  =  7.012E+01

 CURRENT TOTAL NO. OF
 ITERATIONS PERFORMED  =   2


NODAL BOUNDARY CONDITIONS AND TOTAL DISPLACEMENTS
=================================================

    NODE      UX   UY   RZ        X-DISP      Y-DISP      Z-ROTN

      1        0    1    0      4.972E-01   0.000E+00   4.202E-04
      2        0    0    0      4.576E-01   1.019E-01   4.319E-04
      3        0    0    0      4.174E-01   2.059E-01   4.300E-04
      4        0    0    0      3.767E-01   3.020E-01   3.570E-04
```

FIG. 3.4 OUTPUT LISTING OF EXAMPLE 1 (4 of 7)

```
       5        0    0    0      3.355E-01   3.669E-01   1.627E-04
       6        0    0    0      3.027E-01   3.774E-01  -6.409E-05
       7        0    0    0      2.705E-01   3.415E-01  -3.109E-04
       8        0    0    0      2.389E-01   2.604E-01  -5.270E-04
       9        0    0    0      2.080E-01   1.438E-01  -6.733E-04
      10        0    0    0      1.926E-01   7.631E-02  -7.369E-04
      11        0    1    0      1.775E-01   0.000E+00  -8.723E-04
      12        0    0    0      1.589E-01  -1.154E-01  -1.016E-03
      13        0    0    0      1.408E-01  -2.382E-01  -1.017E-03
      14        0    0    0      1.176E-01  -3.914E-01  -9.401E-04
      15        0    0    0      9.452E-02  -5.293E-01  -8.208E-04
      16        0    0    0      7.860E-02  -6.124E-01  -7.158E-04
      17        0    0    0      6.274E-02  -6.833E-01  -5.949E-04
      18        0    0    0      4.695E-02  -7.404E-01  -4.603E-04
      19        0    0    0      3.123E-02  -7.823E-01  -3.145E-04
      20        0    0    0      1.558E-02  -8.080E-01  -1.600E-04
      21        1    0    1      0.000E+00  -8.166E-01   0.000E+00


CURRENT TOTAL JOINT LOADS                   UNBALANCED LOADS AND REACTIONS
=========================                   ==============================

NODE    X-LOAD      Y-LOAD      Z-MOMT      NODE   X-LOAD      Y-LOAD      Z-MOMT

   1  0.000E+00 RESTRAINED   0.000E+00         1  3.268E-02  6.009E+02  1.835E+00
   2  0.000E+00 -1.806E+02   0.000E+00         2  3.723E-03 -4.513E-02 -1.404E+00
   3  0.000E+00 -1.806E+02   0.000E+00         3  1.643E-02 -5.951E-02 -1.297E+01
   4  0.000E+00 -1.806E+02   0.000E+00         4  4.950E-02 -1.006E-01 -3.042E+01
   5  0.000E+00 -1.625E+02   0.000E+00         5  1.626E-02  1.054E-02 -3.794E+01
   6  0.000E+00 -1.445E+02   0.000E+00         6 -7.796E-03  1.551E-01 -1.930E+01
   7  0.000E+00 -1.445E+02   0.000E+00         7 -2.429E-02  1.238E-01  5.864E+00
   8  0.000E+00 -1.445E+02   0.000E+00         8 -2.197E-02  5.275E-02  2.244E+01
   9  0.000E+00 -1.083E+02   0.000E+00         9 -2.470E-02 -2.907E-01  7.190E+00
  10  0.000E+00 -7.223E+01   0.000E+00        10  2.227E-01 -1.079E+00 -5.706E+01
  11  0.000E+00 RESTRAINED   0.000E+00        11  1.291E-01  1.744E+03  1.440E+01
  12  0.000E+00 -9.029E+01   0.000E+00        12 -3.320E-01 -1.005E+00  7.012E+01
  13  0.000E+00 -1.038E+02   0.000E+00        13 -2.772E-02 -1.398E-01  1.146E+01
  14  0.000E+00 -1.174E+02   0.000E+00        14  7.994E-03 -9.668E-03  3.955E+00
  15  0.000E+00 -9.932E+01   0.000E+00        15  6.402E-03 -5.125E-03  2.409E+00
  16  0.000E+00 -8.126E+01   0.000E+00        16  6.592E-03 -3.075E-03  1.524E+00
  17  0.000E+00 -8.126E+01   0.000E+00        17  5.983E-03 -2.738E-03  1.220E+00
  18  0.000E+00 -8.126E+01   0.000E+00        18  4.978E-03 -2.825E-03  9.416E-01
  19  0.000E+00 -8.126E+01   0.000E+00        19  3.623E-03 -3.150E-03  6.531E-01
  20  0.000E+00 -8.126E+01   0.000E+00        20  1.951E-03 -3.534E-03  3.381E-01
  21 RESTRAINED -4.063E+01 RESTRAINED         21  6.943E-02 -1.867E-03  3.087E+05


FRAME ELEMENT FORCES AND MOMENTS
================================

FRAME  COMPONENT    MOMENT-I     MOMENT-J     SHEAR F.     AXIAL F.

  1        TOTAL    1.835E+00    1.226E+05    5.106E+02    3.268E-02
        CONCRETE    1.554E+00    3.880E+03    1.616E+01   -4.807E+03
           STEEL    2.808E-01    1.000E+02    4.155E-01   -1.205E+02
          TENDON    8.890E-12    1.186E+05    4.941E+02    4.928E+03

  2        TOTAL    1.226E+05    2.018E+05    3.301E+02    3.640E-02
        CONCRETE    2.442E+03   -3.918E+03   -2.650E+01   -4.867E+03
           STEEL    6.313E+01   -1.007E+02   -6.826E-01   -1.221E+02
          TENDON    1.200E+05    2.058E+05    3.573E+02    4.989E+03

  3        TOTAL    2.018E+05    2.377E+05    1.496E+02    5.283E-02
        CONCRETE   -6.596E+03   -2.242E+04   -6.593E+01   -4.931E+03
           STEEL   -1.698E+02   -5.793E+02   -1.706E+00   -1.238E+02
          TENDON    2.085E+05    2.607E+05    2.172E+02    5.055E+03
```

```
  4        TOTAL    2.376E+05    2.302E+05   -3.089E+01    1.023E-01
        CONCRETE   -2.567E+04   -5.005E+04   -1.016E+02   -4.994E+03
           STEEL   -6.643E+02   -1.300E+03   -2.647E+00   -1.256E+02
          TENDON    2.640E+05    2.816E+05    7.332E+01    5.119E+03

  5        TOTAL    2.302E+05    1.930E+05   -1.934E+02    1.186E-01
        CONCRETE   -4.726E+04   -6.272E+04   -8.048E+01   -4.942E+03
           STEEL   -1.226E+03   -1.630E+03   -2.105E+00   -1.244E+02
          TENDON    2.787E+05    2.574E+05   -1.108E+02    5.067E+03

  6        TOTAL    1.930E+05    1.281E+05   -3.380E+02    1.108E-01
        CONCRETE   -5.887E+04   -6.086E+04   -1.038E+01   -4.866E+03
           STEEL   -1.528E+03   -1.580E+03   -2.715E-01   -1.225E+02
          TENDON    2.534E+05    1.906E+05   -3.274E+02    4.989E+03

  7        TOTAL    1.281E+05    3.547E+04   -4.826E+02    8.650E-02
        CONCRETE   -5.757E+04   -4.759E+04    5.201E+01   -4.780E+03
           STEEL   -1.491E+03   -1.231E+03    1.356E+00   -1.202E+02
          TENDON    1.872E+05    8.429E+04   -5.360E+02    4.900E+03

  8        TOTAL    3.549E+04   -8.492E+04   -6.271E+02    6.453E-02
        CONCRETE   -4.608E+04   -2.556E+04    1.069E+02   -4.694E+03
           STEEL   -1.190E+03   -6.581E+02    2.769E+00   -1.177E+02
          TENDON    8.276E+04   -5.870E+04   -7.368E+02    4.811E+03

  9        TOTAL   -8.491E+04   -1.555E+05   -7.352E+02    3.984E-02
        CONCRETE   -2.613E+04   -3.643E+04   -1.073E+02   -4.646E+03
           STEEL   -6.724E+02   -9.388E+02   -2.774E+00   -1.164E+02
          TENDON   -5.811E+04   -1.181E+05   -6.251E+02    4.763E+03

