TECHNICAL REPORT STANDARD TITLE PAGE

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.

4. Title and Subtitle	5. Report Date
NONLINEAR MATERIAL AND TIME DEPENDENT ANALYSIS OF SEGMENTALLY ERECTED REINFORCED AND PRESTRESSED CONCRETE COMPOSITE 3D FRAME STRUCTURES	May 1990
	6. Performing Organization Code
7. Author(s)	8. Performing Organization Report No.
Fouad A. Kasti	UCB/SEMM-90/03
9. Performing Organization Name and Address	10. Work Unit No.
Department of Civil Engineering University of California Berkeley, California 94720	11. Contract or Grant No.
12. Sponsoring Agency Name and Address	13. Type of Report and Period Covered
The Byron L. and Elvira E. Nishkian Professorship, University of California at Berkeley	Final Report
	14. Sponsoring Agency Code

15. Supplementary Notes

16. Abstract

Segmental erection across the depth of three dimensional frame element cross sections is investigated. An original and rational procedure based on the finite element approach is described. Using this procedure, a method of analysis and computer program is developed for the nonlinear material and time dependent analysis of segmentally erected reinforced and prestressed concrete composite three dimensional frame structures. Segmental erection can be accomplished element by element along the length of the members or through their depths.

The computer program SPCF3D developed in the present study is a general purpose program that traces the nonlinear (arbitrary) material behaviour of segmentally erected prestressed concrete composite three dimensional frame structures. It extends the capabilities of previous programs, SFRAME by Ketchum [27] for segmental linear analysis of plane frames and PCF3D by Mari [25] for nonsegmental, nonlinear analysis of three dimensional frames. The finite element used to model the structural element is an incremental beam element based on improved Bernoulli-Euler kinematics. The filamented cross-section is assumed to be constant along the length of the structural element and may allow composite, segmental erection across its depth. Time dependent effects such as creep, shrinkage, temperature and relaxation are considered automatically. Precast or cast in place structural segments, pretensioned or post-tensioned prestressing tendons, traveling formworks in addition to force and displacement boundary conditions may be modeled within any statically feasible construction sequence. Numerical examples demonstrate these features and capabilities.

17. Key Words	18. Distribution Statement
Structural element, Frame superelement, Subsectioning, Prestressed concrete, Finite element, Segmental erection, Composite construction, Nonlinear material, Time dependent analysis.	Unlimited

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price
Unclassified	Unclassified		242

Form DOT F 1700.7 (8-69)

Department of Civil Engineering
Division of Structural Engineering
and Structural Mechanics

UCB/SEMM Report No. 90-03

NONLINEAR MATERIAL AND TIME DEPENDENT ANALYSIS OF SEGMENTALLY ERECTED REINFORCED AND PRESTRESSED CONCRETE COMPOSITE 3D FRAME STRUCTURES

by

Fouad A. Kasti

Faculty Supervisor : A.C. Scordelis

Prepared under the Sponsorship of :

The Byron L. and Elvira E. Nishkian
Professorship of Structural Engineering

College of Engineering
Office of Research Services
University of California
Berkeley, California
May 1990

NONLINEAR MATERIAL AND TIME DEPENDENT ANALYSIS OF SEGMENTALLY ERECTED REINFORCED AND PRESTRESSED CONCRETE COMPOSITE 3D FRAME STRUCTURES

Fouad Kasti

Doctor of Philosophy

Civil Engineering

ABSTRACT

Segmental erection across the depth of three dimensional frame element cross-sections is investigated. An original and rational procedure based on the finite element approach is described. Using this procedure, a method of analysis and computer program is developed for the nonlinear material and time dependent analysis of segemtally erected reinforced and prestressed concrete composite three dimensional frame structures. Segmental erection can be accomplished element by element along the length of the members or through their depths.

The computer program **SPCF3D** developed in the present study is a general purpose program that traces the nonlinear (arbitrary) material behaviour of segmentally erected prestressed concrete composite three dimensional frame structures. It extends the capabilities of previous programs, **SFRAME** by Ketchum [27] for segmental linear analysis of plane frames and **PCF3D** by Mari [25] for nonsegmental, nonlinear analysis of three dimensional frames. The finite element used to model the structural element is an incremental beam element based on improved Bernoulli-Euler kinematics. The filamented cross-section is assumed to be constant along the length of the structural element and may allow composite, segmental erection across its depth. Time dependent effects such as creep, shrinkage, temperature and relaxation are considered automatically. Precast or cast in place structural segments, prestensioned or post-tensioned prestressing tendons, traveling formworks in addition to force and displacement boundary conditions may be modeled within any statically feasible construction sequence.

Numerical examples obtained with the computer program **SPCF3D** developed for this study are presented to demonstrate the validity of the theory on which the incremental beam element is based as well as the accuracy of the implementation and the various features and capabilities of the computer program.

Alex C. Scordelis
May 31, 1990

A. C. Scordelis

Chairman of Committee

NONLINEAR MATERIAL AND TIME DEPENDENT ANALYSIS OF SEGMENTALLY ERECTED REINFORCED AND PRESTRESSED CONCRETE COMPOSITE 3D FRAME STRUCTURES

Table of Contents

CHAPTER 1 : Introduction

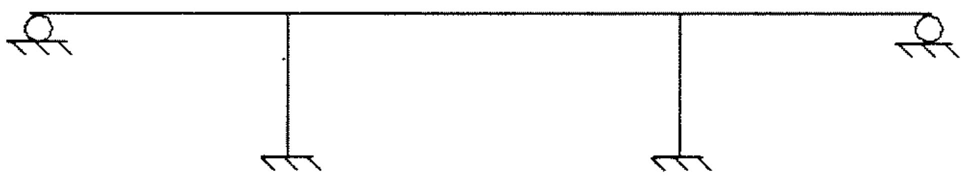
Introduction

Segmentally erected prestressed concrete bridges have been widely used for medium and long span bridges in today's modern highway system. Segmental erection provides a highly attractive solution for economic, aesthetic and environmental limitations, particularly when continuously supported formwork is impractical or uneconomical. A detailed description of their analysis, design and construction has been presented by Ketchum [27] for planar frames on which the current introduction is largely based.

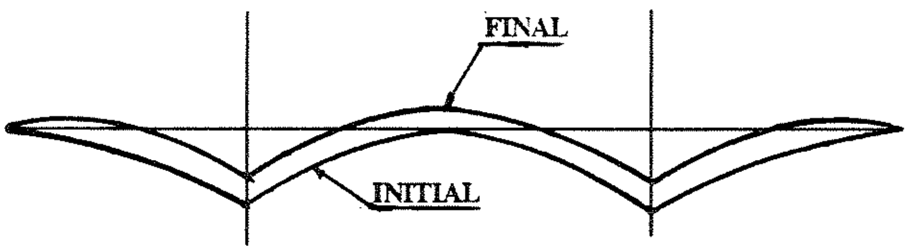
An analytical solution of the true response of segmentally erected frame structures, particularly bridges, has always been a cumbersome task complicated by the factors common to any other concrete structural system namely, structure indeterminancy, concrete and steel composite action, material and geometrical nonlinearities, construction sequence and time dependent redistribution of internal stresses which takes place over the life of the structure due to creep and shrinkage of concrete as well as relaxation of prestressing steel **Fig. 1.0.1**. As a consequence, current and previous designs of segmental bridges have been based on simplified structural theories and semi-empirical approaches to compute the response of internal actions in the structure. In that respect, the assumption of linear elastic homogeneous uncraked concrete structural system has been widely used, even though such an assumption may have been misleading and more extensive analysis might have been required. Recently, a load factor method, based on ultimate strength has been adopted but the method offers little guidance for complex segmentally erected structures.

More accurate analysis is required for the aforementioned classes of structures particularly in the event where the assumptions on which the design approaches are based is not applicable to the structure under study or in the event where the structure configuration is not within the reach of the conventional structural analysis capabilities. Simple spans made continuous is a good illustration of the latter case **Fig. 1.1.1** and the objective of the present study is to present a new method that could

A) INITIAL STRUCTURAL SYSTEM (CANTILEVER)

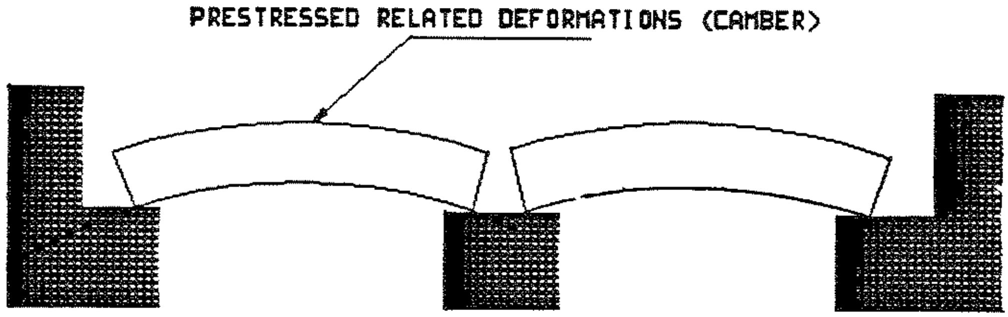


B) FINAL STRUCTURAL SYSTEM (CONTINUOUS)

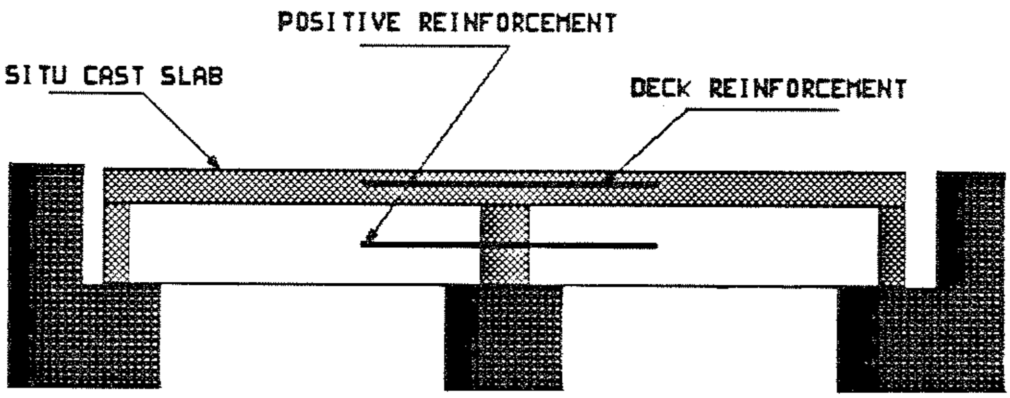


C) VARIATION OF INTERNAL MOMENTS

Fig. 1.0.1 MOMENT REDISTRIBUTION Ketchum [27]



A) PRECAST PRESTRESSED GIRDERS PLACED ON SUBSTRUCTURE



B) DECK SLAB AND DIAPHRAGMS

Fig. 1.1.1 CONSTRUCTION SEQUENCE FOR A TWO SPAN BRIDGE WITH PRECAST, PRESTRESSED GIRDERS MADE CONTINUOUS FOR LIVE LOADS. Oesterle [36] be used to account for such operations.

1.1 Segmental Bridge Construction

Segmetal construction is an expression that is used to describe a wide range of structures, the structural configuration of which during erection operation is dependent upon time. The most common segmental methods in practice may belong to either of the following two categories : the first category includes those methods of construction in which erection operation is carried out along the longitudinal axis of the structure. The analysis of these structural systems has been investigated to a great extent by Ketchum [27]. The second category consists of the construction methods in which erection operation is carried out across the depth of the cross section. The term subsectioning will be used in the analysis of the latter class of construction methods since the cross section of any element is conceived as a collection of sub-cross sections, each possibly built at a different time and consequently having distinct material properties.

Cantilever construction, span by span and incremental launching may be classified under the first category, Ketchum [27]. Simple spans made continuous is an ideal case that illustrates the second category of construction **Fig. 1.1.1**. These simply supported precast girder bridges made continuous with a cast in place deck have become widespread given that continuity over the supports will result in higher restraint hence lower span moments and deflections and thus more economical sections. Continuity will eliminate the need for construction joints between adjacent spans leading to improved riding surfaces and prevention of drainage through the deck hence lower construction and maintenance cost. Continuity will result in higher redundancy meaning higher overload capacity. Pretensioning at a precast plant is more cost effective than post tensioning on the site. Also the rigorous quality control at a precast plant results in higher mechanical properties at lower cost and the precast girders may be used to support, without any additional scaffoldings, the forms needed for the cast in place deck. Freyermuth, PCA [5]

The previous discussion emphasizes the fact that continuity plays an important role in such a contruction method. In that respect, several continuity connections have been used in segmental con-

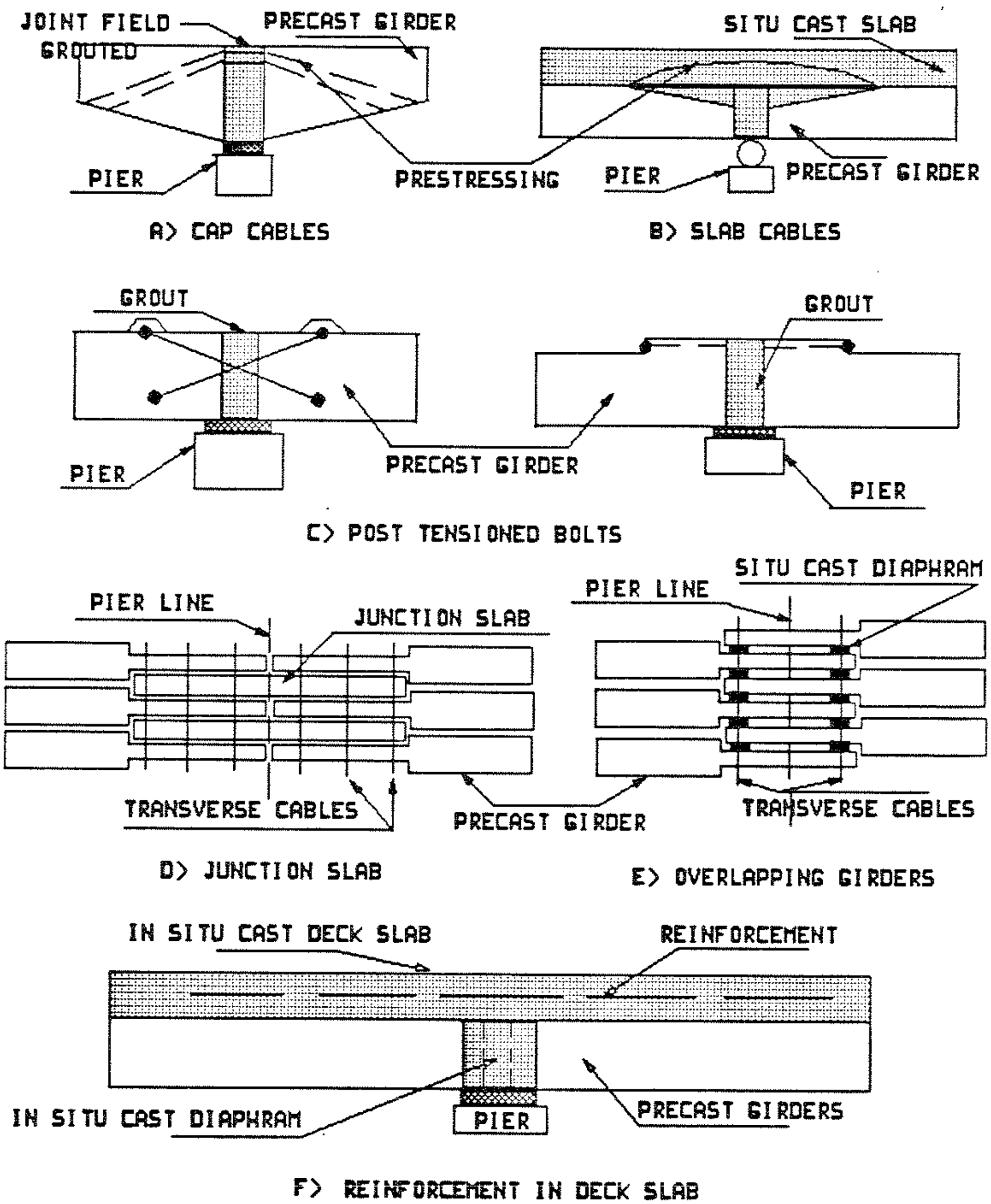


Fig. 1.1.2 CONTINUITY CONNECTIONS Freyermuth [5]

struction namely cap cables in which case the cost of stressing and anchorage is low however considerable friction loss is associated with their sharp curvature **Fig. 1.1.2a**, post-tensioned slab cables that are slightly curved and embedded in the situ deck **Fig. 1.1.2b**, post-tensioned bolts as shown in **Fig. 1.1.2c**, transverse prestressing **Fig. 1.1.2d** and conventional steel reinforcement embedded in the insitu deck which contrary to the previous continuity connections does not involve considerable field labor resulting in a simpler and more efficient construction operation **Fig. 1.1.2e**. Freyermuth, PCA [5]

1.2 Previous Studies

A limited number of research investigations have been directed toward the study of those classes of structures that includes segmental erection across the depth of the structural element in the construction operations such as Suttikan [12], Ghali and Elbadry [33] and Millanes and Calzon [35].

Suttikan [12] at the University of Texas at Austin developed a computer program **PBEAM** for the analysis of planar simply supported prestressed concrete beams made continuous. The program accounts for time dependent effects and has been used extensively in a recent study by Oesterle, et al [36]. A layered cross-section is used to model the beam element and the construction operation across the depth of the element is done layer by layer. The program fails however to treat the added deck as a separate element and the finite element used to model the transverse construction operation and the nonlinear behaviour of the material does not adequately account for the shift in the neutral axis from the reference and centroidal axes. This latter argument applies to many studies conducted in that area.