 10        TOTAL   -1.555E+05   -2.330E+05   -8.063E+02    2.626E-01
        CONCRETE   -3.776E+04   -9.392E+04   -5.850E+02   -4.594E+03
           STEEL   -9.709E+02   -2.435E+03   -1.525E+01   -1.156E+02
          TENDON   -1.168E+05   -1.366E+05   -2.061E+02    4.710E+03

 11        TOTAL   -2.329E+05   -1.302E+05    8.566E+02    3.917E-01
        CONCRETE   -9.669E+04   -1.538E+04    6.775E+02   -4.499E+03
           STEEL   -2.502E+03   -3.908E+02    1.759E+01   -1.130E+02
          TENDON   -1.337E+05   -1.144E+05    1.614E+02    4.612E+03

 12        TOTAL   -1.301E+05   -3.801E+04    7.673E+02    5.962E-02
        CONCRETE   -1.774E+04    1.661E+04    2.862E+02   -4.402E+03
           STEEL   -4.534E+02    4.243E+02    7.314E+00   -1.100E+02
          TENDON   -1.119E+05   -5.504E+04    4.737E+02    4.512E+03

 13        TOTAL   -3.800E+04    6.552E+04    6.636E+02    3.190E-02
        CONCRETE    1.589E+04    2.976E+04    8.891E+01   -4.342E+03
           STEEL    4.051E+02    7.571E+02    2.257E+00   -1.085E+02
          TENDON   -5.429E+04    3.500E+04    5.724E+02    4.450E+03

 14        TOTAL    6.552E+04    1.507E+05    5.462E+02    3.990E-02
        CONCRETE    2.995E+04    4.176E+04    7.569E+01   -4.318E+03
           STEEL    7.616E+02    1.060E+03    1.915E+00   -1.080E+02
          TENDON    3.481E+04    1.079E+05    4.686E+02    4.426E+03

 15        TOTAL    1.507E+05    1.990E+05    4.469E+02    4.630E-02
        CONCRETE    4.230E+04    4.919E+04    6.385E+01   -4.296E+03
           STEEL    1.074E+03    1.248E+03    1.611E+00   -1.075E+02
          TENDON    1.074E+05    1.486E+05    3.814E+02    4.403E+03

 16        TOTAL    1.990E+05    2.385E+05    3.657E+02    5.289E-02
        CONCRETE    4.988E+04    5.567E+04    5.367E+01   -4.276E+03
           STEEL    1.264E+03    1.410E+03    1.351E+00   -1.071E+02
          TENDON    1.479E+05    1.814E+05    3.106E+02    4.383E+03
```

FIG. 3.4 OUTPUT LISTING OF EXAMPLE 1 (5 of 7)

```
 17      TOTAL   2.385E+05   2.692E+05   2.844E+02   5.887E-02
      CONCRETE   5.651E+04   6.114E+04   4.286E+01  -4.255E+03
         STEEL   1.431E+03   1.547E+03   1.077E+00  -1.066E+02
        TENDON   1.806E+05   2.065E+05   2.405E+02   4.362E+03

 18      TOTAL   2.692E+05   2.911E+05   2.031E+02   6.385E-02
      CONCRETE   6.214E+04   6.554E+04   3.145E+01  -4.234E+03
         STEEL   1.572E+03   1.657E+03   7.894E-01  -1.062E+02
        TENDON   2.055E+05   2.240E+05   1.709E+02   4.340E+03

 19      TOTAL   2.911E+05   3.043E+05   1.219E+02   6.748E-02
      CONCRETE   6.668E+04   6.878E+04   1.939E+01  -4.212E+03
         STEEL   1.686E+03   1.738E+03   4.861E-01  -1.057E+02
        TENDON   2.228E+05   2.338E+05   1.020E+02   4.318E+03

 20      TOTAL   3.043E+05   3.087E+05   4.063E+01   6.943E-02
      CONCRETE   6.980E+04   7.052E+04   6.613E+00  -4.193E+03
         STEEL   1.763E+03   1.781E+03   1.656E-01  -1.052E+02
        TENDON   2.327E+05   2.364E+05   3.385E+01   4.298E+03



=============================================================================
= S U M M A R Y   O U T P U T   O F   T O T A L   R E S P O N S E           =
=============================================================================


       SOLUTION DATE  =  2.800E+01
 DISPLACEMENT STEP NO  =  1
     ITERATION NUMBER  = 11

   DISPLACEMENT RATIO  =  3.805E-06
   CONTROLLING DOF NO  =  57
     MULTIPLIER ALPHA  =  9.424E-02

MAX UNBALANCED FORCE   =  9.892E+00
MAX UNBALANCED MOMENT  =  7.783E+02

 CURRENT TOTAL NO. OF
 ITERATIONS PERFORMED  =  70

NODAL BOUNDARY CONDITIONS AND TOTAL DISPLACEMENTS
=================================================

    NODE         UX   UY   RZ        X-DISP      Y-DISP      Z-ROTN

      1          0    1    0      -2.402E+00   0.000E+00   7.828E-03
      2          0    0    0      -2.442E+00   1.868E+00   7.694E-03
      3          0    0    0      -2.482E+00   3.669E+00   7.255E-03
      4          0    0    0      -2.524E+00   5.321E+00   6.452E-03
      5          0    0    0      -2.567E+00   6.732E+00   5.225E-03
      6          0    0    0      -2.601E+00   7.616E+00   3.940E-03
      7          0    0    0      -2.637E+00   8.229E+00   2.409E-03
      8          0    0    0      -2.159E+00   7.610E+00  -8.978E-03
      9          0    0    0      -1.740E+00   4.850E+00  -1.974E-02
     10          0    0    0      -1.553E+00   2.715E+00  -2.479E-02
     11          0    1    0      -1.196E+00   0.000E+00  -3.311E-02
     12          0    0    0      -1.014E+00  -4.341E+00  -3.847E-02
     13          0    0    0      -1.034E+00  -9.009E+00  -3.925E-02
     14          0    0    0      -1.058E+00  -1.517E+01  -3.969E-02
     15          0    0    0      -1.082E+00  -2.136E+01  -3.959E-02
     16          0    0    0      -1.098E+00  -2.562E+01  -3.924E-02
     17          0    0    0      -1.070E+00  -2.976E+01  -3.722E-02
     18          0    0    0      -9.735E-01  -3.355E+01  -3.300E-02
     19          0    0    0      -7.950E-01  -3.676E+01  -2.631E-02
```

```
     20          0    0    0      -5.119E-01  -3.908E+01  -1.650E-02
     21          1    0    1       0.000E+00  -4.000E+01   0.000E+00


CURRENT TOTAL JOINT LOADS                       UNBALANCED LOADS AND REACTIONS
=========================                       ==============================

NODE    X-LOAD      Y-LOAD      Z-MOMT          NODE    X-LOAD      Y-LOAD      Z-MOMT

   1  0.000E+00  RESTRAINED   0.000E+00            1  -2.344E-06   3.518E+02   9.938E-03
   2  0.000E+00  -1.806E+02   0.000E+00            2  -5.541E-05   3.869E-04   7.366E-02
   3  0.000E+00  -1.806E+02   0.000E+00            3  -4.778E-05   6.367E-04   1.953E-01
   4  0.000E+00  -1.806E+02   0.000E+00            4  -5.404E-05   9.641E-04   3.863E-01
   5  0.000E+00  -1.625E+02   0.000E+00            5   1.820E-05   9.473E-04   5.467E-01
   6  0.000E+00  -1.445E+02   0.000E+00            6  -9.368E-05   5.288E-04   6.329E-01
   7  0.000E+00  -1.445E+02   0.000E+00            7  -1.994E-02  -6.450E-02  -6.258E+00
   8  0.000E+00  -1.445E+02   0.000E+00            8   8.792E-02   9.151E-02   3.211E-01
   9  0.000E+00  -1.083E+02   0.000E+00            9  -6.329E-02  -1.256E-02   1.287E+00
  10  0.000E+00  -7.223E+01   0.000E+00           10   3.822E+00  -9.892E+00  -1.975E+02
  11  0.000E+00  RESTRAINED   0.000E+00           11  -3.925E+00   2.984E+03  -7.783E+02
  12  0.000E+00  -9.029E+01   0.000E+00           12   9.865E-02   3.659E-01  -1.718E+01
  13  0.000E+00  -1.038E+02   0.000E+00           13  -1.304E-04   5.276E-03  -1.189E+00
  14  0.000E+00  -1.174E+02   0.000E+00           14   3.023E-03   4.281E-03  -7.058E-01
  15  0.000E+00  -3.844E+02   0.000E+00           15  -2.445E-03   2.294E-03  -5.785E-02
  16  0.000E+00  -8.126E+01   0.000E+00           16  -7.813E-04  -9.131E-03  -4.686E-01
  17  0.000E+00  -3.664E+02   0.000E+00           17   3.210E-04  -4.169E-03  -1.184E+00
  18  0.000E+00  -8.126E+01   0.000E+00           18   1.918E-04   1.068E-03  -1.344E+00
  19  0.000E+00  -3.664E+02   0.000E+00           19   2.850E-04   7.627E-03  -8.820E-01
  20  0.000E+00  -8.126E+01   0.000E+00           20  -4.608E-04  -8.169E-05  -4.550E-01
  21 RESTRAINED  -1.832E+02  RESTRAINED           21  -5.566E-04   4.428E-03   6.583E+05


FRAME ELEMENT FORCES AND MOMENTS
================================

FRAME  COMPONENT    MOMENT-I    MOMENT-J    SHEAR F.    AXIAL F.

   1       TOTAL   9.938E-03   6.276E+04   2.615E+02  -2.344E-06
        CONCRETE  -2.416E+00  -5.427E+04  -2.261E+02  -4.799E+03
           STEEL   2.426E+00  -1.365E+03  -5.698E+00  -1.210E+02
          TENDON   8.877E-12   1.184E+05   4.933E+02   4.920E+03

   2       TOTAL   6.276E+04   8.218E+04   8.091E+01  -5.775E-05
        CONCRETE  -5.482E+04  -1.187E+05  -2.662E+02  -4.820E+03
           STEEL  -1.380E+03  -3.037E+03  -6.904E+00  -1.235E+02
          TENDON   1.190E+05   2.039E+05   3.540E+02   4.944E+03

   3       TOTAL   8.218E+04   5.826E+04  -9.966E+01  -1.055E-04
        CONCRETE  -1.190E+05  -1.920E+05  -3.042E+02  -4.824E+03
           STEEL  -3.048E+03  -5.004E+03  -8.151E+00  -1.264E+02
          TENDON   2.042E+05   2.552E+05   2.127E+02   4.950E+03

   4       TOTAL   5.826E+04  -8.999E+03  -2.802E+02  -1.596E-04
        CONCRETE  -1.919E+05  -2.739E+05  -3.416E+02  -4.818E+03
           STEEL  -5.012E+03  -7.293E+03  -9.507E+00  -1.306E+02
          TENDON   2.552E+05   2.722E+05   7.088E+01   4.948E+03

   5       TOTAL  -8.998E+03  -9.401E+04  -4.428E+02  -1.414E-04
        CONCRETE  -2.685E+05  -3.313E+05  -3.273E+02  -4.715E+03
           STEEL  -7.148E+03  -8.954E+03  -9.406E+00  -1.329E+02
          TENDON   2.666E+05   2.463E+05  -1.061E+02   4.848E+03

   6       TOTAL  -9.401E+04  -2.068E+05  -5.872E+02  -2.350E-04
        CONCRETE  -3.271E+05  -3.783E+05  -2.668E+02  -4.627E+03
           STEEL  -8.838E+03  -1.035E+04  -7.871E+00  -1.355E+02
          TENDON   2.420E+05   1.819E+05  -3.126E+02   4.763E+03
```

FIG. 3.4 OUTPUT LISTING OF EXAMPLE 1 (6 of 7)

```
 7     TOTAL  -2.068E+05  -3.472E+05  -7.316E+02  -2.017E-02
    CONCRETE  -3.742E+05  -4.004E+05  -1.365E+02  -6.199E+03
       STEEL  -3.790E+04  -3.931E+04  -7.323E+00   8.244E+02
      TENDON   2.053E+05   9.244E+04  -5.878E+02   5.374E+03

 8     TOTAL  -3.472E+05  -5.155E+05  -8.762E+02   6.775E-02
    CONCRETE  -4.049E+05  -4.055E+05  -2.856E+00  -6.462E+03
       STEEL  -4.006E+04  -4.062E+04  -2.922E+00   7.783E+02
      TENDON   9.777E+04  -6.935E+04  -8.704E+02   5.684E+03

 9     TOTAL  -5.155E+05  -6.100E+05  -9.845E+02   4.454E-03
    CONCRETE  -4.034E+05  -4.227E+05  -2.015E+02  -6.637E+03
       STEEL  -4.035E+04  -4.143E+04  -1.126E+01   7.574E+02
      TENDON  -7.174E+04  -1.458E+05  -7.717E+02   5.880E+03

10     TOTAL  -6.102E+05  -7.107E+05  -1.047E+03   3.826E+00
    CONCRETE  -4.114E+05  -4.805E+05  -7.198E+02  -7.162E+03
       STEEL  -4.031E+04  -4.487E+04  -4.752E+01   7.764E+02
      TENDON  -1.585E+05  -1.853E+05  -2.795E+02   6.389E+03

11     TOTAL  -7.114E+05  -4.887E+05   1.856E+03  -9.921E-02
    CONCRETE  -4.963E+05  -3.087E+05   1.563E+03  -6.560E+03
       STEEL  -4.593E+04  -3.529E+04   8.870E+01   7.239E+02
      TENDON  -1.692E+05  -1.447E+05   2.043E+02   5.836E+03

12     TOTAL  -4.887E+05  -2.769E+05   1.765E+03  -5.596E-04
    CONCRETE  -3.652E+05  -2.154E+05   1.248E+03  -4.457E+03
       STEEL  -9.873E+03  -5.591E+03   3.568E+01  -1.258E+02
      TENDON  -1.137E+05  -5.592E+04   4.812E+02   4.583E+03

13     TOTAL  -2.769E+05  -1.767E+04   1.662E+03  -6.899E-04
    CONCRETE  -2.169E+05  -5.145E+04   1.061E+03  -4.342E+03
       STEEL  -5.601E+03  -1.260E+03   2.782E+01  -1.132E+02
      TENDON  -5.435E+04   3.504E+04   5.730E+02   4.455E+03

14     TOTAL  -1.767E+04   2.232E+05   1.544E+03   2.333E-03
    CONCRETE  -5.124E+04   1.125E+05   1.049E+03  -4.318E+03
       STEEL  -1.251E+03   2.817E+03   2.607E+01  -1.089E+02
      TENDON   3.482E+04   1.079E+05   4.687E+02   4.427E+03

15     TOTAL   2.232E+05   3.485E+05   1.160E+03  -1.123E-04
    CONCRETE   1.120E+05   1.936E+05   7.557E+02  -4.334E+03
       STEEL   2.797E+03   4.843E+03   1.894E+01  -1.117E+02
      TENDON   1.084E+05   1.500E+05   3.851E+02   4.446E+03

16     TOTAL   3.485E+05   4.650E+05   1.079E+03  -8.936E-04
    CONCRETE   1.548E+05   2.265E+05   6.641E+02  -5.343E+03
       STEEL   1.987E+04   2.519E+04   4.921E+01   1.908E+02
      TENDON   1.738E+05   2.133E+05   3.652E+02   5.152E+03

17     TOTAL   4.650E+05   5.419E+05   7.121E+02  -5.726E-04
    CONCRETE   1.967E+05   2.384E+05   3.857E+02  -6.122E+03
       STEEL   3.238E+04   3.371E+04   1.233E+01   4.243E+02
      TENDON   2.358E+05   2.697E+05   3.141E+02   5.697E+03

18     TOTAL   5.419E+05   6.100E+05   6.309E+02  -3.808E-04
    CONCRETE   2.167E+05   2.575E+05   3.775E+02  -6.650E+03
       STEEL   3.327E+04   3.442E+04   1.065E+01   4.858E+02
      TENDON   2.919E+05   3.181E+05   2.427E+02   6.164E+03

19     TOTAL   6.100E+05   6.386E+05   2.645E+02  -9.582E-05
    CONCRETE   2.440E+05   2.558E+05   1.094E+02  -6.958E+03
       STEEL   3.354E+04   3.384E+04   2.796E+00   5.141E+02
      TENDON   3.325E+05   3.489E+05   1.522E+02   6.444E+03
```