Ketchum at the University of California, Berkeley [27] has extensively investigated the linear analysis of segmentally erected prestressed concrete two dimensional structures. The program **SFRAME** developed for that purpose allows segmental erection along the length of the structure and uses the beam element that satisfies the classical Bernoulli-Euler kinematics to model the structural element. The element used guaranties static equilibrium between external loads and internal resisting actions. The cross-section is assumed to be uniform, constant along the length of the element and consists of a set of rectangular elements. Time dependent effects such as creep, shrinkage, temperature and relaxation are considered automatically. The program is equipped with a free-field input format interpreter that facilitates the implementation of the construction operations.

Kasti [31] extended the work of Ketchum above to analyse segmentally erected prestressed concrete composite bridges in which segmental operation might be carried out across the depth of the element. The element formulation still guaranties static equilibrium between external forces and internal resisting actions since the end actions due to internal stresses are evaluated by exact integration over the element cross-sectional area. The remaining features of the SFRAME program have been maintained. The present study is an extension of the previous work and is intended to allow nonlinear material analysis of three dimensional frame structures.

Recently, Millanes and Calzon [35] proposed a method to analyse linear elastic composite two dimensional bridges and systems. These classes of structures have been treated extensively in the research work conducted by Kasti [31] above.

1.3 Objectives of This Study

Little guidance has been provided by the AASHTO and other codes towards the analysis and design of segmentally erected structures and bridges that involves segmental erection along the length of the structure or the depth of the structural elements.

The lack of such guidance and the differences in the prediction of behaviour, particularly for complex structures, demonstrates the need for a rational approach to treat these classes of structures.

The primary purpose of the present study is to propose an original method that deals with such construction sequences and to implement the method in a computer program, called **SPCF3D**, that could be used as a powerful tool to analyse such structures as well as the standard types of construction methods.

The study is an extension of the work that has been undertaken at the University at California, Berkeley [31] on segmental erection of prestressed concrete structures. The work conducted by Ketchum [27] on linear analysis of planar frames will be used primarily as a basis for this study and implementation. The computer program written **SPCF3D** is a general purpose computer program developed to trace the nonlinear material behaviour of segmentally erected prestressed concrete composite three dimensional frame structures. The program is equipped with a command structure that facilitates the description of the different construction operations. This is absolutely necessary to analyse complex segmentally erected prestressed concrete structures with minimum effort and adequate accuracy.

CHAPTER 2 : MATERIAL TIME DEPENDENT BEHAVIOUR

Introduction

Among the significant factors affecting the response of segmentally erected prestressed concrete structures is the material time dependent behaviour of concrete, prestressing steel and mild steel throughout the loading and construction history of the structure. Consequently, proper representation of these time dependent parameters is required for an accurate prediction of the behaviour of these classes of structures.

For concrete, time dependent behaviour includes aging, creep and shrinkage effects while for prestressing steel relaxation strain is the dominant parameter. Except for temperature strains which is common to all material types, these time dependent factors are generally significant during the initial stages of loading or construction operations.

2.1 Superposition of Strains

At all times, the uniaxial strain in concrete, prestressing steel and mild steel is a function of mechanical and non-mechanical time dependent strains. Theoretically, these two categories as well as the time dependent strains that constitute the non-mechanical strains are interrelated, but for all practical purposes they are assumed independent and cumulative.

For concrete, the total uniaxial strain $\epsilon^t(t)$ at time t is the superposition of the corresponding mechanical $\epsilon^m(t)$ and non-mechanical $\epsilon^n(t)$ components :

$$\epsilon^t(t) = \epsilon^m(t) + \epsilon^n(t) \qquad \text{(Eq. 2.1.1)}$$

Where the non-mechanical component is the superposition of creep $\epsilon_c^c(t)$, shrinkage $\epsilon_c^s(t)$, aging $\epsilon_c^a(t)$ and temperature strains $\epsilon_c^T(t)$:

$$\epsilon_c^n(t) = \epsilon_c^c(t) + \epsilon_c^s(t) + \epsilon_c^a(t) + \epsilon_c^T(t) \qquad \text{(Eq. 2.1.2)}$$

Fig. 2.1.1 illustrates the concept of superposition of concrete strains.

In the case of prestressing steel, the same concept is applicable. However, the non-mechanical strains are simply the superposition of relaxation $\epsilon_p^r(t)$ and temperature $\epsilon_p^T(t)$ strains :

$$\epsilon_p^n(t) = \epsilon_p^r(t)+\epsilon_p^T(t) \qquad \text{(Eq. 2.1.3)}$$

Similarly, the mild steel non-mechanical strains consists of temperature strains $\epsilon_m^T(t)$ only.

$$\epsilon_m^n(t) = \epsilon_m^T(t) \qquad \text{(Eq. 2.1.4)}$$

A more detailed explanation of each of the above strain component is given in the following sections.

2.1.1 Mechanical Strain

In the general case where the stress-strain relationship is assumed to be nonlinear, the total stress of a given fiber $\sigma(t)$ may be expressed in terms of the corresponding total mechanical strain $\epsilon_c^m(t)$ and the strain to stress function $\epsilon\!-\!to\!-\!\sigma$ according to the following relationship :

$$\sigma(t) = \epsilon\!-\!to\!-\!\sigma \ \ (\epsilon^m(t)) \qquad \text{(Eq. 2.1.1.1)}$$

The total mechanical strain is therefore stress related and in the particular event where the material stress-strain relationship is linear, the stress to strain function $\epsilon\!-\!to\!-\!\sigma$ reduces to the standard linear relationship in total mechanical strain.

All other parameters being constant, the concrete total mechanical strain for a given constant stress at observation time t will decrease with time due to the increase in concrete compressive strength that magnifies the normalized stress-strain relationship. The initial stress-strain relationship tangent modulus is used for non-time dependent computations such as stiffness and nodal load calculations.

2.1.2 Concrete Aging Strains

Since the total concrete mechanical strain decreases with time, a fictitious corrective strain applied to the mechanical strain and referred to as aging strain is needed to account for the increase in compressive strength and modulus of elasticity over time. Consequently, the aging strain is not a physical deformation and may be expressed in terms of the stress to strain function $\sigma\!-\!to\!-\!\epsilon_n$ at time

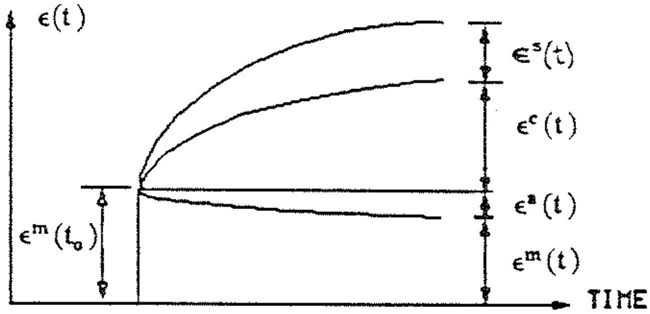


Fig. 2.1.1 DEFINITION OF STRAIN COMPONENTS

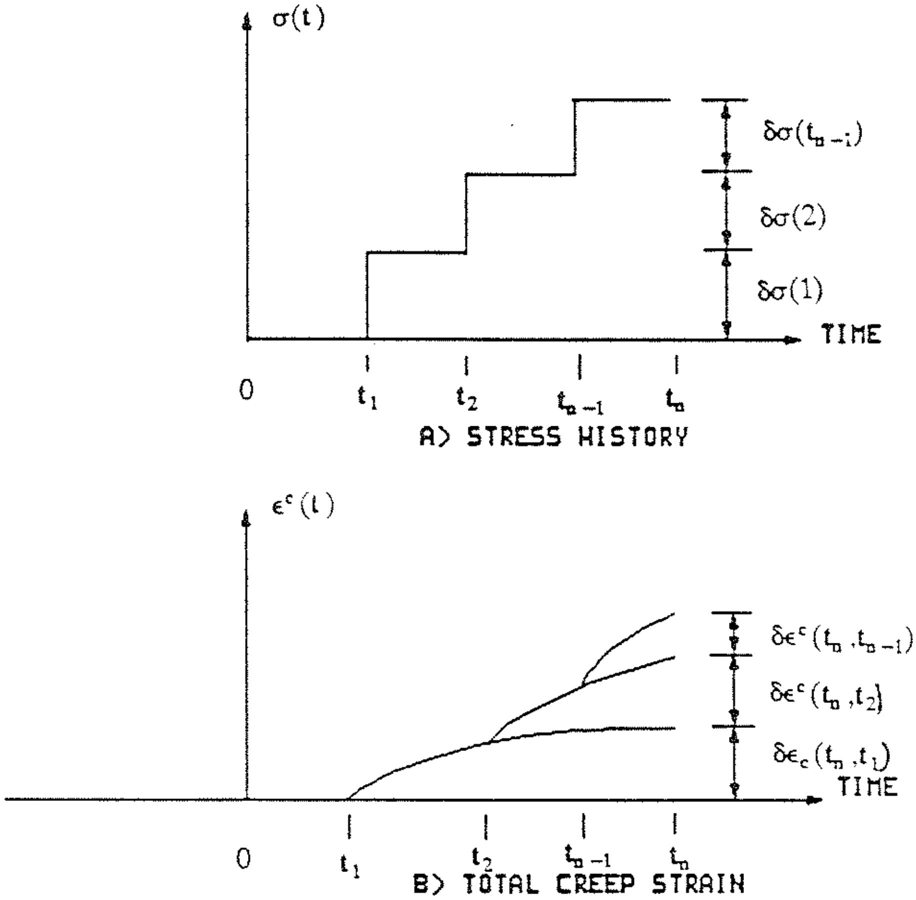


Fig. 2.1.3.1 LINEAR SUPERPOSITION OF CREEP STRAINS

step n and the total constant stress σ_{n-1} in the following manner :

$$\delta\epsilon_c^a = \sigma\text{-}to\text{-}\epsilon_n(\sigma_{n-1}) - \sigma\text{-}to\text{-}\epsilon_{n-1}(\sigma_{n-1}) \qquad \text{(Eq. 2.1.2.1)}$$

The stress to strain function in the previous expression is the inverse function of the strain to stress function mentioned in the previous sections and the above equation implicitly assumes that the aging strain is the change in the mechanical strains between consecutive concrete states of stress subjected to the same value of stress applied in a short period of time.

Internally, the consecutive states of stress are determined by conducting a linear search over the stress-strain discrete points starting with the first point and proceeding forward until the first occurence of the stress value is detected.

2.1.3 Concrete Creep Strain

Creep strain $\epsilon_c^c(t)$ in concrete is the change in total uniaxial strain under a sustained stress over a given period of time. Creep in concrete is extremely sensitive to the mix constituents, ambient conditions, specimen characteristics and loading history. A semi-empirical relationship is used to predict the effects of the first three factors, while the need for an efficient algorithm that accounts for the entire stress history with the minimum storage requirement has necessitated the use of the so-called degenerate kernel in the implementation of creep.

In the implementon of the creep model, a linear stress-creep relationship is assumed. This is not true for high level of stress which may result in a nonlinear relationship. An equivalent magnified stress has been considered in the analysis to accomodate such nonlinearity. Refer to section **2.3.3** for more details.

The variation of creep strains with a typical stress history is depicted in **Fig. 2.1.3.1**.

More details about these issues are given in the following sections.

2.1.4 Concrete Shrinkage Strain

Contrary to creep strain, shrinkage strain $\epsilon_c^s(t)$ is non-stress originated due in one part to the drying phenomenon and in another part to carbonation.

Since shrinkage strain is a non-stress originated strain, shrinkage strain is not affected by stress history and is primarily sensitive to the mix constituents and the ambient conditions.

In the current implementation, shrinkage strain distribution within each frame element cross section is assumed to be planar. The equation of such a plane is known once three points in that plane are defined. The shrinkage strain distribution is assumed to be planar whithin the frame element cross section, any other arbitrary variation may be brought to a collection of such piecewise planar distributions.

2.1.5 Prestressing Steel, Mild Steel and Concrete Temperature Strain

Prestressing steel, mild steel and concrete temperature strains $\epsilon_p^T(t)$, $\epsilon_m^T(t)$ and $\epsilon_c^T(t)$ respectively are non-stress originated strains and are due in full to temperature variation over the life of the structure. Temperature strains are sensitive to the material thermal expansion coefficient α through the following equations for prestressing steel, mild steel and concrete respectively :

$$\epsilon_p^T(t) = \alpha_p * \delta T \qquad \text{(Eq. 2.1.5.1)}$$

$$\epsilon_m^T(t) = \alpha_m * \delta T \qquad \text{(Eq. 2.1.5.2)}$$

$$\epsilon_c^T(t) = \alpha_c * \delta T \qquad \text{(Eq. 2.1.5.3)}$$

where α is the coefficient of thermal expansion, δT is the temperature change and * is the multiplication operator.

Similar to the shrinkage strain distribution, the temperature distribution within the frame element cross-section is assumed to be planar, any other arbitrary variation may be brought to a collection of such piecewise planar distributions.

2.1.6 Prestressing Steel Relaxation Strain

Relaxation is the reduction in stress of a prestressing tendon when subjected a constant strain over a given period of time. Though the material stress-strain relationship of prestressing steel is assumed to be nonlinear, the stress-strain relationship for relaxation computation is linear since relaxation is a reduction in stress or in other words an unloading stress parallel to the initial tangent modulus of the material constitutive curve. Proposed by Magura et al and used by Ketchum [27], the following equation in terms of the 0.001 offset yield stress f_{sy} and the tendon type relaxation constant $clrx$ (10.0 for ordinary strands) is used for the computation of the relaxed steel stress f_{ps} at time t (in hours after stressing) when the tendon is subjected to an initial stress f_{pi} :

$$\frac{f_{ps}}{f_{pi}} = 1.0 - \frac{\log t}{crlx} * (\frac{f_{pi}}{f_{py}} - 0.55) \qquad \text{(Eq. 2.1.6.1)}$$

Relaxation stress variation may be neglected whenever the tendon stress to yield stress ratio does not exceed 0.55. Otherwise, relaxation should be accounted for in the analysis and the above equation should be used indirectly for that purpose since it has been derived based on the assumptions that constant strains prevails during the whole operation and that the initial prestressing stress is the only applied stress. The discussion carried out in Chapter 4 outlines in details the procedure followed in the implementation.

2.2 Prediction of Time Dependent Material Properties for Concrete

The prediction of the time dependent material properties of concrete has been investigated by various researchers. Consequently, a large number of empirical expressions have been recommended as a substitute for non-existing experimental test data needed for the analysis of time dependent strains in prestressed concrete structures. ACI committee 209 and CEB - FIP recommendation are among those recommendations that have been widely used in that respect, especially in bridge design and analysis.

The program **SPCF3D** written for the current study may generate the concrete time dependent properties according to the ACI 209 and the CEB - FIP recommendations automatically before they are used in the analysis. Any arbitrary experimental results to predict the concrete parameters may be also accomodated. The procedures used are based on those developed by Ketchum [27] in his program **SFRAME**.

A general time dependent analysis of a given structure necessitates the knowledge of the value of concrete ultimate compressive strength, concrete tensile strength, stiffness or modulus of elasticity, creep strains and shrinkage strains at all times throughout the analysis. The empirical expressions used in the computer program **SPCF3D** to predict these time dependent parameters are listed in the following sections and have been converted internally to lb-in units whenever necessary.

2.2.1 ACI Committee 209 Recommendations

(a) Compressive Strength

The compressive strength of concrete $f'_c(t)$ based on a 28-day compressive strength $f'_c(28-days)$ at any time t in days after concrete is cast is computed from the following expression :

$$f'_c(t) = \frac{t}{aaci + baci*t} * f'_c(28-days) \qquad \text{(Eq. 2.2.1.1)}$$

where $aaci$ and $baci$ are parameters reflecting the type of cement used. For moist cured cement type I, $aaci = 4.0$ and $baci = 0.85$. $f'_c(t)$ and $f'_c(28-days)$ have consistent units.

(b) Tensile Strength

The tensile strength of concrete $f'_t(t)$ in psi at observation time t in terms of the compressive strength $f'_c(t)$ in psi and the unit weight W in pcf may be expressed according to the equation :

$$f'_t(t) = r_t * \sqrt{W * f'_c(t)} \qquad \text{(Eq. 2.2.1.2)}$$

The expression implicitly assumes that the tensile strength of concrete is identical to the modulus of rupture. The constant r_t ranges between 0.6 and 1.0, the default being 0.8.

(c) Modulus of Elasticity

For time dependent creep strain material parameters, the initial elastic modulus $E_i(t)$ in psi has been computed in terms of the compressive cylinder strength $f'_c(t)$ in psi and the concrete unit weight W in psf at observation time t using the following formula :

$$E_i(t) = 33.0 * W^{1.5} * [f'_c(t)]^{0.5} \qquad \text{(Eq. 2.2.1.3)}$$

The initial elastic modulus used in other than time dependent material properties computations is retrieved from the concrete material stress-strain diagram which is implicitly time dependent through the compressive strength scale factor.