```
20     TOTAL   6.386E+05   6.583E+05   1.832E+02  -5.566E-04
    CONCRETE   2.519E+05   2.655E+05   1.259E+02  -7.091E+03
       STEEL   3.303E+04   3.366E+04   5.897E+00   5.603E+02
      TENDON   3.536E+05   3.592E+05   5.143E+01   6.531E+03


CONCRETE AND STEEL LAYER DATA
=============================


CONCRETE LAYER DATA FOR FRAME ELEMENT NO. 11 ,  GAUSS POINT NO.  2
-------------------------------------------------------------------

LAYER  MAT      TOTAL       TOTAL      MECHANICAL  NON-MECHAN.    CREEP
   NO  CODE     STRESS      STRAIN      STRAIN      STRAIN       STRAIN

    1   6    0.000E+00   4.304E-03   4.304E-03   0.000E+00   0.000E+00
    2   6    0.000E+00   4.006E-03   4.006E-03   0.000E+00   0.000E+00
    3   6    0.000E+00   3.410E-03   3.410E-03   0.000E+00   0.000E+00
    4   6    0.000E+00   2.515E-03   2.515E-03   0.000E+00   0.000E+00
    5   6    0.000E+00   1.621E-03   1.621E-03   0.000E+00   0.000E+00
    6   6    0.000E+00   7.269E-04   7.269E-04   0.000E+00   0.000E+00
    7   8   -4.338E-01  -1.673E-04  -1.673E-04   0.000E+00   0.000E+00
    8   2   -2.912E+00  -1.062E-03  -1.062E-03   0.000E+00   0.000E+00
    9   2   -3.716E+00  -1.628E-03  -1.628E-03   0.000E+00   0.000E+00
   10   2   -3.899E+00  -1.866E-03  -1.866E-03   0.000E+00   0.000E+00


STEEL LAYER DATA FOR FRAME ELEMENT NO. 11 ,  GAUSS POINT NO.  2
----------------------------------------------------------------

LAYER  MAT      TOTAL       TOTAL      MECHANICAL  NON-MECHAN.
   NO  CODE     STRESS      STRAIN      STRAIN      STRAIN

    1   2    6.000E+01   4.155E-03   4.155E-03   0.000E+00
    2   2    6.000E+01   2.963E-03   2.963E-03   0.000E+00
    3   1    3.405E+01   1.174E-03   1.174E-03   0.000E+00
    4   1   -1.782E+01  -6.144E-04  -6.144E-04   0.000E+00
    5   1   -5.067E+01  -1.747E-03  -1.747E-03   0.000E+00


CONCRETE LAYER DATA FOR FRAME ELEMENT NO. 20 ,  GAUSS POINT NO.  3
-------------------------------------------------------------------

LAYER  MAT      TOTAL       TOTAL      MECHANICAL  NON-MECHAN.    CREEP
   NO  CODE     STRESS      STRAIN      STRAIN      STRAIN       STRAIN

    1   2   -2.858E+00  -1.034E-03  -1.034E-03   0.000E+00   0.000E+00
    2   8   -6.823E-01  -2.186E-04  -2.186E-04   0.000E+00   0.000E+00
    3   6    0.000E+00   1.411E-03   1.411E-03   0.000E+00   0.000E+00
    4   6    0.000E+00   3.856E-03   3.856E-03   0.000E+00   0.000E+00
    5   6    0.000E+00   6.301E-03   6.301E-03   0.000E+00   0.000E+00
    6   6    0.000E+00   8.747E-03   8.747E-03   0.000E+00   0.000E+00
    7   6    0.000E+00   1.119E-02   1.119E-02   0.000E+00   0.000E+00
    8   6    0.000E+00   1.364E-02   1.364E-02   0.000E+00   0.000E+00
    9   6    0.000E+00   1.519E-02   1.519E-02   0.000E+00   0.000E+00
   10   6    0.000E+00   1.584E-02   1.584E-02   0.000E+00   0.000E+00


STEEL LAYER DATA FOR FRAME ELEMENT NO. 20 ,  GAUSS POINT NO.  3
----------------------------------------------------------------

LAYER  MAT      TOTAL       TOTAL      MECHANICAL  NON-MECHAN.
   NO  CODE     STRESS      STRAIN      STRAIN      STRAIN

    1   1   -1.816E+01  -6.261E-04  -6.261E-04   0.000E+00
```

FIG. 3.4 OUTPUT LISTING OF EXAMPLE 1 (7 of 7)

```
   2    2      6.000E+01   2.634E-03   2.634E-03   0.000E+00
   3    2      6.000E+01   7.524E-03   7.524E-03   0.000E+00
   4    2      6.000E+01   1.241E-02   1.241E-02   0.000E+00
   5    2      6.000E+01   1.551E-02   1.551E-02   0.000E+00


TENDON FORCES, STRAINS AND MATERIAL STATES
==========================================

TENDON  SEG  FRM  MAT      TOTAL   PERCENT    TOTAL      TOTAL   MECHANICAL
    NO   NO   NO CODE      FORCE      LOSS   STRESS     STRAIN       STRAIN

     1    1    1    1  4.945E+03      .15  1.879E+02  6.833E-03  6.833E-03
     1    2    2    1  4.956E+03      .91  1.883E+02  6.848E-03  6.848E-03
     1    3    3    1  4.955E+03     2.08  1.883E+02  6.846E-03  6.846E-03
     1    4    4    1  4.949E+03     3.34  1.880E+02  6.838E-03  6.838E-03
     1    5    5    1  4.849E+03     4.32  1.842E+02  6.701E-03  6.701E-03
     1    6    6    1  4.773E+03     4.52  1.814E+02  6.596E-03  6.596E-03
     1    7    7    2  5.406E+03    -9.67  2.054E+02  7.846E-03  7.846E-03
     1    8    8    2  5.750E+03   -18.14  2.185E+02  8.880E-03  8.880E-03
     1    9    9    3  5.930E+03   -23.46  2.253E+02  9.665E-03  9.665E-03
     1   10   10    4  6.395E+03   -35.65  2.430E+02  1.269E-02  1.269E-02
     1   11   11    3  5.840E+03   -26.54  2.219E+02  9.233E-03  9.233E-03
     1   12   12    1  4.608E+03    -1.59  1.751E+02  6.368E-03  6.368E-03
     1   13   13    1  4.492E+03     -.10  1.707E+02  6.206E-03  6.206E-03
     1   14   14    1  4.452E+03     -.02  1.691E+02  6.151E-03  6.151E-03
     1   15   15    1  4.463E+03     -.97  1.696E+02  6.166E-03  6.166E-03
     1   16   16    1  5.165E+03   -17.56  1.962E+02  7.137E-03  7.137E-03
     1   17   17    2  5.706E+03   -30.62  2.168E+02  8.747E-03  8.747E-03
     1   18   18    3  6.169E+03   -42.02  2.344E+02  1.080E-02  1.080E-02
     1   19   19    4  6.446E+03   -49.24  2.449E+02  1.346E-02  1.346E-02
     1   20   20    5  6.531E+03   -51.94  2.481E+02  1.908E-02  1.908E-02