(d) Creep Strain

Creep strain is the time dependent strain due to the application of a constant stress loading over a given period of time. The creep strain expression in terms of the slump correction factor $saci$, the humidity correction factor *haci*, the minimum thickness correction factor *taci*, the age at loading correction factor *τaci*, the time-loading age correction factor *ctim*(t) and the creep coefficient *ucrp* may be written in the following manner :

$$caci(t) = saci*haci*taci*\tau aci*ctim(t)*ucrp \qquad \text{(Eq. 2.2.1.4)}$$

$$saci = 0.81 + 0.0007*s$$

$$haci = 1.27 - 0.0067*H,\ H > 40\ \%$$

$$taci = 1.00 - 0.0167*(thick - 6.0),\ thick > 6\ inches$$

$$\tau aci = 1.25*\tau^{-0.118} \text{ for } 7\ days\ moist\ cured\ concrete$$

$$ctim(t) = \frac{(t-\tau)^{0.60}}{10 + (t-\tau)^{0.60}}$$

caci(t) is the ratio of creep strain at observation time t in days to the initial instantaneous strain, *ucrp* is the ratio of ultimate creep strain to the initial instantaneous strain, s is the *slump* in inches, *thick* is the minimum size of the member in inches, H is the relative humidity and τ the age at loading in days. For standard conditions, all correction coefficients take the value 1.0 except for ucrp the value of which is 2.35.

(e) Shrinkage Strain

The shrinkage strain $\epsilon_c^s(t)$ at observation time t in days in terms of the slump correction factor *saci*, member size correction factor *taci*, relative humidity correction factor *haci*, the time-age of curing correction factor *stim*(t) and the ultimate shrinkage strain *ushr* may be expressed as follows :

$$\epsilon^s(t) = saci*taci*haci*stim(t)*ushr \qquad \text{(Eq. 2.2.1.5)}$$

$$stim(t) = \frac{(t-t_0)^e}{f + (t-t_0)^e}$$

t_0 is the age at curing, f and e are parameters the value of which varies between 20 to 130 and 0.90 to 1.10 respectively. For 7 days moist cured concrete, the values of *ushr*, f and e are 800e-06, 35 and 1.0 respectively. For 3 days moist cured, these values becomes 730e-06, 55 and 1.0 respectively [34].

Fig. 2.2.1.1 presents the variation of the slump, humidity and size correction factors respectively.

2.2.2 CEB-FIP Committee Recommendations

Kristek and Smerda [15] have recently published analytical representations to approximate the figures and tables adopted by the CEB-FIP [13] for time dependent material properties estimation. These approximations are suitable for computer analysis and are incorporated in the computer program described in this report. Internally, these expressions are converted to the pound-inch system of units before they are used.

(a) Compressive Strength

The compressive strength $f'_c(t)$ at observation time t in days after casting may be approximately computed in terms of the compressive strength at time infinity from the following expression :

$$f'_c(t) = \frac{1.45*t^{0.75}}{t^{0.75}+5.5}*f'_c(\infty) \qquad \text{(Eq. 2.2.2.1)}$$

$f'_c(t)$ and $f'_c(28-days)$ have consistent units.

(b) Tensile Strength

The following equation has been used in the implementation to predict the tensile strength f'_t in terms of the compressive strength $f'_c(t)$ at observation time t in days :

$$f'_t(t) = 120*[f'_c(t)]^{0.5} \qquad \text{(Eq. 2.2.2.2)}$$

(c) Modulus of Elasticity

For material time dependent calculations only, the initial elastic modulus estimate in terms of the compressive strength $f'_c(t)$ at observation time t in days is based on the following equation :

$$E_i(t) = 66000*[f'_c(t)]^{0.5} \qquad \text{(Eq. 2.2.2.3)}$$

where E and $f'_c(t)$ are in N/cm^2.

(d) Creep Strain

Contrary to ACI 209, the creep strain according to CEB-FIP recommendations depends on the instantaneous strain based on a 28-day loading age modulus of elasticity E(28-day) and a constant sustained stress σ_0 for a loading age τ and observation time t through the following equation :

$$\epsilon_c^c(t,\tau) = \frac{\sigma_0}{E(28-day)}*\varnothing(t,\tau) \qquad \text{(Eq. 2.2.2.4)}$$

An explanation related to the remaining term $\varnothing(t,\tau)$ in the above expression is given in detail by Ketchum [27].

The creep coefficient $\varnothing(t,\tau)$ is the superposition of three components each representing a different type of creep strain. Irreversible creep strain during the initial stages of loading, recoverable delayed elastic creep strain which is independent of aging and the irreversible flow creep strain which is function of loading age are these three components.

The creep coefficient is dependent upon the geometrical properties of the cross-section and the ambient conditions. In that respect the age of concrete at observation time t in days is adjusted for temperature and cement type variations. No such modification is needed for concrete cured at 20° C (68° F) and normal ASTM type I (normal slow hardening) cement.

(e) Shrinkage Strain

The shrinkage strain $\epsilon_c^s(t,\tau)$ for a time period $[\tau,t]$ in terms of the ultimate shrinkage strain ϵ_c^{su} and the shrinkage strain function $\beta_s(t)$ is given by the following relationship :

$$\epsilon_c^s(t,\tau) = \epsilon_c^{su} * [\beta_s(t) - \beta_s(\tau)] \qquad \text{(Eq. 2.2.2.5)}$$

The ultimate shrinkage strain is the product of two components. The first component is dependent on humidity only while the other component is exclusively function of notional thickness. Similarly, the shrinkage strain function depends on the notional thickness and the concept of corrected age described in the previous section is also applicable.

2.2.3 Experimental Option

In addition to the automatic ACI 209 and the CEB-FIP empirical formula options , any arbitrary recommendation may be adaptable by providing the necessary routines related to the input, output and processing phases. Currently, only the laboratory option is implemented and requires the input of the concrete time dependent parameters at the specified discrete times. These parameters are simply the loading ages, the compressive strength, the tensile strength, the shrinkage strains and the creep strains at these observation times.

2.3 Mathematical Modelling of the Creep Strain Component of Concrete

In the following sections, an insight to the approximations introduced in the creep formulation will be laid out. Memory limitation and numerical efficiency are the main factors behind the need to store the loading history in a compact form.

2.3.1 Background

Assuming that the principle of superposition of strains is applicable and the creep strain - applied stress relationship is linear, the creep strain in concrete $\epsilon_c^c(t)$ at observation time t may be obtained through the convolution integral of an aging viscoelastic material. The need to save on storage necessitates the use of a degenerate kernel instead of the specific creep compliance function in the convolution integral. Kabir [8], Ketchum [27] and many other researchers have followed this approach where the degenerate kernel takes the form of a Dirichilet series function of the creep compliance coefficients $a_i(\tau)$, observation time t, loading age τ and the retardation times Γ_i, i being the term number in the series.

Further manipulation of the convolution integral leads to the following recursive relationship in the creep strain in terms of the stress rate $\dot{\sigma}$, creep compliance coefficients $a_i(\tau)$ and the time function $y_i(t)=t/\Gamma_i$ with its first and second derivatives $\dot{y_i}$ and $\ddot{y_i}$ respectively :

$$\ddot{\epsilon}_{ci}+(\dot{y}_i-\frac{\ddot{y}_i}{\dot{y}_i})\dot{\epsilon}_{ci} = a_i(t)*\dot{y}_i*\dot{\sigma} \qquad \text{(Eq. 2.3.1.1)}$$

2.3.2 Calculation of the Creep Strain Increment

Similar to the approach followed by Ketchum [27], three different stress and material relationships with respect to time have been assumed in the computation of creep strain from the above equations. A detailed derivation of the equations given below may be found in Ketchum [27].

In the event constant stress and constant material variation over a given time step δt is considered, the following relationship for creep strain variation $\delta\epsilon_c^c(t)$ at observation time t may be derived :

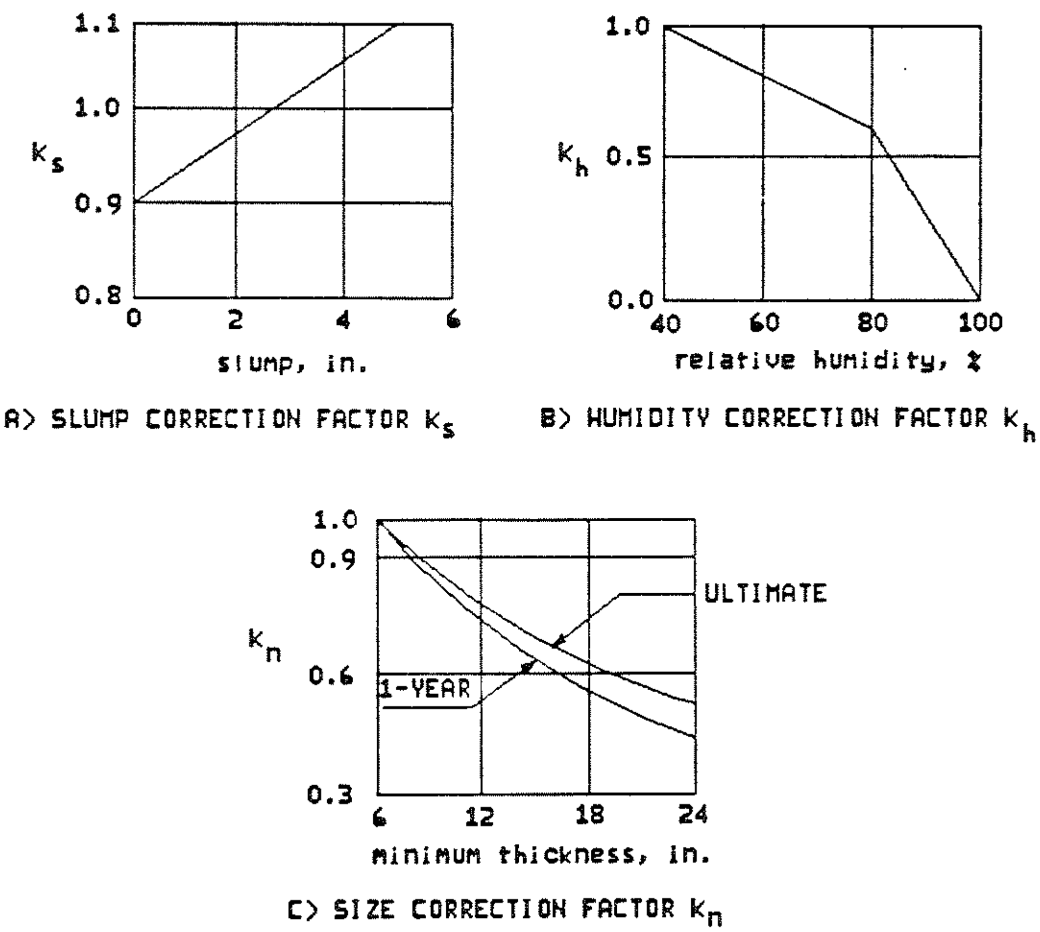


Fig. 2.2.1.1 ACI 209 SHRINKAGE CORRECTION FACTORS

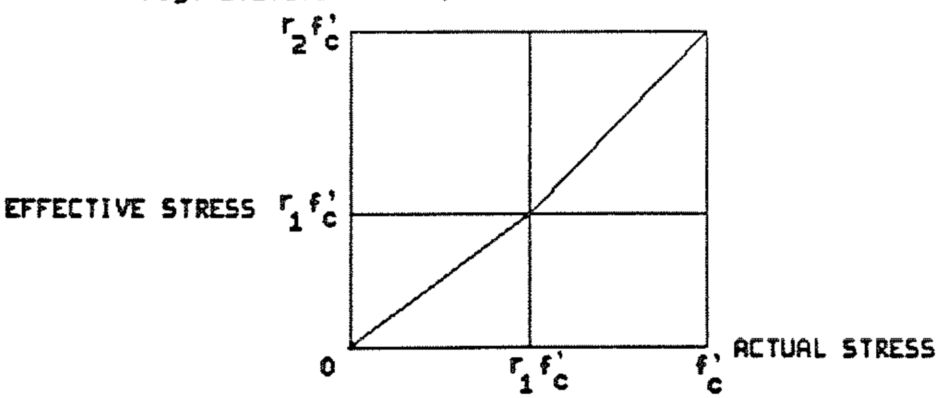


Fig. 2.3.3.1 EFFECTIVE STRESS FUNCTION TO ACCOUNT FOR NONLINEAR STRESS-CREEP RELATIONSHIP Ketchum [27]

$$\delta\epsilon_c^c = \sum A_s(t_{i-1})*[1-e^{-\delta t/\Gamma_s}] \qquad \text{(Eq. 2.3.2.1)}$$

where t_{i-1} is the observation time at the beginning of the time step, Γ_s is the retardation time and A_s is a time function updated for each time step.

In the case linear stress and constant material variation over a given time step is assumed, the recursive relationship results in the following expression :

$$\delta\epsilon_c^c(t) = R*\delta\sigma+\sum B_s(t_{i-1})*[1-e^{-\delta t/\Gamma_s}] \qquad \text{(Eq. 2.3.2.2)}$$

where R and B_s are time functions and $\delta\sigma$ is the stress change over the time step. Since iterative analysis is conducted to solve for the unknown $\delta\sigma$, the previous expression for creep strain is updated within each iteration.

When linear stress and linear material variation is adopted, the above relationship becomes :

$$\delta\epsilon_c^c = R*\delta\sigma+\sum C_s(t_{i-1,t_i})*[(\delta t/2\Gamma_i)-(1-\varnothing_i)] \qquad \text{(Eq. 2.3.2.3)}$$

where C_s is a time function. The presence of the stress change term necessitates that the expression be upated within each iteration.

2.3.3 Creep at High Stress Levels

In the previous creep strain formulation, linear stress-creep strain relationship has been assumed. This is not true particularly for stress level beyond the $0.4*f_c'$ value. The proposed effective incremental stress approach that was introduced by Becker and Bresler [7] and used by Kang [10] and Mari [25] has been adopted for this study.

The method consists in magnifying the incremental stress whenever it exceeds the value of the linear stress-creep strain limit r_1*f_c'. The following expression that relates the incremental effective stress σ_e in terms of the actual incremental stress σ and the compressive strength f_c' is shown in **Fig. 2.3.3.1** :

$$\sigma_e = \sigma \ , \ \sigma<r_1*f_c' \qquad \text{(Eq. 2.3.3.1)}$$
$$\sigma_e = \sigma*c_1 + f_c'*c_2 \ , \ \textit{otherwise}$$

where r_1 is the stress-strength ratio beyond which magnification of incremental stress is needed and r_2 is the magnification factor at maximum possible stress. The other two parameters c_1 and c_2 are related to r_1 and r_2 through the following pair of equations :

$$c_1 = \frac{r_2}{1-r_1} \; ; \; c_2 = r_1 * (1-c_1) \qquad \text{(Eq. 2.3.3.2)}$$

2.33, -0.465, 0.35 and 1.865 have been used for c_1, c_2, r_1 and r_2 respectively by Becker and Bresler [7].

2.3.4 Determination of Creep Compliance Coefficients

The method used by Kabir [8] and Ketchum [27] to evaluate the degenerate kernel creep compliance coefficients has been adopted in the present study. The method consists in a best fit representation based on a least square computation of the specific compliance curve $J(t,\tau)$ known as the kernel. The least square approximation for the kernel is repeated for a given loading age τ until the least square errors are minimized. The approach is applicable whether the specific compliance value at different observation times t are obtained from analytical empirical expressions, such as ACI 209 or CEB-FIP, or from laboratory measurements.

The procedure is then repeated for different loading ages τ in order to fill the table containing the material time dependent properties at different loading ages and at different consecutive observation times.

For CEB-FIP, the procedure is applied twice since the loading and unloading creep curves are different.

CHAPTER 3 : MATERIAL CONSTITUVE LAWS

Introduction

Concrete, mild steel and prestressing steel are the main constituents of reinforced and prestressed concrete structural elements. Despite the fact that concrete is a heterogeneous material, the assumption that all three material types are homogeneous on a macroscopical level has been commonly adopted.

Concrete as well as steel exhibits material nonlinearities. These material nonlinearities are not exclusively due to nonlinear behaviour in the material constitutive law but may be also attributed to other factors namely tensile cracking in concrete and plastic behaviour in mild steel.

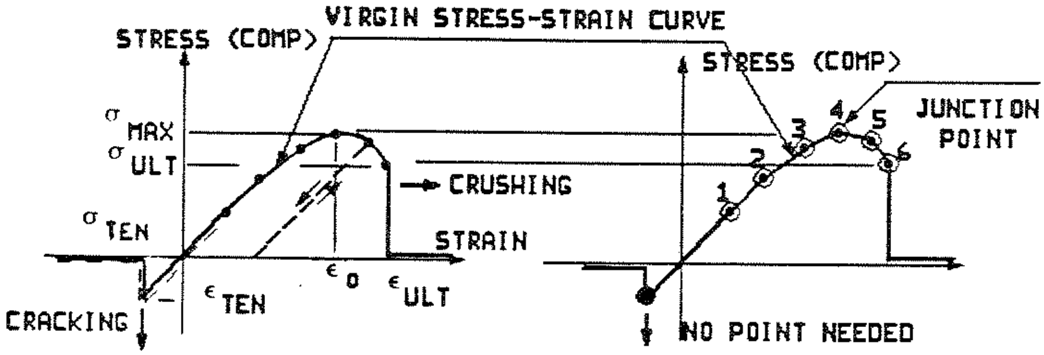
In the present study, dynamic loadings due to earthquake and wind are not taken into consideration. However, a relatively simple approach in modeling loading and unloading due to equivalent static loadings has been implemented.

Cross sectional shapes are assumed to be subdivided into a discrete, unlimited number of concrete and mild steel fibers. Each fiber or prestressed steel tendon is assumed to be in a state of uniaxial strain.

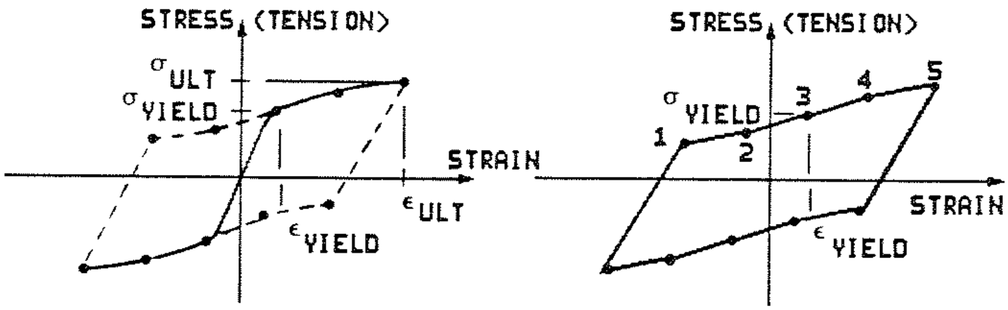
The torque-twist material relationship is assumed linear with the center of twist located on the superelement reference axis spanning the same end nodes I and J. More details related to the uniaxial stress-strain material relationships are given in the following sections.

3.1 Modelling of Material Stress-Strain Relationships

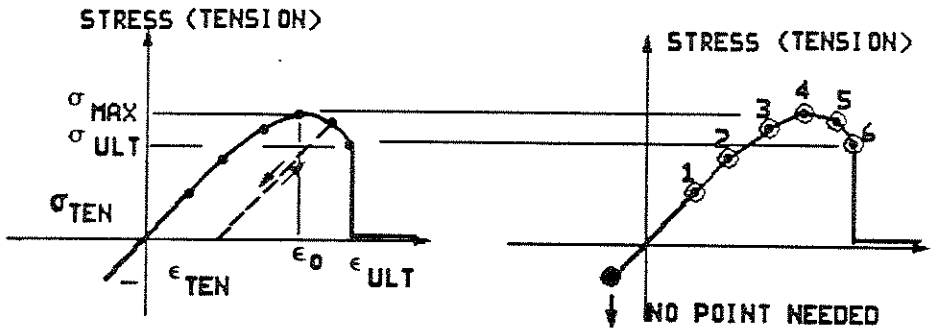
Proper modelling of material constituve laws coupled with the need for high flexibility in such representation has led to the choice of a discretized stress-strain relationship. Two main factors requiring such a flexibility in the representation are the diversity of the uniaxial stress-strain material laws used by different researchers and the versatility needed in maintaining such a representation.



A> TYPICAL CONCRETE STRESS-STRAIN AND DISCRETIZATION



B> TYPICAL MILD STEEL STRESS-STRAIN AND DISCRETIZATION



C> TYPICAL PRESTRESSING STEEL STRESS-STRAIN AND DISCRETIZATION

Fig. 3.1.1 TYPICAL MATERIAL STRESS-STRAIN LAWS FOR CONCRETE, MILD AND PRESTRESSING STEEL

Consequently, a different approach than the one adopted by many researchers such as Kang [10] and [34] has been used. In Kang's studies, the state of stress depends on the location relative to a portion of the stress-strain curve (which smooth and continuous portion of the stress curve) and stress history (loading, unloading, cracked ...) of the stress point. In the current implementation, the state of stress depends on the location of the stress point relative to a discrete point on the virgin curve, consistently taken as the discrete point lying on any segment of the virgin curve that intersects the loading/unloading branch passing through the stress point, and on the stress history.

The current version of the material constituve laws' routines can handle the representation of any stress-strain curve as long as the stress of the main virgin curve defining the stress-strain relationship is single valued with respect to strain. **Fig. 3.1.1** illustrates typical concrete, mild steel and prestressing steel stress-strain relationships with their corresponding discretized idealizations.

Incremental stress path independent analysis has been implemented for all material constitutive laws. Path independent analysis is absolutely required in iterative types of analysis where the state of stress at the end of each time step is independent of the stress path followed during the time step iteration phase.

A consistent approach common to all stress-strain material relationships has been adopted to account for path independent incremental analysis. The approach consists in locating the stress state at the end of the iteration step for a given increment of strain based on the stress state at the beginning of the time step and the stress state at the beginning of the iteration step.

3.2 Solution Strategy and Displacement Tolerances of Material Models

A large number of strategies have been used in the solution of nonlinear global equations. These strategies may be classified into two main categories : the first category **Fig. 3.2.1a**, referred to as the Incremental Load Method, consists in subdividing the total load into increments and a tangent stiffness matrix is then used to solve for the increments of displacements. In contrast, the second category **Fig. 3.2.1b**, referred to as the Iterative Method, follows an iterative approach on the unbalanced load until certain criterias are met. The tangent stiffness used in the iterative phase for the

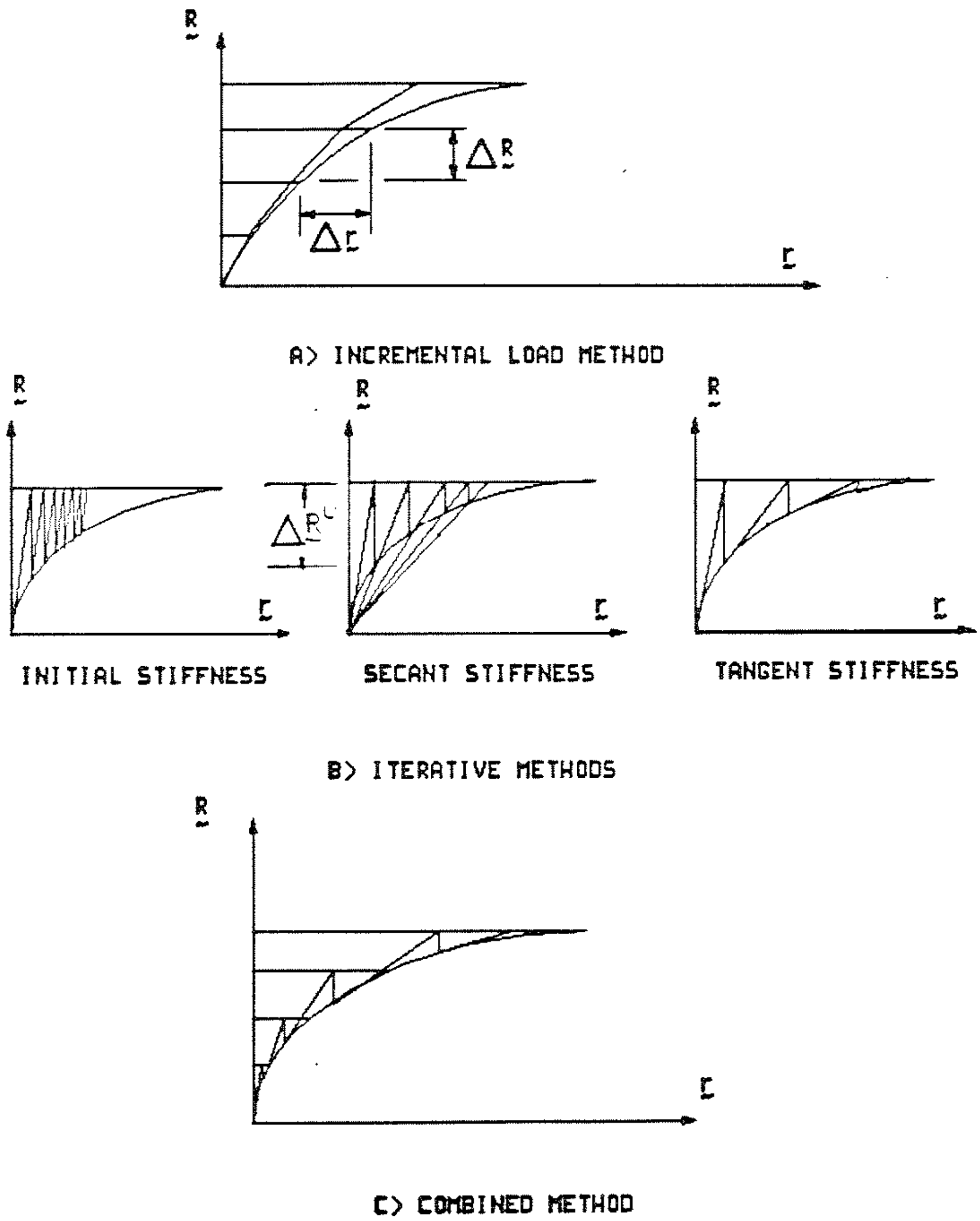
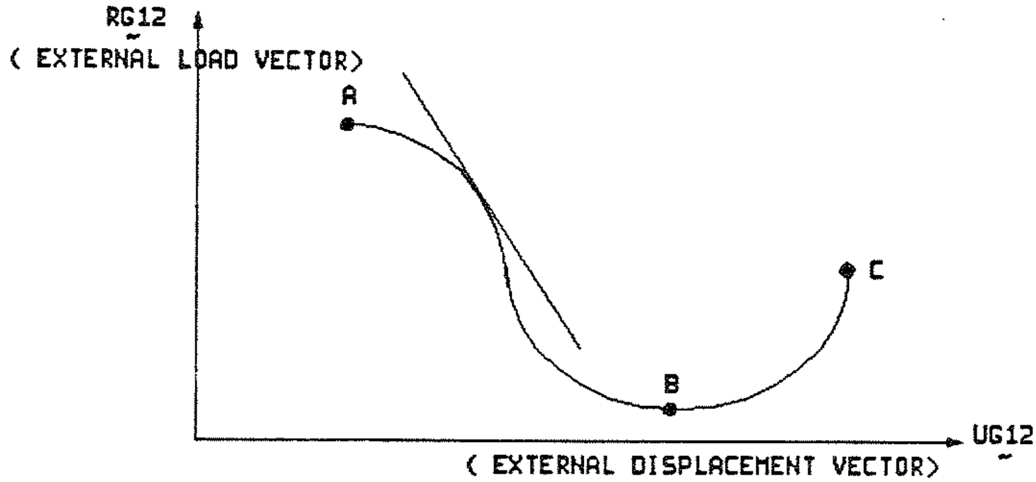
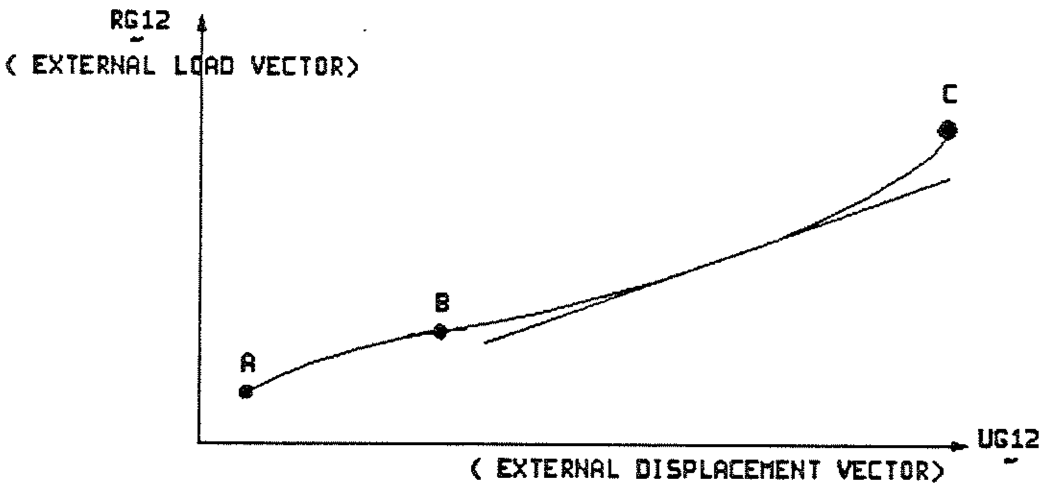


Fig. 3.2.1 SOLUTION METHODS FOR NONLINEAR EQUILIBRIUM EQUATIONS
Kang [10]



A) SNAP THRU, NON POSITIVE DEFINITE STRUCTURE STIFFNESS :
CANNOT BE ACCOMODATED



B) STIFFENING, POSITIVE DEFINITE STRUCTURE STIFFNESS : ACCEPTABLE

Fig. 3.2.2 POSSIBLE STRUCTURE GLOBAL RESPONSE Kang [10] second class of strategies, may be the initial stiffness, the secant stiffness or the tangent stiffness. The choice of a particular stiffness type is a compromise between the number of iterations required for convergence and the computational effort needed in the triangularization phase.

The current implementation can handle the second category only but cannot accomodate a non-positive definite global stiffness matrix. Consequently, 'failure' corresponds to a negative term along the structure global stiffness equivalent to a negative slope on the generalized load-displacement curve **Fig. 3.2.2**. Subsequently, of the structure. However, the individual stress-strain material laws can handle any arbitrary behaviour including non-positive definitiveness **Fig. 3.2.3**.

Two displacement tolerances are provided in order to keep the incremental strain due to nodal displacements within bounds for a given iteration. When properly used, the translational and rotational displacement increment tolerances will guard against unfavorable situations of the displacement overshoot that might occur due to structure stiffening **Fig. 3.2.4a** or load reversal **Fig 3.2.4b**.

3.3 Concrete Stress-Strain Material Law

The concrete stress-strain material relationship is largely dependent on the concrete compressive strength. The higher the strength is the more brittle and the more susceptible to fracture at smaller strain value the material becomes **Fig. 3.3.1a**. The strain rate may be another factor that affects the stress-strain relationship **Fig. 3.3.1b** but its effect has not been included in the present study.

Linearly elastic perfectly plastic **Fig. 3.2.3a**, inelastic-perfectly plastic **Fig. 3.2.3b** and the Modified Kent and Park **3.2.3c** models have been extensively used in the idealization of concrete material relationship. For the current study, a piecewise linear model **Fig. 3.1.1a** consisting of a collection of straight line segments interconnected at discrete points has been used. Such a representation provides maximum possible versatility in its ability to model a wide variety of stress-strain curves.

The user is required to provide a normalized (maximum stress is unity) stress-strain diagram which is subsequently scaled internally by the actual concrete compressive strength.