                                   MEMORY USAGE:
                                    MAX USAGE    4152 INTEGERS
```

4. Programmer's Guide

This programmer's guide is provided for those who maintain the program, modify the program to suit their own purposes, or use this program as a basis for further development.

4.1 Program Organization

The program SPCFRAME consists of a short main program and approximately 130 Fortran subprograms which perform the input and output functions, database management functions, and the numerical computations required for the time-dependent nonlinear solution. These subprograms are grouped into 10 source modules, each of which may be a program overlay if required by the computer used. The program is written around a named incore array database manager [11] which dynamically allocates and manages all numerical array storage, and a free-field input interpreter which provides the user interface.

The program was developed on a personal computer, which is an IBM clone based on Intel 80386 processor, under the MS-DOS 3.30 using Microsoft Fortran 4.1 compiler, and tested on the VAX mini- and micro-computers at the University of California, Berkeley using the UNIX operating system and f77 Fortran compiler and linker. The program can also be run on IBM XT and AT class personal computers. All hardware dependent functions are located in several easily modified subroutines, so that the simple transfer of the program to other machines is possible.

The functions of 10 source modules will be described briefly:

(1) SPCFRAME
Initializes the database, and the problem solution, controls the overall program flow, and calls the other modules when required by the input commands.

(2) UTILITY

Performs common functions required by all the other modules. Includes the input interpreter, in-core array manager, and element array record filing routines. All hardware dependent functions are in subroutines located in this module.

(3) CONCRETE

Inputs and generates the tables of time-dependent concrete material model parameters for creep, shrinkage, and aging of the concrete.

(4) INPUT

Inputs and generates the node coordinates, material properties, cross-section properties, and element types and locations. Initializes and files the frame, concrete and steel layer, and tendon element characteristic matrices.

(5) CHANGE

Interprets all structure configuration change commands. Initializes nodal displacements, tendon segment foces and traveler element characteristic matrices based on input commands. Sets environmental conditions and convergence tolerances for the solution.

(6) LOAD

Inputs, generates and computes various load vectors used in the analysis. These include nodal loads, loads due to creep, shrinkage, temperature change, prestress, imposed displacements and unbalanced loads at various solution time and load or displacement steps.

(7) STIFF

Computes structure tangent stiffness matrix at various steps of solution.

(8) SOLVE

Performs the time-dependent nonlinear solution for the currently defined structure and loading. This module performs the bulk of the numerical operations along with the STATE module.

(9) STATE

Carries out the state determination phase of the analysis after each iteration. Computes current internal forces and internal resisting loads based on the displacement increments of the current iteration.

(10) OUTPUT

Prints and summarizes the convergence information, nodal displacements, unbalanced loads and reactions, frame element and traveler forces and moments, concrete and mild steel layer data, and prestressing tendon stresses and strains.

The overall flow of the program from one module to another is determined by the input commands. A typical analysis will start at the root (SPCFRAME) module, which will first call the CONCRETE and INPUT modules in order to input the structural geometry and material properties. For each construction step, the CHANGE and SOLVE modules will be called once. Then for each time step in the analysis the SOLVE module will be called once. Whenever SOLVE module is called several calls to LOAD, STIFF, STATE and OUTPUT modules will be made. Execution always stops in the root module after saving the in-core array database to a file named ????.COR where ???? is the filename of the input data. The analysis can be restarted by restoring the database from this file and the separate element and layer data files named ????.FRM, ????.CNC, ????.STL, and ????.TDN.

4.2 System Implementation Notes

(1) This version is for Microsoft FORTRAN 4.1 under MS-DOS 3.30. Only minor modifications are required for VAX-UNIX and IBM-CMS versions. Modifications to be made on subroutines START and FREE are marked on comment lines in these routines.

(2) The program can operate in terminal or batch mode. For batch mode, define files NTM and NTR at the JCL level. File NTR should then contain the following lines:

```
<FILENAME> where <FILENAME> is the name of the input file
START R=??
STOP
```

And the JCL would look like:

```
<FILE DEFINITION FOR FILE 'NTM'>
<FILE DEFINITION FOR FILE 'NTR'>
<COMMAND TO EXECUTE SPCFRAME>
```

For interactive mode, the program prompts for this input.

(3) Core memory is set in blank common. To change the core size, change the following two lines in SPCFRAME to reflect the desired size:

```
COMMON MTOT,NP,IA(15000)
MTOT = 15000
```

(4) Disk file sizes are set in START through subroutine calls. Some operating systems do not require setting this size. For very large or very small problems, these sizes can be changed. See the comments in START.

(5) Machine precision is set in START in the array IP. IP(1) is bytes per integer, IP(2) is bytes per real, IP(3) is bytes per character. These values are used for allocating the incore database.

(6) Frame, concrete layer, steel layer and tendon segment data

are kept in direct access disk files. All access is through routines GETFRM, PUTFRM, GETCL, PUTCL, GETSL, PUTSL, GETTDN, PUTTDN. These routines may need to be changed for some operating systems.

(7) All input is translated to uppercase before it is interpreted by the program. Subroutine UPPER does this, assuming ASCII character coding. This subroutine will need to be changed if EBCDIC code is used.

(8) There are a few undocumented commands for debugging purposes:

P	Prints any array in the database.
L	Lists a directory of arrays in the database.
CLEAR	Reinitializes the database.
SUBMIT	Reads commands from file when in interactive mode.
RETURN	Go back to interactive mode.

4.3 Detailed Description of Arrays and Variables

Major arrays and variables in in-core database, out-of-core files for frame elements, concrete layers, mild steel layers and tendon segments, and in labeled COMMON will be described in detail. The description includes the names of the subroutines where these arrays and variables are defined, computed, updated and deleted. Temporary in-core arrays for generating concrete parameter tables and arrays for the parametric generation of prestressing tendon points are excluded.

4.3.1 Directory of In-core Arrays

1. %CPT(4,NCPT) Concrete parameter type control data
   NCPT = No. of different concrete parameter types
   1 = KM : Model No. (LAB=1,ACI=2,CEB=3)
   2 = NA : No. of loading ages
   3 = NR : No. of retardation times
   4 = KL : No. of loading terms (1 for LAB & ACI,2 for CEB)
   defined in CONCRETE
   computed in CNCTYP

2. %PRM(NA,NCPC,NCPT) Creep data for each concrete parameter type
   NA = No. of loading ages
   NCPC = 5 + 2*NR
   NCPC 1 = Loading age τ
   2 = EMOD at τ for $f'_c(28)=1.$
   3 = Shrinkage strain at τ for $\epsilon^s_u = 1.$
   4 = $f'_c(\tau)$ for $f'_c(28)=1.$
   5 = $f'_t(\tau)$ for $f'_c(28)=1.$
   6 to 5+NR = Creep coefficients $a_i(\tau)$ for $C_u=1.$
   and $f'_c(28)=1.$
   6+NR to 5+2NR = Second set of $a_i(\tau)$ for CEB
   (KL=2,recoverable creep only)
   defined in CONCRETE
   computed in CRPCFT

3. %GAM(NCPT) Smallest retardation times
   defined in CONCRETE
   computed in CNCTYP

4. $CNC(13,NCNC) Concrete property data for each concrete type
   NCNC = No. of different concrete types
   1 = fpc, Cylinder strength $f'_c(28)$
   2 = ultcrp, Ultimate creep factor C_u

3 = ultshr, Ultimate shrinkage strain ϵ^s_u
4 = weight, Unit weight (for computing dead load only)
5 = alfa, Thermal expansion coefficient α
6 = gam0, Smallest retardation time Γ_0
7 = rc in $f''_c=rc*f'_c$
8 = epsu, Ultimate strain ϵ_u
9 = eps20, Strain at the end of descending straight line
10 = r20, Ratio in $f20=r20*f''_c$
11 = c1 in effective stress calculation for creep
12 = c2 in effective stress calculation for creep
13 = Ratio $r1=f/f''_c$ up to which $\sigma_e=\sigma$ in creep calc.
defined in MESHIN
computed in INPCNC

5. $CNI(6,NCNC) Concrete creep data and stress-strain curve type
1 = Parameter type No.
2 = KM, Concrete model No. (LAB=1,ACI=2,CEB=3)
3 = NA, No. of loading ages
4 = NR, No. of retardation times
5 = KL, No. of loading terms in a_i
6 = kcurve, Stress-strain curve type (linear=1,nonl=2)
7 = kts, tension stiffening code (0=no,1=included)
defined in MESHIN
computed in INPCNC

6. $MSL(5,NMSL) Mild steel properties for each steel type
NMSL = No. of different steel types
1 = Es1, 1st modulus in bilinear stress-strain curve
2 = Es2, 2nd modulus in bilinear stress-strain curve
3 = fsy, Yield stress σ_{sy}
4 = epsu, Ultimate strain ϵ_{su}
5 = alfa, Thermal expansion coefficient α
defined in MESHIN
computed in INPMSL

7. $PSL(5,NPSL) Prestressing steel properties for each P.S. type

NPSL = No. of different prestressing steel types
1 = crvf, Curvature friction coefficient μ
2 = wblf, Wobble friction coefficient κ
3 = fpy, 0.1% offset yield stress
4 = rlxp, Relaxation coefficient
5 = alfa, Thermal expansion coefficient α
defined in MESHIN
computed in INPPSL

8. $FPS(npt,npsl) Stress values at discrete points on mutilinear stress-strain curve of each prestressing steel type
npt = No. of points on the stress-strain curve
defined in MESHIN
computed in INPPSL

9. $EPS(npt,npsl) Strain values at discrete points on multilinear stress-strain curve of each prestressing steel type
defined in MESHIN
computed in INPPSL

10. $MOD(npt,npsl) Modulus from point (i-1) to point i on multilinear stress-strain curve of each prestressing steel type
defined in MESHIN
computed in INPPSL

11. $SCT(4,NSCT) Section properties of each different section type
NSCT = No. of different section types
1 = nc, Concrete no.
2 = ncl, No. of concrete layers
3 = nsl, No. of steel layers
4 = kshr, Shrinkage distribution code
(0=none, 1=uniform, 2=linear, 3=nonlinear)
defined in MESHIN

computed in INPSCT

12. $ACL(mcl,nsct) Areas of concrete layers for each section type
    mcl = No. of maximum concrete layers in any section
    defined in MESHIN
    computed in INPSCT

13. $YCL(mcl,nsct) Y coordinates of concrete layers for each section type
    defined in MESHIN
    computed in INPSCT

14. $YCT(mcl1,nsct) Y coordinates of concrete layer boundaries
    mcl1 = mcl + 1
    defined in MESHIN
    computed in INPSCT

15. $SHK(mcl,nsct) Shrinkage factors of concrete layers for each section
    defined in MESHIN
    computed in INPSCT

16. $ASL(msl,nsct) Areas of steel layers for each section type
    msl = No. of maximum steel layers in any section
    defined in MESHIN
    computed in INPSCT

17. $YSL(msl,nsct) Y coordinates of steel layers for each section type
    defined in MESHIN
    computed in INPSCT

18. $SLS(msl,nsct) Material no's of steel layers for each section type
    defined in MESHIN
    computed in INPSCT

19\. $SLC(msl,nsct) Corresponding concrete layer no.'s of steel layers for temperature calculations for each section type

defined in MESHIN
computed in INPSCT

20\. $TMP(mcl,ntdp) Temperatures of concrete layers for each different temperature distribution pattern

ntdp = No. of different temperature distribution patterns
defined in MESHIN
computed in INPTMP

21\. $FRM(3,NFML) Frame existence and node data for each frame

NFML = No. of frame elements
1 = Frame element existence flag
    = 0 : Element has never existed
    = 2 : To install next assembly
    = 1 : Currently installed
    = -1 : Currently removed
    = -2 : To remove next assembly
2 = Node I
3 = Node J
defined in MESHIN
computed in INPFRM,FRMINS
updated in FRMILD,FRMRMV

22\. &TYP(3,NFML) Frame property data for each frame

1 = Section type No.
2 = Casting date
3 = Creep integration type No.
defined in MESHIN
computed in INPFRM (IFTP(3,NFML) in INPFRM)
deleted in MESHIN

23\. &SEC(NFML) Section No.'s of each frame

defined in MESHIN
computed in INPFRM

24. $FTD(8,nfml) Frame and tendon interchange data for state determination
    1 = clm, Current length
    2 = cosm
    3 = sinm
    4 = oang, Original angle in radians
    5 = esap, 0.75ES/EA term for internal dof recovery
    6 = s1p, Contribution of tendon or traveler to s1
    7 = s2p, Contribution of tendon or traveler to s2
    8 = s3p, Contribution of tendon or traveler to s3
    defined in MESHIN
    computed in INPFRM(1-4),STIFF(5),TDNINC,TRVINC(6-8)
    updated in FRMINC(1-3)

25. $CSL(2,nfml) Layer record No. pointers for each frame
    1 = Zeroth concrete layer No.
    2 = Zeroth steel layer No.
    defined in MESHIN
    computed in INPFRM

26. &TDP(nfml) Temperature distribution pattern No. for each frame
    defined in LODINP
    computed in LODFRM
    deleted in LODINP

27. &MLT(nfml) Temperature multiplier for each frame
    defined in LODINP
    computed in LODFRM
    deleted in LODINP

28. $TDN(2,NTDN) Tendon data for each tendon
    NTDN = No. of prestressing steel tendons
    1 = Tendon existence flag
        = 0 : Tendon does not exist
        = 2 : To install next assembly

= 1 : Currently installed
2 = No. of segments in the tendon
defined in MESHIN
computed in TDNINP

29. $TDR(NTDN) Zeroth segment No. of each tendon
defined in MESHIN
computed in TDNINP

30. $TRV(2,NTRV) Traveler existence data for each traveler
NTRV = No. of travelers
1 = Traveler existence flag (0 or 1)
2 = No. of nodes
defined in MESHIN
computed in TRVCHG

31. $TRR(NTRV) Zeroth element No. of each traveler
defined in MESHIN
computed in INPTRV

32. $TRP(4,NTRV) Traveler property data for each traveler
1 = Area
2 = Inertia
3 = EMOD
4 = Total weight
defined in MESHIN
computed in INPTRV

33. $TRN(NNIM,NTRV) Node No.'s of each traveler