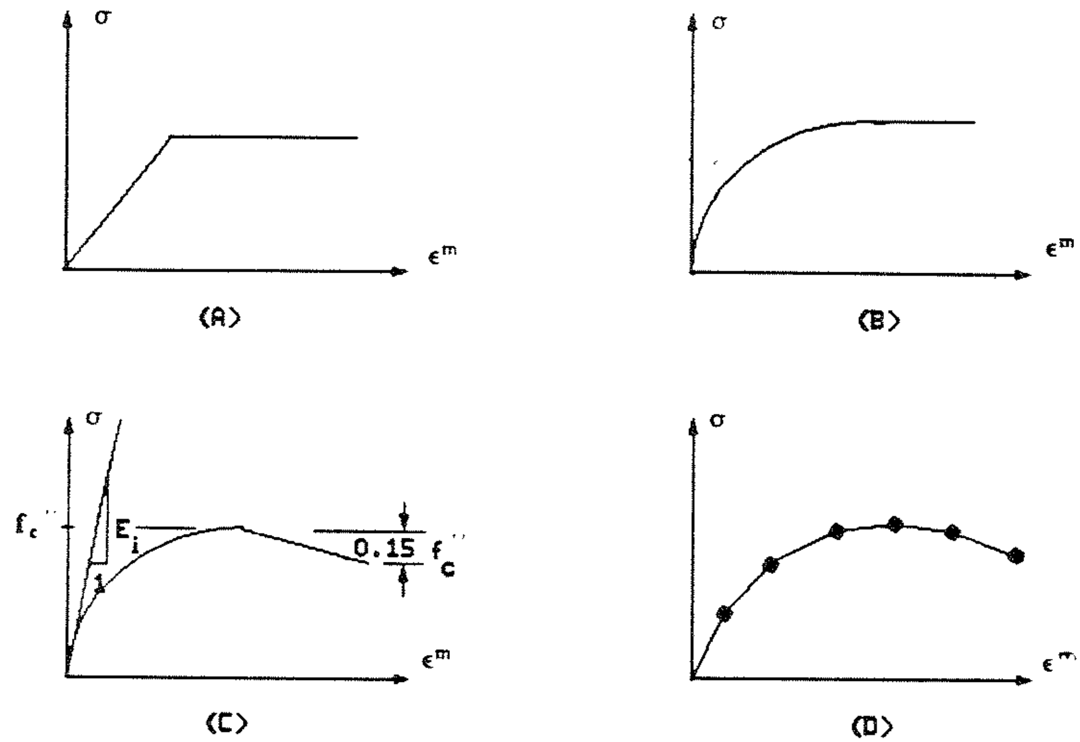


Fig. 3.2.3 IDEALIZATION OF STRESS-STRAIN CURVE

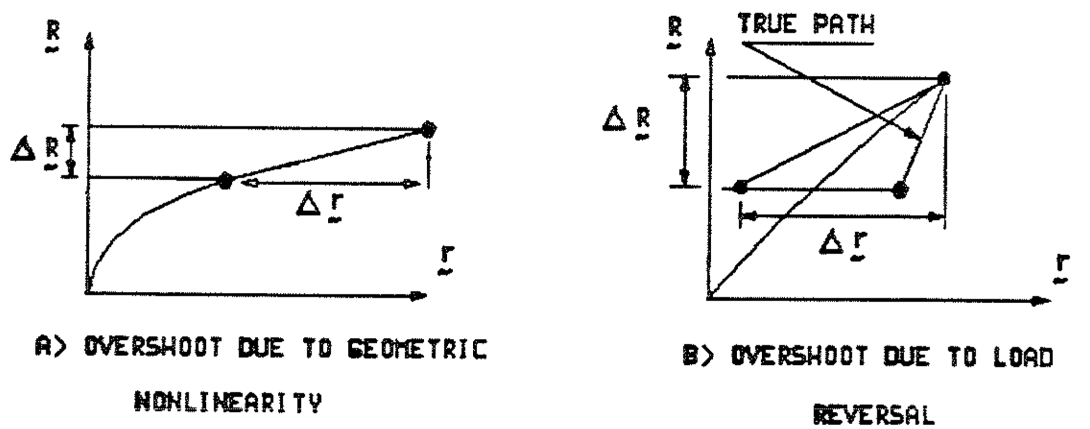
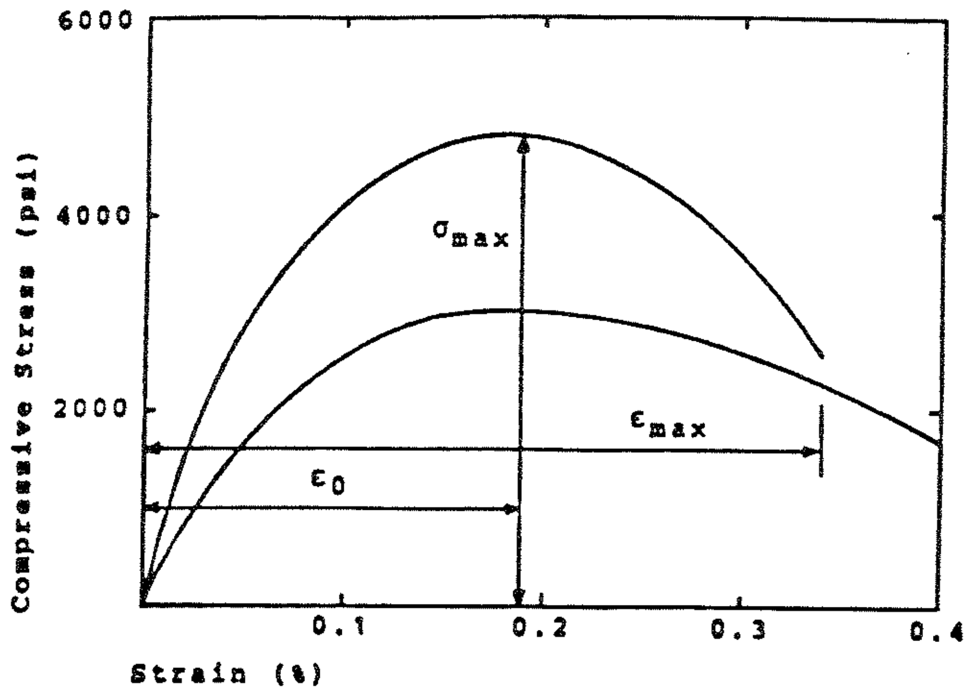
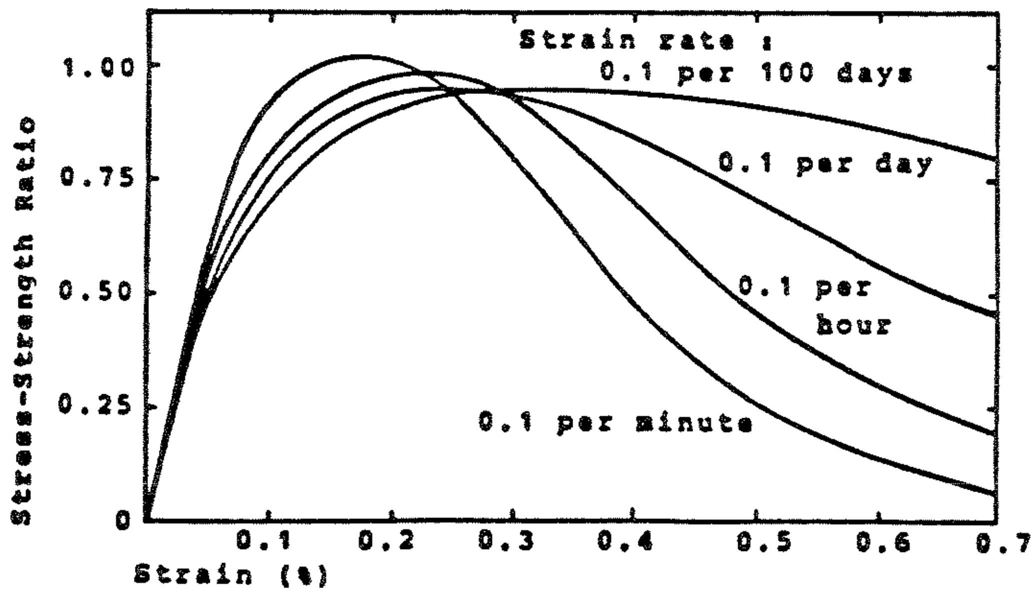


Fig. 3.2.4 CONVERGENCE TOLERANCES Kang [10]



(a) Typical Compressive Stress-Strain Curve of Concrete



(b) Influence of the Rate of Application of Strain on the Shape of the Stress-Strain Curve

Fig. 3.3.1 COMPRESSIVE STRESS-STRAIN CURVE FOR CONCRETE Kang [10]

Tension stiffnening effects have not been included in the current implementation.

3.3.1 Laboratory/Experimental Option or Empirical Curve

The computer program module handling stress-strain representation and related operations has been carefully structured to provide maximum flexibility in all material cases. In fact, any laboratory/experimental curve of arbitrary shape as well as the Modified Kent and Park options may be handled at the present time. Any other empirical curve is adaptable by providing the necessary routines that handle its input and output phases. In the case of the laboratory option, only the stress and strain coordinates of the discrete points are needed.

3.3.2 Modified Kent and Park Option

The laboratory option described previously may provide maximum flexibility whenever parametric studies are conducted. The effects of confinement on ductility and strength is ideal and must be included in such analyses otherwise large deviations from the actual behaviour may result.

The Modified Kent and Park has been widely used for that purpose and has been implemented in the current study with the minimum required input parameters needed to define the curve **Fig. 3.3.2.1**. Internally, the discretized representation of the Modified Kent and Park is subsequently derived and the stress-strain coordinates only at the discrete points are retained for future analysis.

Referring to the same figure, the equation of the parabolic portion of the stress-strain curve in terms of the compressive strength $f_c{}'$, the uniaxial strain ϵ_c and the compressive strength strain ϵ_o may be expressed in the following manner :

$$f_c = f_c{}' * (\frac{2\epsilon_c}{\epsilon_o} - (\frac{\epsilon_c}{\epsilon_o})^2),\ \epsilon =< \epsilon_o = 0.002 \qquad \text{(Eq. 3.3.2.1)}$$

Similarly, the equation of the descending branch of the stress-stress curve in terms of its slope Z, the uniaxial strain ϵ_c and the compressive strength strain ϵ_o is expressed in the following manner :

$$f_c = f_c{}' * (1 - Z * (\epsilon_c - \epsilon_o)) \qquad \text{(Eq. 3.3.2.2)}$$

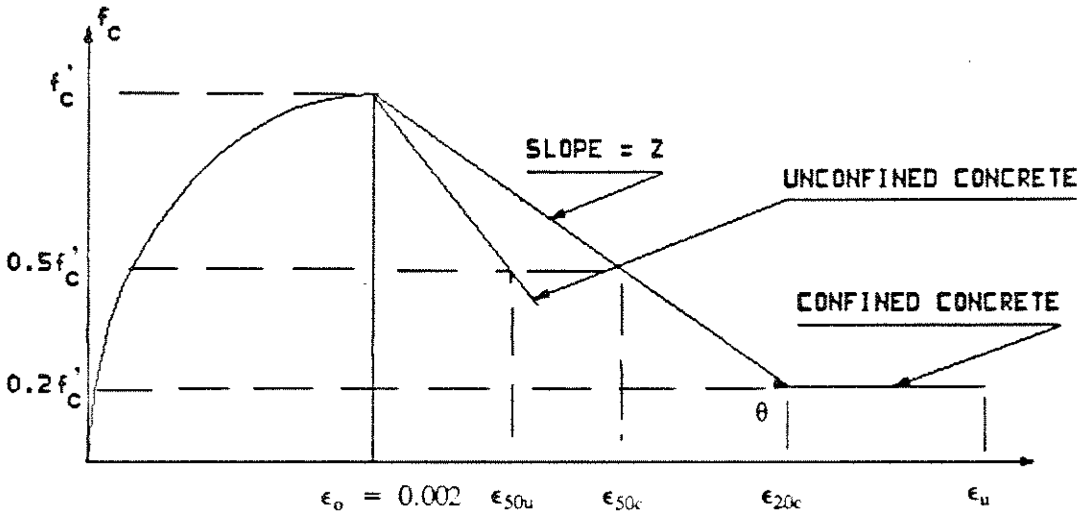


Fig. 3.3.2.1 MODIFIED KENT AND PARK CONCRETE STRESS-STRAIN RELATIONSHIP NEGLECTING ANY INCREASE IN STRENGTH WITH CONFINEMENT

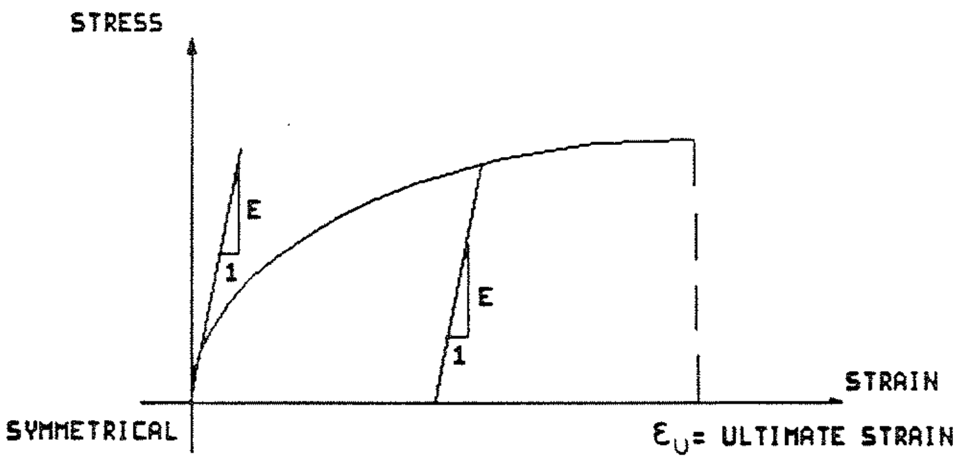


Fig. 3.4.2.1 RAMBERG-OSGOOD RELATIONSHIP

3.4 Mild Steel Stress-Strain Material Law

Unlike concrete, the properties of reinforcing steel are not generally dependent on the environmental conditions or time. In this study, a multilinear model which is symmetrical about the origin is used. Such a typical stress-strain relationship is shown in **Fig. 3.1.1b** with the corresponding discretization. The slope of the load reversal is assumed to be the same as the initial modulus of elasticity. The other difference in behaviour compared to the concrete model is that mild steel exhibits load reversal instead of tension cracking.

Bauschinger effects have not been account for in this study.

3.4.1 Laboratory/Experimental Option

Similar to the concrete inplementation, the program module handling stress-strain representation for the mild steel model and related operations has been carefully structured to provide maximum flexibility. In fact, the laboratory option may have any arbitrary shape while the empirical Ramberg-Osgood curve option is the sole additional option handled at the present time, any other desired shape being adaptable by providing the necessary routines to handle its input and output phases. For the laboratory option, the only input parameters required are the stresses and strains for the discrete points defining the stress-strain curve. Due to symmetry, only the points located in the tension quadrant need to be specified.

3.4.2 Ramberg-Osgood Option

Ramberg-Osgood stress-strain relationship with stress reversal has been implemented in order to simplify the effort required by the user for the discretized input phase. The analytical curve defined by its characteristic parameters shown in **Fig. 3.4.2.1** is internally discretized by the proper module and is therefore treated as any arbitrary piecewise linear curve.

The equation of the stress-strain curve may be expressed in the following manner :

$$\frac{\epsilon}{\epsilon_o} = \frac{\sigma}{\sigma_o} + \left(\frac{\sigma}{E}\right)^n * \frac{a}{\epsilon_o} \qquad \text{(Eq. 3.4.2)}$$

σ_o , ϵ_o , n , a , the ultimate strain ϵ_u and the number of discrete points N into which the curve is to be discretized are input parameters to be specified by the user. σ_o and ϵ_o are the stress and strain coordinates of the intersection point between the initial tangent straight line and the asymptote to the curve at infinity. The parameter n is called the knee factor and represents how steep the asymtote to the curve at infinity is.

3.5 Prestressing Steel Material Law

Contrary to the mild steel stress-strain relationship, the prestressing steel material law does not exhibit a definite yield plateau **Fig. 3.1.1.1c**. Load reversal is assumed to take place parallel to the initial tangent modulus of elasticity and no bounds have been set on the compressive stress value in such a case.

For prestressing steel, only the laboratory option has been implemented.

CHAPTER 4 : COMPUTER PROGRAM (SPCF3D) FOR THE NONLINEAR MATERIAL TIME DEPENDENT ANALYSIS OF SEGMENTALLY ERECTED REINFORCED AND PRESTRESSED CONCRETE COMPOSITE 3D FRAME STRUCTURES

Introduction

The objective of the present study has been to investigate the nonlinear behaviour time dependent effects of segmentally erected reinforced and prestressed concrete three dimensional frames. For that purpose, the computer program **SPCF3D** has been implemented to trace the time response of these classes of structures for all possible changes in structural configuration and loading application.

Step forward integration through a discretized time domain coupled with an incremental one dimensional beam element has been used to model the girders and the columns of the three dimensional frame. The superelement consisting of the newly added frame element and the existing frame elements spanning the same end nodes is treated as a Bernoulli beam element under all subsequent load increments. For traveler and tendon elements, the standard structural one dimensional beam and truss elements have been adopted respectively. The traveler element models the moving formwork while the tendon element is needed to idealize the prestressing.

In the context of segmental construction, time dependent changes in structural configuration are allowed for nodes, frame elements, prestressing tendon elements and traveler elements operations. The computer program **SPCF3D** has been equipped with a structured set of commmands to cover all the aspects of these construction phases with minimum input effort.

The architecture and organization adopted in the two dimensional **SFRAME** computer program developed by Ketchum [27] has been maintained. Consequently, some similarities exist between **SFRAME** and the three **SPCF3D** dimensional computer program developed for this study. Segmental erection across the depth of the element known as composite construction, arbitrary piecewise nonlinear material uniaxial stress-strain relationships and three dimensional tendon geometry

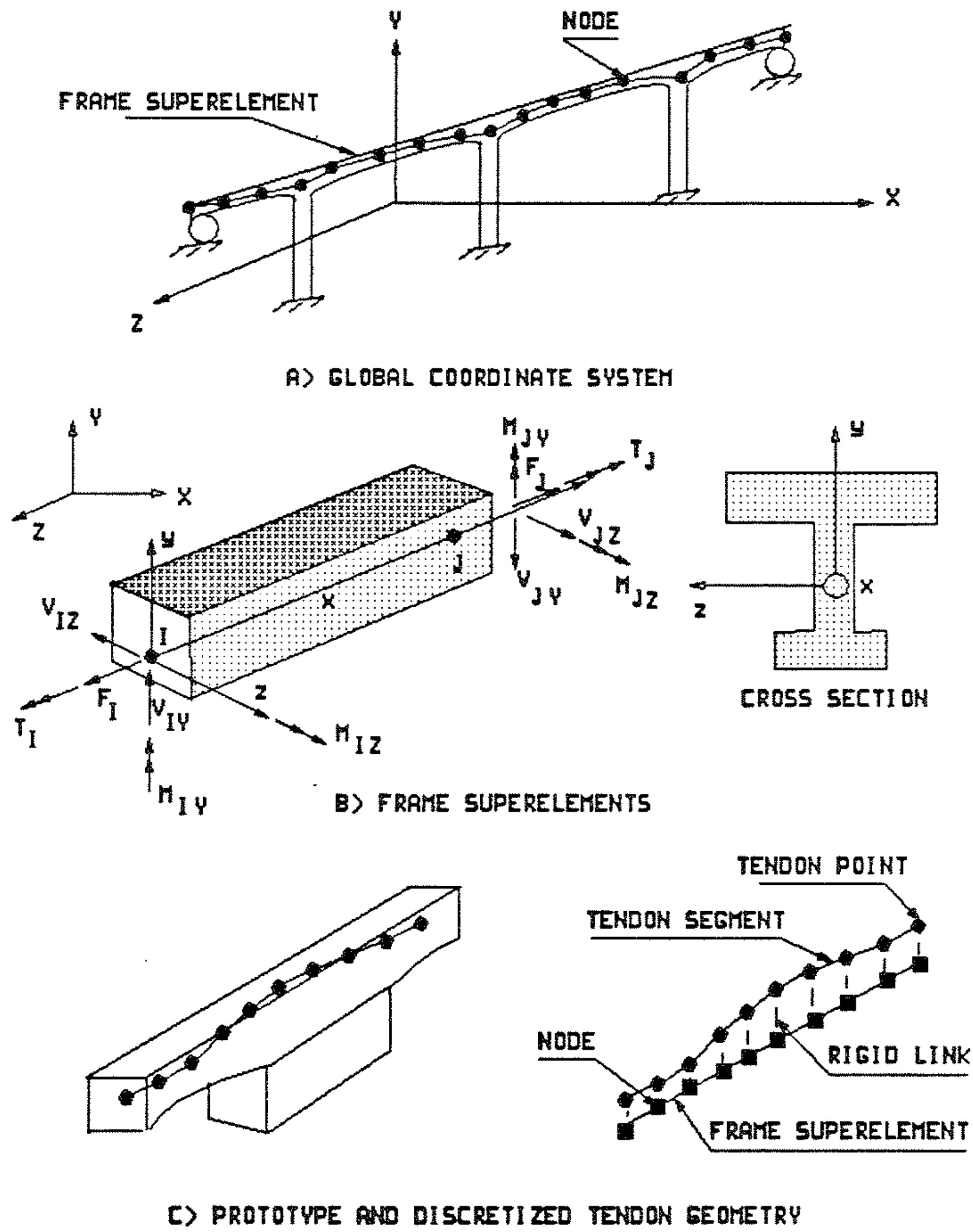


Fig. 4.1.1 STRUCTURAL MODELING

generation scheme are the main different and distinctive features of the **SPCF3D** program.

4.1 Structural Modeling

One dimensional linear elements connecting adjacent nodes are used in modeling the three dimensional frame structure **Fig. 4.1.1a**. An incremental frame element is used to model the girders and columns while a traditional frame element is used to model the travelling formwork. Prestressing is accounted for by tendon one dimensional truss elements.

Any number of such incremental beam elements **Fig. 4.1.1b** are allowed between any given two nodes in order to model the composite behaviour of the superelement cross-section. The group of such frame elements is referred to as frame superelement. Similarly, The frame element cross-section consists of an unlimited number of time dependent concrete and mild steel fibers acting parallel to the frame superelement axis joining end nodes I and J. Consequently, the frame element's cross section must be prismatic.

The global displacements at each node consist of three translational and three rotational degrees of freedom in the global X-, Y- and Z- directions. Segmental construction requires that the boundary condition specification at each degree of freedom be highly flexible and may be changed at any time during the construction and analysis phases. For that purpose, unrestrained, rigidly restrained to zero displacement or rigidly restrained to current total displacement may be specified for each nodal degree of freedom.

The dead load of each frame element is automatically applied as tributary concentrated forces at the centroidal location of the end nodes in the global Y- direction. Subsequently, these forces are internally transferred to the reference axis through the same end nodes based on static equilibrium. Consequently, dead load is modeled as concentrated forces in the global Y- direction and equivalent nodal moments in the global X- and Z- directions. Each element may be installed into the structure at any time within the solution phase but all frame elements belonging to a given superelement should be removed at any time step.

Prestressing tendons are modeled by three dimensional piecewise truss members connected to tendon end points **Fig. 4.1.1d** that are rigidly slaved to the corresponding nodal points. The shape of these tendons may be arbitrary and the tendon end points should be embedded within the superelement **Fig. 4.1.1c** for temperature strain computations. Time denpedent analysis of pre and post tensioned tendon segments may be conducted in this context.

Traveler element formulation is based on a standard frame element. No time dependent effects are included in the formulation.

All possible structure loadings are internally converted to equivalent global nodal loadings whenever necessary. Subsequently, all structure loadings may be classified into two major categories. The first category includes those types of loadings that result from externally applied disturbances such as concentrated loads, applied nodal displacements, uniformly distributed global loads and planar variations in temperature or shrinkage. The second class consists of the equivalent nodal loads of the internal stress distribution due to the applied external disturbances. For equilibrium, these two classes should be identical.

The structure configuration may be changed at any time step through the solution and construction phases. However, the structure geometry, material porperties and structural modeling should be performed before the solution phase begins.

4.2 Numerical Solution Strategy

A step forward integration through the discretized time domain coupled with a finite element formulation have been adopted in the current implementation to analyse the class of segmentally erected three dimensional composite frame structures.

The solution phase seeks the states of stress and deformation of the structure configuation at the end of each time step. The stress and strain profile within the structure is known at the beginning of the time step. Based on a direct stiffness formulation, the global stiffness matrix and the equivalent incremental nodal load vector are needed to solve the nonlinear system for the unknown incremental global displacements of the structure.

The global stiffness matrix is formed by an integration over all frame elements cross sectional fibers based on material properties prevailing at the start of the solution phase. The initial global stiffness matrix formed at the beginning of the time step may be used throughout all iteration steps or may be updated at the user request within each iteration. The equivalent incremental global load vector is the contribution of the total equivalent nodal load component including dead load, the internal load vector and the time dependent equivalent load vector due to non-mechanical strains.

Nonlinearity in the solution of the above system of equations is due to nonlinear material and time dependent effects. On the one hand due to the fact that the non-mechanical equivalent load vector is dependent on the increments in deformations in case the linear stress-creep relationship is assumed and on another hand due to the fact that the strain recovery relationship from the structure global nodal displacement which is dependent on the superelement parameters may be updated within each iteration at the user's request.

The procedure followed in the solution of these nonlinear equations is iterative in the unbalanced global nodal load vector. The solution is terminated when the stress increment and the variation in the updated non-mechanical equivalent incremental load vector for any iteration are negligible.

In any event, all externally applied increments of loads are assumed to be gradual with respect to time and the resulting deformations and stresses are assumed to vary linearly from the initial to the final state.

The solution phase of the above nonlinear system of equilibrium equations may be conceptually conceived as a sequential execution of independent computer program modules.

The solution phase is initialized with an "Assembly Stage". Such a standard structural analysis action consists in setting up the global equilibrium equations by forming the global stiffness matrix and the global equivalent incremental nodal load vector. The former matrix, the size of which depends on the current structure configuration, may be obtained from the individual contributions of the frame, traveler and tendon elements. The latter matrix is basically the contributions of the internal resisting load vector, the dead load vector and the initial equivalent non-mechanical strain load vector of the frame, traveler and tendon elements respectively. Both of these matrices are set-up using the material properties of the elements existing at the beginning of the time step. In the case of prestressing tendons, the contribution to the global stiffness is considered in all solution steps, except for the first solution step in order to model unbonded behaviour during stressing operations. In relation to superelement contributions to the global stiffness matrix, only frame and tendons elements are considered since full strain compatability would be otherwise existing between the traveler element and the superelement spanning the same end nodes which is not the case. Similar reasoning applies to the traveler element strain-displacement transformation matrix which is independent of the material properties of the corresponding superelement. The final step in the assembly phase consists in the LU factorization of the global stiffness matrix and the procedure is repeated whenever updated global stiffness is requested at each iteration.

The "Iterative Solution Stage" is next during which the strain and stress deformation profile within the structure is determined based on a forward reduction and a back substitution of the global nodal load vector. The global incremental displacement vector for the structure is extracted first from the incremental load vector using the global stiffness matrix formed in the "Assembly Stage" then the incremental displacement vector is transformed to internal incremental strain through proper strain-displacement transformation matrices of the individual elements. These internal strains are converted into stresses through the material stress-strain relationships and are consequently intregated to get the corresponding internal global nodal load vector. Upon the user's request, an intermediate iterative process in the strain-displacement relationship may be initiated. The material parameters affecting such a relationship are successively updated for the same unbalanced load vector of a given iteration until the change in their values is within a specified tolerance.

The "Iterative Stage" is terminated whenever the stress change, unbalanced load norms and strain-displacement relationship parameters are within specified tolerances. The solution of the system is performed by an active column equation solver commonly used in finite element applications.

The "Structure Update Stage" is the final phase in the solution of the nonlinear system of equations in which the time dependent and material parameters of frames, tendons and traveler elements as well as the structure total displacement and unbalanced load vectors are updated in preparation for the output and subsequent time step phases.

4.3 Loadings and Boundary Conditions

For segmental construction, the nodal degree of freedom at any time within the erection and analysis sequences may be allowed an unrestricted displacement, restrained to current total displacement or restrained to zero total displacement in which case the total displacement is set to zero value by simply applying an external displacement of equal magnitude but in opposite direction to the already existing value.

In the event the nodal degree of freedom is allowed an unrestricted displacement, that is unrestrained, a number in the nonlinear system of equations is assigned to the degree of freedom and any applied load in that direction is included in the structure global external load vector. The total displacement at that degree of freedom is consequently updated at the end of the solution phase and externally applied displacements on unrestrained degree of freedom are neglected. In the event the degree of freedom is restrained, no equation number is assigned to it nor is its total displacement updated when solution of the system of equations is performed. However, the reaction for that degree of freedom is obtained from the contribution of the external loads and the internal resisting load vectors at that point. Subsequently, such a reaction is applied to the structure whenever the degree of freedom is released.

In relation to loading, any form of external disturbance may be classified in that category since its effects will be transformed ultimately into an equivalent concentrated nodal load. Concentrated nodal loads and moments, nodal displacements, uniformly applied element loads and temperature variations belongs to such class of external disturbances.

Frame and traveler elements dead loads, with appropriate gravity multipliers, are automatically included as nodal forces in the global Y- direction and nodal moments in the XX- and ZZ- directions. Similarly, uniform loads are automatically included as nodal forces in the global X-, Y-, Z- directions. Dead load equivalent concentrated nodal forces are assumed to act at the centroidal level of the corresponding element and externally applied nodal forces or element uniform loads may be subjected to nodes and elements already installed in the current configuration only and remain active in all subsequent erection and analysis phases until removed. Uniform loads are assumed to be applied along the centerline of the frame superelement spanning its end nodes.

External displacements applied to restrained degrees of freedom only are indirectly included in the analysis as equivalent initial internal strains transformed to equivalent global nodal loads through individual element strain-displacement relationships. The internal strain deformations are computed assuming the appropriate displacement at the degree of freedom under consideration and zero displacements at all the others. Such an approach eliminates the need to introduce external springs to model applied displacements.

Temperature variations may be applied to already initialized frame elements only and are assumed active until they are explicitly removed through equal but opposite disturbances. Temperature changes are assumed to be constant over the length of the frame elements and vary linearly in any direction over the frame element cross-section, meaning planar distribution **Fig. 4.3.1**. Any tendon element embedded within such a cross section is subjected to a temperature variation based on its location within the cross-section. Similar to time dependent strains, temperature changes are treated as initial strain loading converted to equivalent global nodal loads through the strain-displacement relationship of individual elements.

In the case of frame elements, the temperature field is used directly to compute the initial strain for each fiber. For prestressing tendon elements, constant strain along the element is assumed and the temperature at the mid-length of the tendon segment is assumed to be representative. The latter value is simply the average of the end temperatures of the tendon segment. The initial stress-free temperature for each element may be specified independently for each of the elements.

4.4 Segmental Construction Operations

Segmental construction operations may be implemented through frame elements, tendon elements, traveler elements and nodal boundary conditions operations. Installation operation of any ele-

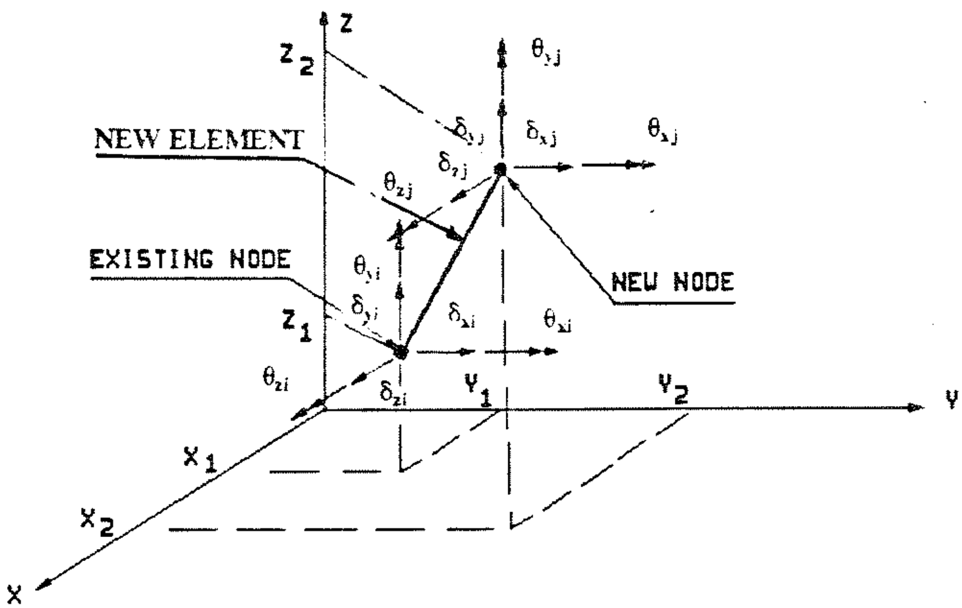


Fig. 4.4.1 NODE DISPLACEMENT INITIALIZATION

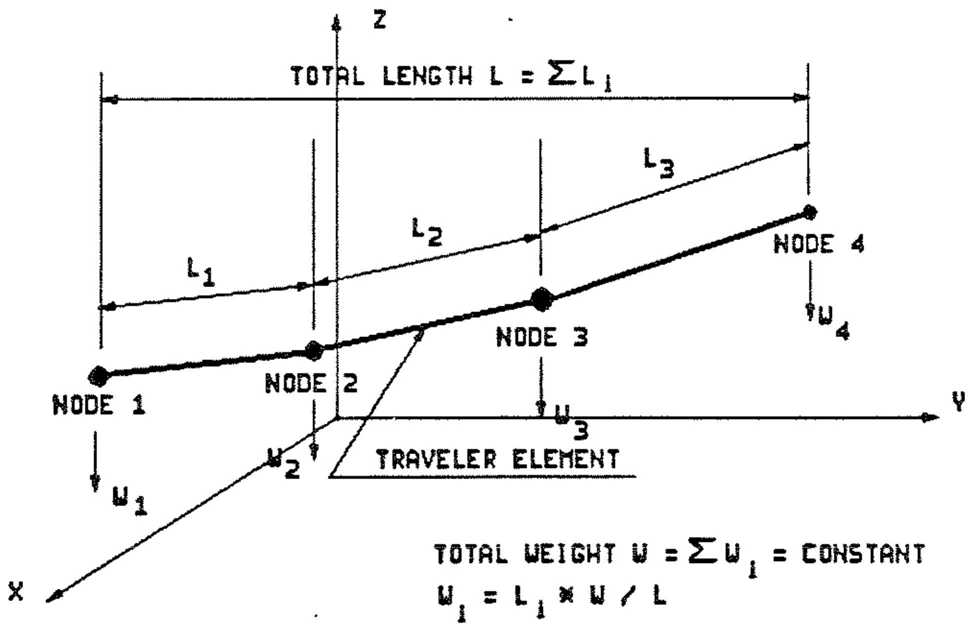


Fig. 4.4.1.2 TRAVELING FORMWORK DEAD LOAD

ment is done by including its contribution to the global stiffness matrix, superelement parameters, dead load global nodal vector, equivalent internal global nodal load vector and time dependent equivalent global nodal load vector in all subsequent steps and whenever applicable. The opposite is true for element removal.

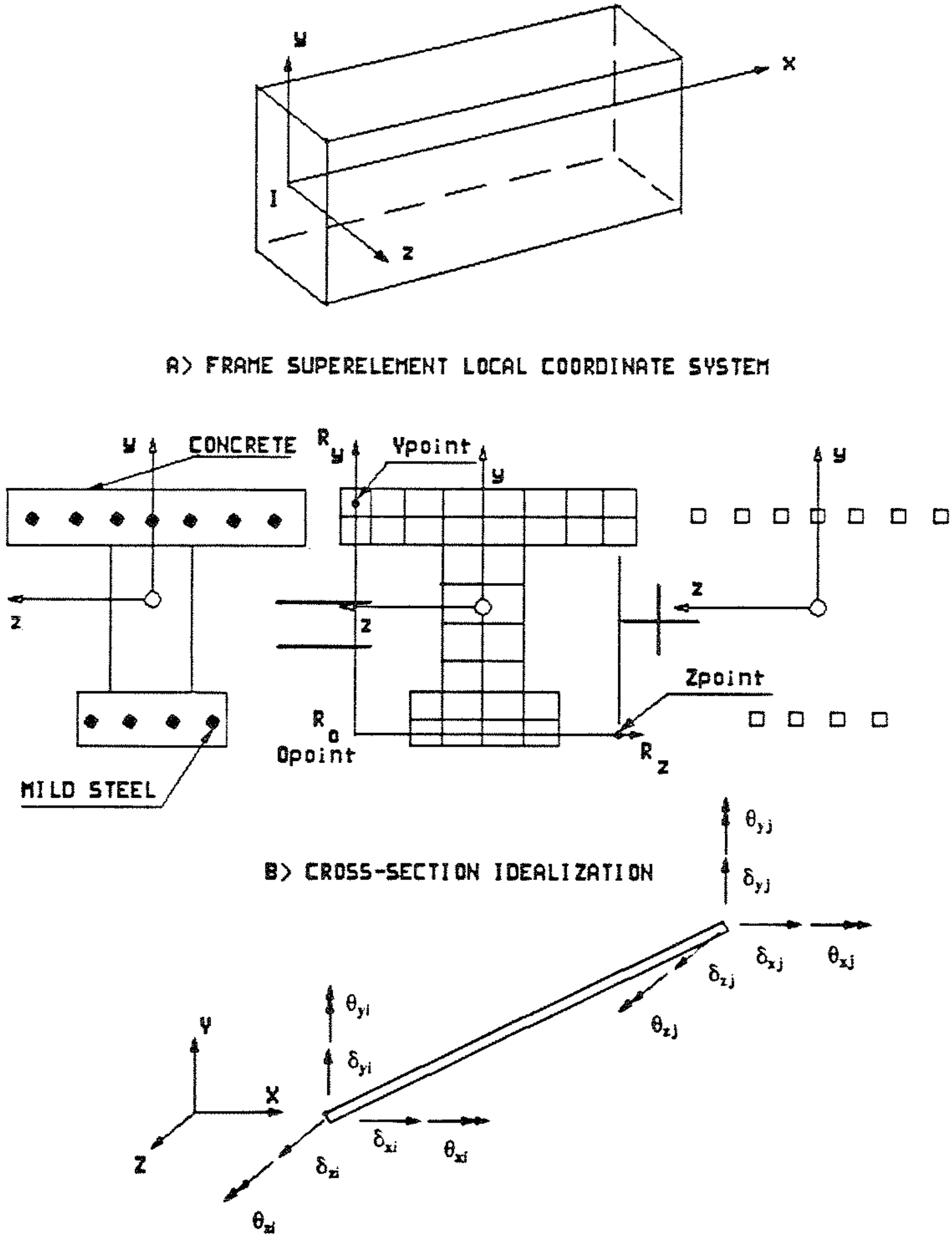
Frame elements operations consist in frame elements installation and removal at any time step through the consrtuction and analysis phases. Installation of frame elements is unrestricted but frame elements removal should be done simultaneously with the removal of all the frame elements belonging to the same superelement since the current implementation can not handle boundary shearing stresses on the interface of adjacent frame elements. In the event a frame element end node has not been already connected to another element, installation will initialize its displacement components based on a rigid body motion of the element applied to the total displacement components existing at the other end node **Fig. 4.4.1.1**.