NNIM = Maximum No. of nodes for any traveler
defined in INPTRV
computed in TRVCHG

34. $TRL(NNIM,NTRV) Nodal dead loads of each traveler
defined in INPTRV
computed in TRVSTF

35. $TRF(NLIM,NTRV) Frame element No.'s of each traveler
NLIM = Maximum No. of elements for any traveler
defined in INPTRV
computed in TRVCHG

36. $TRA(86,NLTT) Traveler properties for all traveler elements
NLTT = Total No. of traveler elements
FSTF(6,6), DTOF(4,6), FTOR(6,3), FORC(4),FPRV(4)
of each traveler element
defined in INPTRV
computed in TRVSTF

37. $AMB(2) Environment data
1 = Current temperature
2 = Convergence acceleration factor
defined in SET
computed in SETCHG

38. $DAY(3) Dates
in SET
1 = Current date
in SOLVE
1 = Date at the end of current solution step
2 = Date at the end of previous solution step
defined in SET
computed in SETCHG,SOLVE

39. $GRV(2) Gravity load multipliers in X and Y directions
defined in SET
computed in SETCHG

40. $TOL(8) Convergence tolerances
1 = tols, Stress ratio tolerance
2 = tolf, Disp. ratio tol. for final load step
3 = toli, Disp. ratio tol. for intermediate load steps
4 = tolc, Disp. ratio tol. for changing stiffness

5 = toll, Maximum unbalanced force
6 = tolm, Maximum unbalanced moment
7 = told, Maximum allowed translation
8 = tolr, Maximum allowed rotation
defined in SET
computed in SETCHG

41. $XIT(3) Maximum No. of iterations allowed
1 = for final load step
2 = for intermediate load steps
3 = Current total No. of iterations performed
defined in SET
computed in SETCHG

42. -TDT(NSTP) Day No.'s at the end of each solution time step
NSTP = No. of solution time steps
defined in SPCFRAME
computed in SOLTIM
deleted in SPCFRAME

43. $XYZ(2,NNOD) X and Y coordinates of each node
NNOD = No. of nodes in the structure
defined in MESHIN
computed in INPXYZ

44. $SEQ(NNOD) Node No. sequence in the global equilibrium equations
defined in MESHIN
computed in INPSEQ

45. $NXF(NNOD) Node existence flag of each node (0,1 or -ND)
defined in MESHIN
computed in FRMINS,FRMRMV,TRVCHG,CHGNXF

46. $MBC(3,NNOD) Boundary condition code for each DOF (0 or 1)
= 0 if free to displace

= 1 if restrained
defined in MESHIN
computed in RSTINP

47. $XLD(3,NNOD) External load increment $\{\Delta R^j\}$
defined in MESHIN
computed in LODNOD,LODTOT

48. $XDS(3,NNOD) External displacement increment
defined in MESHIN
computed in LODNOD,RSTINP

49. $NTD(3,NNOD) Node total displacement $\{r\}$
defined in MESHIN
computed in INCDSP

50. $NTF(3,NNOD) Node total joint load $\{R^j\}$
defined in MESHIN
computed in LODTOT

51. $UBI(3,NNOD) Unbalanced loads $\{R^u\}$
defined in MESHIN
computed in LODLDS,FRMINC,UBLOAD

52. $RAC(3,NNOD) Internal resisting loads $\{R^i\}$
defined in MESHIN
computed in FRMINC,TDNINC,TRVINC

53. -EQN(3,NNOD) Equation No.'s of 3 dof at each node
defined in SOLVE
computed in EQNMBR
deleted in SOLVE

54. -ADR(NEQN) Diagonal element addresses of each equation No.
in 1-D structure stiffness matrix array
NEQN = No. of active structure dof

defined in SOLVE
computed in STFADR
deleted in SOLVE

55. -DLI(3,NNOD) Nodal dead load $\{\Delta R^d\}$
defined in SOLVE
computed in FRMILD,TRVILD
deleted in SOLVE

56. -ILD(3,NNOD) Nodal load due to non-mechanical strains $\{\Delta R^{nm}\}$
defined in SOLVE
computed in FRMILD,TDNILD,TRVILD
deleted in SOLVE

57. -GSF(NSIZ) 1-D structure stiffness matrix
NSIZ = Size of 1-D structure stiffness matrix
defined in SOLVE
computed in STIFF,FRMSTF,TDNSTF,STFSUM
deleted in SOLVE

58. -GDS(NEQN) Global load increment or displacement increment
defined in SOLVE
computed in LODSRT,COLSOL
deleted in SOLVE

59. -GDU(NEQN) Global load increment or displacement increment due to unbalanced load in displacement control solution scheme
defined in SOLVE
computed in LODSRT,COLSOL
deleted in SOLVE

60. -LST(NMEL) List of active frames, tendons or travelers
NMEL = MAX0(NTDN,NFML)
defined in SOLVE
computed in FRMILD,TDNILD,TRVILD

deleted in SOLVE

4.3.2 Directory of Frame File (NT8) Arrays

1. L	Frame No.
2. ITYP(3)	Frame type identitification No.'s 1 = ic, Concrete No. 2 = ix, Section No. 3 = im, Creep integration type computed in INPFRM
3. LEQN(6)	Equation No.'s of nodes i and j computed in STIFF,FRMILD
4. s123(9)	Frame element forces in element coordinates 1 - 3 = Total values of s1,s2,s3 4 - 6 = Concrete contributions to s1,s2,s3 7 - 9 = Steel contributions to s1,s2,s3 computed in FRMINC
5. DLDV(6)	Dead load (Fx_i,Fy_i,Fx_j,Fy_j) computed in FRMILD
6. DATE(3)	Dates and initial elastic modulus 1 = Casting date computed in FRMINS 2 = t_{j-1}, Date at the end of previous time step computed in FRMSTP 3 = EMOD, Initial elastic modulus of concrete computed in FRMINS,FRMILD
7. CCOF(5,2)	$a_i(t_j)$, Current creep coefficients for NR=1,5 and KL=1,2

computed in FRMILD

8. CCOP(5,2)	$a_i(t_{j-1})$, Previous creep coefficients for NR=1,5 and KL=1,2 computed in FRMSTP
9. ALFA(5,2)	$\beta_i(t_j)$ for NR=1,5 KL=1,2 computed in FRMILD
10. RLXK(2)	$R(t_j)$ for each KL computed in FRMILD
11. PHIT(5)	Current values of $(1 - \phi_i)$ for NR=1,5 computed in FRMILD
12. fdpc	Current value of f''_c computed in FRMINS,FRMILD
13. fdpcp	Previous value of f''_c computed in FRMILD
14. fpt	Current value of f'_t computed in FRMINS,FRMILD

4.3.3 Directory of Concrete Layer File (NT3) Arrays

1. l	Frame No.
2. kcl	Concrete layer No.
3. tempcr	Reference temperature computed in FRMINS updated in TEMPER

4. tempcc	Current temperature computed in FRMINS updated in TEMPER
5. epsscp	Previous shrinkage strain computed in FRMILD
6. FHSV(5,3)	$g_i(t_{j-1})$, Values of the hidden state variable at previous time step at each Gauss points for NR=1,5 computed in FRMSTP
7. DERP(3)	Previous values of $R\cdot\Delta\sigma_j$ at each Gauss point computed in CNCSTD
8. mcodec(3)	Concrete material state No's at each Gauss point computed in CNCSTD
9. emodc(3)	Concrete elastic modulus values at each Gauss point computed in CNCSTD
10. sigmac(3)	Total concrete stresses σ at each Gauss point computed in CNCSTD
11. sigmap(3)	Total concrete stresses σ_p at previous time step computed in FRMSTP
12. tepsc(3)	Total concrete strains ϵ^t at each Gauss point computed in CNCSTD
13. tepsic(3)	Total non-mechanical concrete strains ϵ^{nm} computed in FRMILD
14. tepsmc(3)	Total mechanical concrete strains ϵ^m computed in CNCSTD

15. tcreep(3)	Total creep strains ϵ^c computed in FRMILD,CNCSTD
16. dcrep1(3)	Creep strain increments $\Delta\epsilon^c$ computed in FRMILD
17. dcrep2(3)	Creep strain increments for CEB, recoverable part only computed in FRMILD,CNCSTD
18. epsrc(3)	Residual mechanical strains of concrete ϵ^r in case of load reversal computed in CNCSTD

4.3.4 Directory of Steel Layer File (NT4) Arrays

1. l	Frame No.
2. ksl	Steel layer No.
3. mcnc	Corresponding concrete layer No. computed in INPSCT,FRMINS
4. tempsr	Reference temperature computed in FRMINS updated in TEMPER
5. tempsc	Current temperature computed in FRMINS updated in TEMPER
6. mcodes(3)	Steel material state No,'s at each Gauss point computed in FRMINS updated in STLSTD

7. emods(3)	Steel elastic modulus values computed in FRMINS updated in STLSTD
8. sigmas(3)	Steel stresses σ_s computed in STLSTD
9. tepss(3)	Total steel strains ϵ^t computed in STLSTD
10. tepsms(3)	Total mechanical steel strains ϵ^m computed in STLSTD
11. tepsis(3)	Total non-mechanical steel strains ϵ^{nm} computed in STLILD
12. epsrs(3)	Residual mechanical strains of steel ϵ^r in case of load reversal computed in STLSTD

4.3.5 Directory of Tendon File (NT7) Arrays

1. MTDN	Tendon No.
2. MSEG	Segment No.
3. MFRM	Frame No. computed in TDNFIL
4. NODI	Node i computed in TDNFIL
5. NODJ	Node j computed in TDNFIL

6. MATL	Material No. computed in TDNINP
7. AREA	Area computed in TDNINP
8. ANGL	Cumulative angle change for friction calculation computed in TDNFIL
9. geom(6)	Geometric properties 1 = olps, Original length 2 = clps, Current length 3 = ecci, Eccentricity at node i in local coordinates 4 = eccj, Eccentricity at node j in local coordinates 5 = cosp, Cos α in local coordinates 6 = sinp, Sin α in local coordinates computed in TDNFIL updated in TDNINC
10. FINI	Initial force computed in TDNLOS
11. FORC	Current force computed in TDNLOS updated in TDNILD,TDNINC
12. rlxp	Current total relaxation in prestress force computed in TDNILD
13. mcodep	Material state code No. computed in TDNFIL updated in TDNINC
14. emodp	Elastic modulus

	computed in TDNLOS updated in TDNINC
15. tepsp	Total strain ϵ_p computed in TDNLOS updated in TDNINC
16. tepsmp	Total mechanical strain ϵ^t computed in TDNLOS updated in TDNINC
17. tepsip	Total non-mechanical strain ϵ^{nm} computed in TDNILD
18. epspp	Previous value of mechanical strain on the main stress-strain curve in case of load reversal computed in TDNINC
19. TREF	Reference temperature computed in TDNLOS updated in TEMPER
20. tcur	Current temperature computed in TDNLOS updated in TEMPER
21. LEQN(6)	Equation No.'s of node i and j computed in TDNSTF,TDNILD
22. DATE(2)	Dates 1 = Stressing date 2 = t_{j-1}, Date at the end of previous time step computed in TDNLOS updated in TDNSTP