For prestressing tendon elements, the initial tendon forces after installation and stressing from either end account for short term curvature, wobbling and anchorage slip losses.

Traveler elements may be automatically installed, moved and removed from the current location to a new location any time during the analysis. The traveler element internal strains and stresses are cleared when moved to a new location. Its contribution to the global nodal dead load vector is computed based on tributary area and total traveler dead load assumption **Fig. 4.4.1.2**. The latter is assumed constant independent of location.

The above erection operations allows the analysis of various segmental construction schemes whether the structure includes construction along the longitudinal axis or transverse erection across the superelement cross-section. A zero time interval is needed when a construction operation, assumed to be instantaneous, is to be implemented.

Nodal degrees of freedom may be restrained and unrestrained any time within the analysis. External disturbances such as forces, moments and displacements may be applied to initialized degrees of freedoms and should be removed from the structure whenever the corresponding node is not connected at any stage within the analysis. Only those nodal degrees of freedoms that belong to



A) FRAME SUPERELEMENT LOCAL COORDINATE SYSTEM

B) CROSS-SECTION IDEALIZATION

C) GLOBAL DISPLACEMENT DEGREES OF FREEDOM UG12

Fig. 4.5.1.1 FRAME SUPERELEMENT GEOMETRY AND

GLOBAL DEGREES OF FREEDOM

nodes that are attached to structural elements are included in the equilibrium equations.

4.5 Frame Element Formulation

A one dimensional incremental beam element based on classical Bernoulli-Euler kinematics has been used in **SPCF3D**. Axial and bending deformations have been included, however shear deformation has been neglected in the formulation. The frame element consists of two independent concrete and mild steel cross sections needed to model composite construction and referred to as sub-cross sections. Each sub-cross section consists of an unlimited number of fibers. Time dependent effects of creep, shrinkage, aging, relaxation and temperature has been included in the formulation as well as short term friction and slip losses in tendon elements. Nonlinear material behaviour has been considered in the present study, but nonlinear geometry has not.

4.5.1 Modelling of Frame Element Cross-Section

The frame element cross section consists of two separate concrete and mild steel filamented sub-cross sections **Fig. 4.5.1.1b**. For efficiency purposes, the procedure followed in the current implementation in defining the geometry of any sub-cross section is an improvement over the approach proposed by Warner [4] and modified by Mari [25]. The subcross section is made of an unlimited number of filaments or fibers enclosed within a rectangular grid to facilitate the input phase. The grid is of given dimensions and the filaments are defined relative to a right handed system of coordinates referred to as the rectangular system. The origin of the rectangular system coincides with a grid fiber and the position of any fiber with respect to a local system is computed automatically by the program. The local system is the right handed system whose y- and z- axes are parallel to the y- and z- axes of the rectangular system respectively and whose x- axis joins the end nodes I and J of the frame element. Normally one of the rectangular grid corners is taken as the origin of the rectangular system. The coordinates of the filaments with respect to the rectangular system are restricted to be an integer multiple of the size of the grid fibers. Such a restriction has been adopted since the storage requirement and access time of the section filaments properties is drastically reduced.

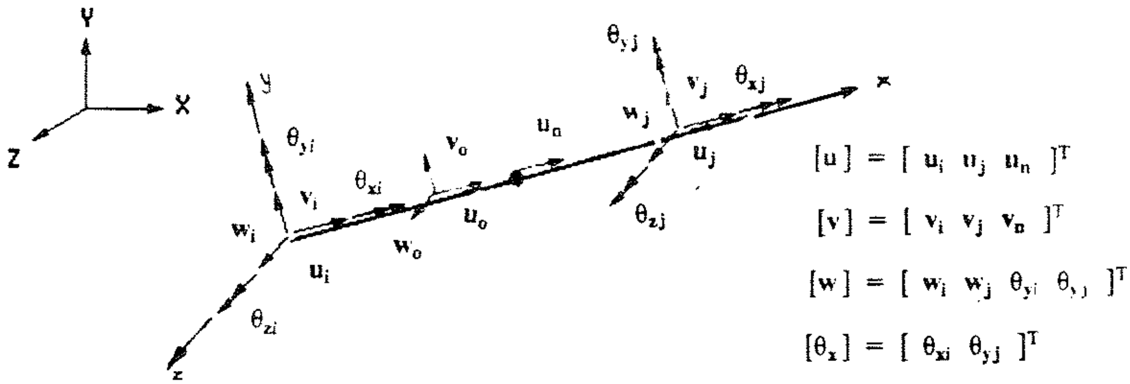
The frame superelement cross section consists of a collection of frame element sub-cross sections. The frame superelement reference axis is common to all these sub-cross sections. The shrinkage and temperature variations are assumed to have a planar distribution across the concrete sub-cross section. The equation of such a plane is known whenever three points are defined within the subcross section. For the present study, the three corner points lying on the rectangular axis and referred to as O-, Y- and Z- points have been used **Fig. 4.5.1.1**. Once the temperature plane equation has been set-up for the concrete sub-cross section, it is used in the computations of temperature strains for the mild steel sub-cross section of the same frame element.

Finally, no constraint such as symmetry about one local axis on the shape of the sub-cross section has been imposed. However, it must be always kept in mind that the rectangular y- and z- axes must be always parallel to the respective local y- and z- axis.

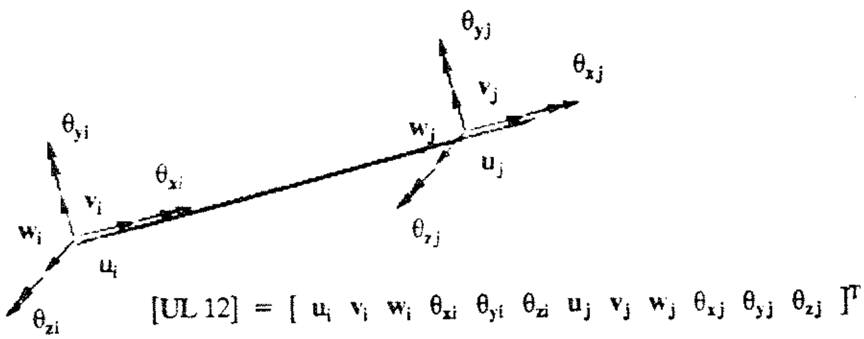
4.5.2 Geometry and Internal Displacements Fields

The reference or local axis of a given frame element may be arbitrarily chosen and does not coincide necessarily with the frame element centroidal axis. The local axis is defined by the frame element end nodes I and J located in the global (X-,Y-,Z-) right handed coordinate system **Fig. 4.5.1.1a**. Node I is the origin of the local system. The local x- axis which is common to all frame elements belonging to a given superelement is defined by the direction vector joining node I to node J. The frame element sub-cross section is defined in the local (y-,z-) plane and is assumed constant along the local x- axis of the element **Fig. 4.5.1.1b**.

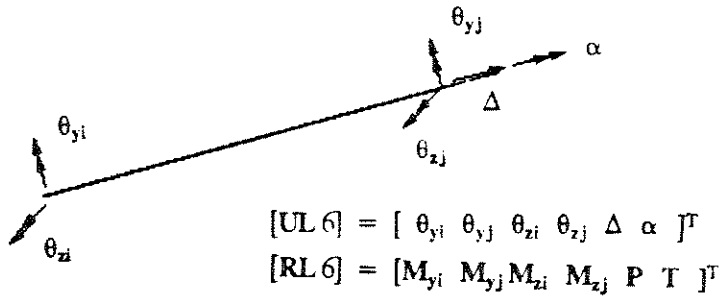
The superelement consists of a collection of an unlimited number of frame elements and is modeled as a one dimensional incremental frame element spanning between nodes I and J. A frame element may be added to the frame superelement at any time step during the analysis and the frame superelement formed by the already existing frame elements and the newly added frame element is treated as the new beam element spanning the same end nodes. The already existing frame elements are assumed to maintain their individual state of stress.



A) THIRTEEN LOCAL DISPLACEMENT DEGREES OF FREEDOM



B) REDUCED TWELVE LOCAL DEGREES OF FREEDOM



C) LOCAL DEFORMATIONAL SIX DEGREES OF FREEDOM

Fig. 4.5.2.1 LOCAL DISPLACEMENT DEGREES OF FREEDOM FOR THE FRAME SUPERELEMENT

The superelement has thirteen local displacement degrees of freedom **Fig. 4.5.2.1a**, consisting of three translations and three rotations at each end node in addition to a translation at the incompatible middle node. The latter translational degree of freedom is necessary to account for the shift in the neutral axis from the location of the local x- and centroidal axes. External loads are either directly applied to or internally converted to statically equivalent loads at the end nodes I and J at the reference axis. For distributed loads, the equivalent nodal loads consists of concentrated loads. For dead loads, these consists of additional moments about the global X- and Y- axes.

The superelement displacement field due to axial and bending deformation may be represented in terms of the three displacement components at the local superelement x- axis level and along the local x-, y- and z directions (u_0, v_0, w_0) respectively through cubic hermitian polynomial displacement interpolation functions and local displacement degress of freedom **Fig. 4.5.2.1** in the following manner :

$$u_0 = [\ \varnothing_{u1}\ \varnothing_{u2}\ \varnothing_{u3}\ ] * [\ u_I\ u_J\ u_n\ ]^T \qquad \text{(Eq. 4.5.2.1)}$$
$$v_0 = [\ \varnothing_{v1}\ \varnothing_{v2}\ \varnothing_{v3}\ \varnothing_{v4}\ ] * [\ v_I\ v_J\ \theta z_I\ \theta z_J\ ]^T$$
$$z_0 = [\ \varnothing_{z1}\ \varnothing_{z2}\ \varnothing_{z3}\ \varnothing_{z4}\ ] * [\ w_I\ w_J\ \theta y_I\ \theta y_J\ ]^T$$

where,

$$\varnothing_{u1} = 1 - s$$

$$\varnothing_{u2} = s$$

$$\varnothing_{u3} = 4 * s * (1 - s)$$

$$\varnothing_{v1} = 1 + 2 * s^3 - 3 * s^2$$

$$\varnothing_{v2} = 3 * s^2 - 2 * s^3$$

$$\varnothing_{v3} = L * (s - 2 * s^2 + s^3)$$

$$\varnothing_{v4} = L * (s^3 - s^2)$$

$$\varnothing_{z1} = \varnothing_{v1}$$

$$\varnothing_{z2} = \varnothing_{v2}$$

$$\varnothing_{z3} = - \varnothing_{v3}$$

$\varnothing_{z4} = - \varnothing_{v4}$

L is the length of the frame superelement along the local x-axis and s is the ratio $\Delta x/L$ where Δx is the relative deformation between end nodes I and J measured along the local x-axis.

The displacement u_x along the local x- axis at any point within the superelement may be obtained from the assumption of plane sections before deformation remain plane after deformation :

$$u_x = u_0 - z*\frac{dw_0}{dx} - y*\frac{dv_0}{dx} \qquad \text{(Eq. 4.5.2.2)}$$

Meaning that the displacement at any point at a distance x from node I is the summmation of three components due to direct axial displacement of the node I relative to node J and the displacements due to the rotation of the superelement cross section at the same location about the local y- and z- axes respectively.

Differentiation of the above equation with respect to x yields the following expression for the axial strain along the local x- axis ϵ_x :

$$\epsilon_x = \frac{du_0}{dx} - z*\frac{d^2w_0}{dx^2} - y*\frac{d^2v_0}{dx_2} \qquad \text{(Eq. 4.5.2.3)}$$

Substituting for u_0, v_0 and w_0 from **Eq. 4.5.2.1** into the previous relationship, the axial strain may be expressed in terms of the local thirteen displacements degrees of freedom in the following manner :

$$\epsilon_x = B_u*[u] - y*B_v*[v] - z*B_w*[w] \qquad \text{(Eq. 4.5.2.4)}$$

where,

$$B_u = [\ \varnothing_{u1,x}\ \ \varnothing_{u2,x}\ \ \varnothing_{u3,x}\ ]$$

$$B_v = [\ \varnothing_{v1,xx}\ \ \varnothing_{v2,xx}\ \ \varnothing_{v3,xx}\ \ \varnothing_{v4,xx}\ ]$$

$$B_w = [\ \varnothing_{w1,xx}\ \ \varnothing_{w2,xx}\ \ \varnothing_{w3,xx}\ \ \varnothing_{w4,xx}\ ]$$

$$[u]^T = [\ u_I\ \ u_J\ \ u_n\ ]$$

$$[v]^T = [\ v_I\ \ v_J\ \ v\ \ \theta_{zI}\ \ \theta_{zJ}\ ]$$

$$[w]^T = [\ w_I\ \ w_J\ \ w\ \ \theta_{yI}\ \ \theta_{yJ}\ ]$$

,x and ,xx denotes the first and second derivative with respect to the parameter x. The previous equation indicates that the variation of the axial strain ϵ_x within each superelement varies linearly in the x-, y- and z- directions respectively. The same is therefore true for for all frame elements belonging to a given superelement.

Unrestrained warping torsion is assumed in the formulation meaning that the torsional degrees of freedoms are uncoupled from the remaining translational and rotational degrees of freedom. Consequently, the rotation about the local x-axis θ_{x0} is assumed to vary linearly along the length of the superelement :

$$\theta_{x0} = [\ (1 - s) * s\ ] * \begin{Bmatrix} \theta_{xI} \\ \theta_{xJ} \end{Bmatrix} \qquad \text{(Eq. 4.5.2.5)}$$

and that the twist α along the local x-axis is independent of the parameter x and is given by :

$$\alpha = L^{-1} * [\ -1 \quad +1\ ] * \begin{Bmatrix} \theta x_I \\ \theta x_J \end{Bmatrix} = B_\theta * \theta_x \qquad \text{(Eq. 4.5.2.6)}$$

Such uncoupling implicitly assumes that the shear center of the superelement cross section coincides or is practically close to the origin of the local x-axis, meaning that no shear-torsion interaction exists or is practically negligible.

4.5.3 Material Constitutive Relationships

Each frame superelement cross section consists of an unlimited number of frame elements with concrete and mild steel sub-cross sections. Full compatability exists between all these sub-cross sections, is handled in the displacement field formulation and enforced at the superelement end nodes I and J at the structural level.

Each concrete and mild steel sub-cross section of a given frame element consists of an unlimited number of fibers each subjected to a uniaxial state of stress. Consequently, only the stress component in the local x- direction and the torsional moments contribute to the virtual work expressions in the incremental beam theory formulation. The transverse deformations in the local y- and z- directions are neglected.

The torsional constant J of a given frame element may be specified directly by the user as an input parameter or computed internally by the computer program **SPCF3D** through an integration over all the filaments belonging to the frame element in terms of the fiber area A_f, the distance to the reference axis of the superelement d_i and the arbitrary constant *ctor* as specified by the user according to the following equation :

$$J = ctor * A_f * \Sigma d_i^2 \qquad \text{(Eq. 4.5.3.1)}$$

In the present study, the torque-twist relationship is assumed linear. The shear modulus of elasticity G may be directly specified or computed internally by the computer program in terms of the modulus of elasticity E and Poisson's ratio according to the following equation :

$$G = \frac{E}{2 * (1 + \mu)} \qquad \text{(Eq. 4.5.3.2)}$$

(a) Concrete Components

The total uniaxial stress σ in a concrete fiber at time step t_j in terms of the corresponding total uniaxial strain ϵ_c^t and the non-mechanical strain ϵ_c^n through the strain to stress function $\epsilon - to - \sigma$ is given by the following relationship :

$$\sigma = \epsilon - to - \sigma(\epsilon_c^t - \epsilon_c^n) \qquad \text{(Eq. 4.5.3.3)}$$

The expression between parenthesis represents the total mechanical stress at time t_j and the pseudo-inelastic strain ϵ_c^n is the summation of similar contribution from creep, aging, shrinkage and temperature strains. Despite the fact that aging strains appear in the non-mechanical strain formulation, they do not contribute to the equivalent nodal load vector since they represent fictitious corrective strains to the total mehanical strains due to time dependent variations in stress-strain and strength relationships of concrete.

The creep strain contribution may be computed based on three different assumptions of stress variations over the time step. Such computations may require an iterative approach due to material nonlinearity in the solution of the global equilibrium equations and the recursive nature of the equations used in the creep model. Similar to the approach used by Ketchum [27], a linear interpolation scheme over the creep strain increment $\delta\epsilon_{ci}$ for iteration i has been implemented to speed up the convergence particularly when oscillatory behaviour is observed . Consequently, the corrected creep strain increment $\delta\epsilon_{ci}{}'$ is estimated from the following relationship :

$$\delta\epsilon'_{ci} = \delta\epsilon_{ci-1}{}' + (\delta\epsilon_{ci} - \delta\epsilon_{ci-1}{}')^* \frac{C}{1+C} \qquad \text{(Eq. 4.5.3.4)}$$

where C is the convergence interpolation factor. A value of 0.71 has been found adequate for all applications and is the default value in the pogram.