23. al(3)	Displacement transformation from local frame

	deformation dof to local tendon dof computed in TDNFIL updated in TDNINC
24. a123(3)	Displacement transformation from global frame dof to local tendon dof computed in TDNFIL updated in TDNINC

4.3.6 Directory of Labeled Common Variables and Arrays

1. **common /iolist/ ntm,ntr,nin,not,nsp,nfl,nt7,nt8,nt3,nt4**

ntm = terminal output
ntr = terminal input
nin = input file
not = output file
nsp = camber file
nfl = incore data file
nt7 = tendon data file
nt8 = frame data file
nt3 = concrete layer data file
nt4 = steel layer data file

2. **common /funit / cfunit**

cfunit = conversion factor from lb-in units to other units

Unit problem arises from the fact that concrete parameter data are generated in lb-in units for ACI and CEB models assuming $f'_c(28) = 1$. Since concrete modulus is computed by E = c*sqrt(f'_c), modulus factors in CRPCFT are multiplied by square root of the conversion factor. For example, since 1 psi = 0.144 ksf, modulus factors are multiplied by cfunit = sqrt(0.144). Creep factors represent creep strains produced per unit stress, thus they are divided by cfunit. Actual values of modulus and creep strain corresponding to f'_c input in INPCNC are obtained by multiplying or dividing by sqrt(f'_c) in FRMCNC. The values in INPCNC and FRMCNC are in user selected units defined by UNIT command.

3. common /solvei/ kantim,kanldc,kgn,kgs,it,nit,lds,nlds, ns,nstp,maxkd,maxkr,maxgpt,maxcl.macfrm,ndof

kantim = solution mode flag [0]
= 0 for instantaneous solution
= 1 for time step solution
kanldc = solution strategy flag [0]
= 0 for load control
= 1 for displacement control
kgn = geometric nonlinearity code [0]
= 0 if geometric nonlinearity is not considered
= 1 if geometric nonlinearity is considered
kgs = geometric stiffness code [0]
= 0 if geometric stiffness is not included
= 1 if geometric stiffness is included
it = iteration number
nit = maximum no. of iterations allowed
lds = load or displacement step number
nlds = number of load or displacement steps [1 for kantim=1]
ns = time step number
nstp = number of time steps [1]
maxkd = dof no. where max. translation occurs on 1st iteration
maxkr = dof no. where max. rotation occurs on 1st iteration
maxgpt = gauss point no. where max. stress occurs on 1st iteration
maxcl = conc. layer no. where max. stress occurs on 1st

iteration
maxfrm = frame no. where max. stress occurs on 1st iteration
ndof = dof no. of controlled displacement component

4. common /solvef/ fload(20),fdisp(20),dratio,dmax,rmax,sratio, smax,tdelta,delta,alpha,ufmax,ummax

fload = load step multipliers
fdisp = displacement step multipliers
ufmax = maximum unbalanced force
ummax = maximum unbalanced moment
dmax = total displacement increment at dof no. = maxkd up to previous iteration for current load or disp. step
rmax = total rotation increment at dof no. = maxkr up to previous iteration for current load or disp. step
dratio = displacement ratio to be compared with convergence tolerances, defined as the ratio of the displacement increment at dof no. = maxdof for the current iteration to the total displacement increment at dof no. = maxdof up to previous iteration (dmax or rmax). maxdof is selected from maxkd and maxkr which gives greater value of dratio. ndof cannot be selected as maxdof. if dratio < tolc new stiffness matrix is formed. if dratio < toli or tolf, and ufmax < toll, ummax < tolm iteration is terminated. toli is used for intermediate load or displacement steps and tolf is used for final load or displacement step.
smax = maximum stress value on 1st iteration which occurs at gauss point no. = maxgpt, concrete layer no. = maxcl, frame no. = maxfrm
sratio = stress ratio to be compared with convergence tolerance tols for time step solution. stress ratio is defined as the ratio of stress increment for the current iteration to the total stress increment up to previous iteration for the current time step. stress

increments are checked for smax value which occurs at maxgpt,maxcl,maxfrm. if sratio < tols, and ufmax < toll, ummax < tolm iteration is terminated for the current time step.

tdelta = magnitude of controlled displacement

delta = increment of controlled displacement for the current displacement step, obtained by multiplying tdelta by fdisp(lds)

alpha = joint load multiplier for the current iteration in the displacement step solution

4.4 Limitations on the Problem Size and the Computer Time

The size of the core memory required is set in the main program by assigning the value of MTOT as explained in section 4.2. Due to the efficient in-core and out-of-core database management of the program there are virtually no limits on the size of the problems that can be solved. Most of the storage-consuming data for frame elements, concrete and mild steel layers, prestressing tendon segments are stored in peripheral storage devices such as magnetic disks or tapes for main-frame computers and hard disks for micro-computers. They are brought into the computer's main memory only when they are processed. Even on a personal computer structures having hundreds of joints and elemnts can be analyzed.

Execution time required for an analysis varies considerably depending on the computer system used for a given analysis. The results of the analyses for the three examples listed in Fig. 3.3 (page 69) are summarized in Table 4.1 in order to give a general idea of the execution times required. The examples were analyzed on a personal computer, an IBM clone having Intel 80386-based mother-board with 16-MHz clock speed and Intel 80387 math coprocessor under Micrsoft DOS 3.30 and Micrsoft Fortran 4.1 compiler and linker. Instead of using the hard disk, the virtual disk (or so called RAM drive) was utilized for out-of-core files. This reduces the execution time substantially due to the fact that extensive file handling operations are required in the program. This practice is strongly recommended for personal computer users.

Table 4.1 Comparison of Execution Times

	EXAMPLE 1		EXAMPLE 2	EXAMPLE 3		
	Ultimate Load Analysis	Creep Analysis	Post-buckling Analysis	Cnstruction Analysis	Ultimate Load Analysis	Creep Analysis
Execution time (minutes)	5	8	2	86	65	63
Total No. of iterations	70	70	71	167	69	48
No. of time steps	1	28	1	30	1	20
No. of load/disp. steps	12	1	14	1	11	1
Core array size	4152	4170	2852	12302	12298	12316
No. of joints	21	21	11	43	43	43
No. of DOF	59	59	30	123	123	123
No. of elements	20	20	10	42	42	42
No. of concrete layers	10	10	10	16	16	16
No. of mild steel layers	5	5	0	8	8	8
No. of tendon segments	20	20	20	442	442	442

5. References

1. Scordelis, A.C., "Analytical Models for Nonlinear Material, Geometric, and Time-Dependent Effects," Proceedings, International symposium on Nonlinearity and Continuity in Prestressed Concrete, Waterloo, Ontario, Canada, July 1983.

2. Scordelis, A.C., "Computer Models for Nonlinear Analysis of Reinforced and Prestressed Concrete Structures," PCI Journal Vol. 29, No. 6, November-December 1984.

3. Scordelis, A.C., "Recent Developments at Berkeley in Nonlinear Analysis of Prestressed Concrete Structures," Proceedings of the FIP Symposium, Jerusalem, Israel, September 1988

4. Kang, Y.J., "Nonlinear Geometric, Material, and Time-Dependent Analysis of Reinforced and Prestressed Concrete Frames," University of California, Berkeley, UC-SESM 77-1, January 1977.

5. Kang, Y.J. and Scordelis, A.C., "Nonlinear Analysis of Prestressed Concrete Frames," Journal of the Structural Division, ASCE, Vol. 106, No. ST2, February 1980.

6. Van Zyl, S.F., "Analysis of Curved Segmentally Erected Prestressed Concrete Box Girder Bridges," University of California, Berkeley, UC-SESM 78-2, January 1978.

7. Van Zyl, S.F. and Scordelis, A.C., "Analysis of Curved Prestressed Segmental Bridges," Journal of the Structural Division, ASCE, Vol. 105, No. ST11, November 1979.

8. Van Greunen, J., "Nonlinear Geometric, Material, and Time Dependent Analysis of Reinforced and Prestressed Concrete Slabs and Panels," University of California, Berkeley, UC-SESM 79-3, October 1979.

9. Mari, A.R., "Nonlinear Geometric, Material, and Time-Dependent Analysis of Three-Dimensional Reinforced and Prestressed Concrete Frames," University of California, Berkeley, UC-SESM 84-10, June 1984.

10. Ketchum, M.A., "Redistribution of Stresses in Segmentally Erected Prestressed Concrete Bridges," University of California, Berkeley, UCB/SESM-86/07, May 1986.

11. Wilson, E.L. and Hoit, M.I., "A Computer Adaptive Language for the Development of Structural Analysis Programs," Computers and Structures, Vol. 19, No. 3, 1984.

12. Choudhury, D., "Analysis of Curved Nonprismatic Reinforced and Prestressed Concrete Box Girder Bridges," University of California, Berkeley, UCB/SEMM-86/13, December 1986.

13. Powell, G.H. and Simons, J.W., "Improved Iteration Strategy for Nonlinear Structures," International Journal for Numerical Methods in Engineering, Vol. 17, 1981.