The shrinkage strain contribution is concrete age and ultimate shrinkage strain ϵ_{cu}^{s} dependent. The shrinkage distribution is assumed constant along the local x- axis and planar across each frame element sub-cross section. This approach allows an arbitrary representation of the shrinkage distribution across a superelement cross section with a number of planar shapes. The shrinkage increment over a given time step is the change in shrinkage strains measured at the end discrete time points.

A similar description applies to temperature strain contrubutions. The increment in temperature strain $\delta\epsilon_c^T$ over the discrete time interval [t_{j-1}, t_j] in terms of the temperature expansion coefficient α and the temperature function T may be expressed in the following manner :

$$\delta\epsilon_c^T = \delta T^* \alpha = [\ T(t_j) - T(t_{j-1})]^* \alpha \qquad \text{(Eq. 4.5.3.5)}$$

The aging strain contribution is a fictitious strain required to account for the increase in concrete strength with time. Since the stress-strain relationship at any given time t_j is the normalized stress-strain curve scaled with the concrete strength value, the mechanical strain at a particular stress level for a given time change δt will decrese in absolute value, other time dependent parameters assumed constant. Consequently, an increment in aging strains $\delta\epsilon_c^a$ in terms of the stress to strain function $\sigma - to - \epsilon_j$ at time t_j contributing to the non-mechanical strain is needed for the constitutive relationship to hold at any subsequent observation time :

$$\delta\epsilon_c^a = \sigma - to - \epsilon_{j-1}(\sigma(t_{j-1})) - \sigma - to - \epsilon_j(\sigma(t_{j-1})) \qquad \text{(Eq. 4.5.3.6)}$$

Time dependent material parameters are generated internally by the program before the structural configuration input phase starts and the data at different concrete ages and subsequent observation times is stored in a big table accessible at future times. For intermediate times, the program generates the values using linear interpolation.

(b) Mild Steel

Similar to the concrete material relationship, nonlinear material stress-strain behaviour has been also assumed for mild steel and its constitutive relationship in terms of the total strain ϵ_s^t, non-mechanical temperature strains ϵ_s^T and the strain to stress function $\epsilon\!-\!to\!-\!\sigma_j$ at observation time t_j may be expressed in the following fashion :

$$\sigma \;=\; \epsilon\!-\!to\!-\!\sigma_j(\epsilon_s^t \;-\; \epsilon_s^T) \qquad \text{(Eq. 4.5.3.7)}$$

The temperature distribution across the frame element mild steel sub-cross section is assumed identical to the concrete sub-cross section distribution belonging to the same frame element.

4.5.4 Stiffness and Load Computation

Frame superlement global stiffness and global nodal displacements to internal strains matrices are needed to form the contribution of the frame superelement to the global structure stiffness matrix and the equivalent global nodal load vector due to internal stresses and non-mechanical strains at each time step. The superelement contribution is treated as the sum of similar contributions from frame elements forming the superelement.

Either an initial or an updated tangent global structure stiffness matrix may be used throughout all the iterations of a given time step. The same is true for the material parameters affecting the displacement to strain matrices and the integration approach.

During the output phase, the element end stress resultants and internal load vectors are computed by integrating the computed element internal stresses over the area of the cross-section. These stiffness and load computations are performed using matrix transformations combined with 3-point gauss integration along the length of the frame superelement.

(a) Element Stiffness Matrix

The frame superlement stiffness matrix may be evaluated by applying the principle of virtual work. The superelement local stiffness matrix [k] may be then expressed in terms of the individual fiber modulus of elasticity E, the real fiber uniaxial strain ϵ_x in the local x- direction, the virtual strain $\delta\epsilon_x$, the shear modulus G, the torsional constant J, the real torsional twist angle α in the local x- direction and the virtual torsional twist angle $\delta\alpha$ in the following manner :

$$[k] = \int_V E * \epsilon_x * \delta\epsilon_x dV + \int_L GJ * \alpha * \delta\alpha dx \qquad \text{(Eq. 4.5.4.1)}$$

Substituting the local nodal displacements to uniaxial and torsional strain transformations from **Eq. 4.5.2.2** and **Eq. 2.5.2.4** respectively in the previous relationship, the following expression for the local frame superelement stiffness results :

$$[k] = \int_V [B]^T * E * [B] dV \begin{pmatrix} [u] \\ [v] \\ [w] \end{pmatrix} + \int_L B_\theta{}^T * GJ * B_\theta dx [\theta_x] \qquad \text{(Eq. 4.5.4.2)}$$

where $[B] = [\ B_u \ -y * B_v \ -z * B_w\ ]$

Further manipulation of the previous equation with the help of **Eq. 4.5.2.1** leads to the following symmetrical stiffness matrix :

$$[k] = \begin{bmatrix} K_{uu} & K_{uv} & K_{uw} & 0 \\ & K_{vv} & K_{vw} & 0 \\ & & K_{ww} & 0 \\ & & & K_{\theta\theta} \end{bmatrix} \qquad \text{(Eq. 4.5.4.3)}$$

where,

K_{uu} is a 3x3 submatrix $= \int_L B_u^T * EA * B_u \, dx$

K_{uv} is a 3x4 submatrix $= \int_L B_u^T * EZ * B_v \, dx$

K_{uw} is a 3x4 submatrix $= \int_L B_u^T * EY * B_w \, dx$

K_{vv} is a 4x4 submatrix $= \int_L B_v^T * EI_z * B_v \, dx$

K_{vw} is a 4x4 submatrix $= \int_L B_v^T * EI_{yz} * B_w \, dx$

K_{ww} is a 4x4 submatrix $= \int_L B_w^T * EI_y * B_u \, dx$

$K_{\theta\theta}$ is a 2x2 submatrix $= \int_L B_\theta^T * GJ * B_\theta \, dx$

$$EA = \int_A E \, dA$$

$$EZ = \int_A -y * E\,dA$$

$$EY = \int_A -z * E\,dA$$

$$EI_y = \int_A y^2 * E\,dA$$

$$EI_{yz} = \int_A y * z * E\,dA$$

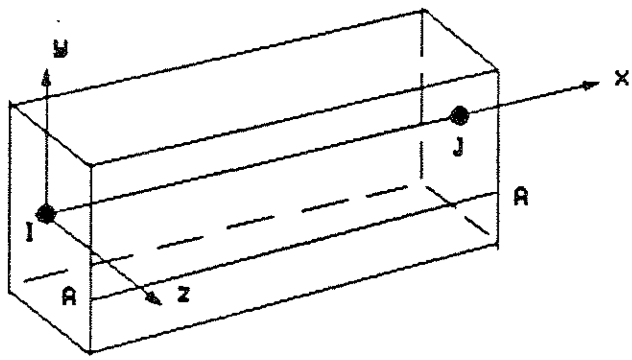
$$EI_z = \int_A z^2 * E\,dA$$

Substituting for B_u, B_v, B_w and B_θ from **Eq. 4.5.2.2** in the above leads to the following sub-matrices :

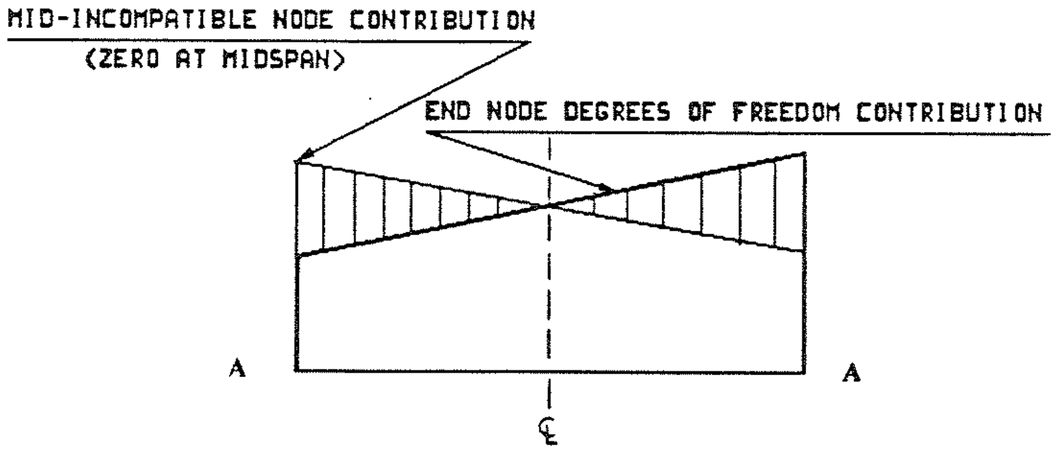
$$K_{uu} = \begin{bmatrix} \frac{1}{L} & \frac{-1}{L} & 0 \\ \frac{-1}{L} & \frac{1}{L} & 0 \\ 0 & 0 & \frac{16}{3L} \end{bmatrix} * EA \qquad \text{(Eq. 4.5.4.4a)}$$

$$K_{uv} = \begin{bmatrix} 0 & 0 & \frac{1}{L} & \frac{-1}{L} \\ 0 & 0 & \frac{-1}{L} & \frac{1}{L} \\ \frac{-8}{L^2} & \frac{8}{L^2} & \frac{-4}{L} & \frac{-4}{L} \end{bmatrix} * EZ \qquad \text{(Eq. 4.5.4.4b)}$$

$$K_{uw} = \begin{bmatrix} 0 & 0 & \frac{-1}{L} & \frac{1}{L} \\ 0 & 0 & \frac{1}{L} & \frac{-1}{L} \\ \frac{-8}{L^2} & \frac{8}{L^2} & \frac{4}{L} & \frac{4}{L} \end{bmatrix} * EY \qquad \text{(Eq. 4.5.4.4c)}$$



A) TYPICAL FRAME SUPERELEMENT



B) STRAIN DISTRIBUTION ALONG TYPICAL FIBER A-A DUE TO THE THIRTEEN LOCAL DEGREES OF FREEDOM

Fig. 4.5.4.1. INTERNAL STRAIN DISTRIBUTION WITHIN THE FRAME SUPERELEMENT

$$K_{vv} = \begin{bmatrix} \frac{12}{L^3} & \frac{-12}{L^3} & \frac{6}{L^2} & \frac{6}{L^2} \\ \frac{-12}{L^3} & \frac{12}{L^3} & \frac{-6}{L^2} & \frac{6}{L^2} \\ \frac{6}{L^2} & \frac{-6}{L^2} & \frac{4}{L} & \frac{2}{L} \\ \frac{6}{L^2} & \frac{6}{L^2} & \frac{2}{L} & \frac{4}{L} \end{bmatrix} * EI_z \qquad \text{(Eq. 4.5.4.4d)}$$

$$K_{vw} = \begin{bmatrix} \frac{12}{L^3} & \frac{-12}{L^3} & \frac{-6}{L^2} & \frac{-6}{L^2} \\ \frac{-12}{L^3} & \frac{12}{L^3} & \frac{6}{L^2} & \frac{6}{L^2} \\ \frac{6}{L^2} & \frac{-6}{L^2} & \frac{-4}{L} & \frac{-2}{L} \\ \frac{6}{L^2} & \frac{-6}{L^2} & \frac{-2}{L} & \frac{-4}{L} \end{bmatrix} * EI_{yz} \qquad \text{(Eq. 4.5.4.4e)}$$

$$K_{ww} = \begin{bmatrix} \frac{12}{L^3} & \frac{-12}{L^3} & \frac{-6}{L^2} & \frac{-6}{L^2} \\ \frac{-12}{L^3} & \frac{12}{L^3} & \frac{6}{L^2} & \frac{6}{L^2} \\ \frac{-6}{L^2} & \frac{6}{L^2} & \frac{4}{L} & \frac{2}{L} \\ \frac{-6}{L^2} & \frac{6}{L^2} & \frac{2}{L} & \frac{4}{L} \end{bmatrix} * EI_y \qquad \text{(Eq. 4.5.4.4f)}$$

$$K_{\theta\theta} = \begin{bmatrix} \frac{1}{L} & \frac{-1}{L} \\ \frac{-1}{L} & \frac{1}{L} \end{bmatrix} * GJ \qquad \text{(Eq. 4.5.4.4g)}$$

The terms EA, EZ, EY, EI_z, EI_{yz} and EI_y are to be integrated over the cross sectional area of the frame superelement. In the current implementation, exact integration will be substituted by a simple summation over the fibers of the frame elements belonging to the corresponding superelement. On the other hand, since it is not crucial to have an exact structure global stiffness in the solution phase, the mid-length state of stress has been assumed representative for the computations of these parameters.

The size of the previous frame superelement stiffness matrix **Eq. 4.5.4.2** may be reduced by static condensation, that is eliminating the explicit contribution of the mid-length incompatible translational degree of freedom. Consequently, a twelve by twelve superelement stiffness matrix relating the local twelve displacement degrees of freedoms UL12 **Fig. 4.5.2.1b** at the superelement end nodes I and J to the corresponding forces is retrieved by enforcing the zero force boundary condition at the mid-length incompatible node. Such a constraint, with the use of **Eq. 4.5.4.3** and **Eq. 4.5.4.4** provides the following equation :

$$u_n = \frac{-3L}{16EA}*[\ 0\ \ \frac{-8EZ}{L^2}\ \ \frac{-8EY}{L^2}\ \ 0\ \ \frac{+4EY}{L}\ \ \frac{-4EZ}{L}\ \ 0\ \ \frac{+8EZ}{L^2}\ \ \frac{+8EZ}{L^2}\ \ 0\ \ \frac{+4EY}{L}\ \ \frac{-4EZ}{L}\ ]*UL\,12 \qquad \text{(Eq. 4.5.4.5)}$$

Substituting the above relationship in **Eq. 4.5.4.3** provides the reduced twelve by twelve local superelement stiffness matrix. Further reduction to the six local deformational degrees of freedoms brings the size of such square matrix to six, hence increasing the efficiency and storage requirements. The resulting local stiffness matrix, referred to as the local deformational stiffness matrix, expresses the local deformational forces SL6 at the end nodes in terms of their corresponding local nodal deformational degrees of freedom UL6 **Fig. 4.5.2.1c** in the following manner :

$$SL\,6 = UL\,6\text{-}to\text{-}SL\,6_{ELM}*UL\,6 + UL\,6\text{-}to\text{-}SL\,6_{SUP}*UL\,6 \qquad \text{(Eq. 4.5.4.6)}$$

$$SL\,6 = [\ M_{yI}\ \ M_{yJ}\ \ M_{zI}\ \ M_{zJ}\ \ P\ \ T\ ]^T$$

$$UL\,6 = [\ \theta_{yI}\ \ \theta_{yJ}\ \ \theta_{zI}\ \ \theta_{zJ}\ \ \Delta\ \ \alpha\ ]^T$$

UL6-to-SL6 is the transformation matrix relating the local deformational forces and corresponding deformational displacements. $UL\,6\text{-}to\text{-}SL\,6_{ELM}$ is the transformation matrix component formed by summation of similar contribution over the frame elements belonging to the same superelement.

$$UL\,6\text{-}to\text{-}SL\,6_{ELM} = \begin{bmatrix} +4EI_y/L & -4EI_{yz}/L & +2EI_y/L & -2EI_{yz}/L & -EY/L & 0 \\ -4EI_{yz}/L & +4EI_z/L & -2EI_{yz}/L & +2EI_z/L & +EZ/L & 0 \\ +2EI_y/L & -2EI_{yz}/L & +4EI_y/L & -4EI_{yz}/L & +EY/L & 0 \\ -2EI_{yz}/L & +2EI_z/L & -4EI_{yz}/L & +4EI_z/L & -EZ/L & 0 \\ -EY/L & +EZ/L & +EY/L & -EZ/L & +EA/L & 0 \\ 0 & 0 & 0 & 0 & 0 & GJ/L \end{bmatrix}$$

$UL\,6\text{-}to\text{-}SL\,6_{SUP}$ is the transformation matrix component that is dependent upon the frame superelement parameters EA, EY and EZ in a manner that it cannot be treated as a simple addition of similar contribution over the frame elements. The reason is the fact that the ratios $\frac{EY}{EA}$ and $\frac{EZ}{EA}$ cannot be computed individually for each frame element belonging to the superelement unless all frame elements contributions to the EA parameter are known at that particular phase.

$$UL\,6\text{-}to\text{-}SL\,6_{SUP} = \begin{bmatrix} -EYY/L & +EYZ/L & -EYY/L & +EYZ/L & 0 & 0 \\ +EYZ/L & -EZZ/L & +EYZ/L & -EZZ/L & 0 & 0 \\ -EYY/L & +EYZ/L & -EYY/L & +EYZ/L & 0 & 0 \\ +EYZ/L & -EZZ/L & +EYZ/L & -EZZ/L & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

where EYY = $48*EY^2/L$, EZZ = $48*EZ^2/L$ and EYZ = 48*EY*EZ/L

The equations relating the local six deformational displacements degrees of freedom to the local twelve displacement degrees of freedom are simply :

$$\Delta = (u_J - u_I) \quad \text{(Eq. 4.5.4.7a)}$$

$$\alpha = (\alpha_J - \alpha_I) \quad \text{(Eq. 4.5.4.7b)}$$

$$\theta_{zI} = \theta_{zI} + L*(v_J - v_I) \quad \text{(Eq. 4.5.4.7c)}$$

$$\theta_{zJ} = \theta_{zJ} + L*(v_J - v_I) \quad \text{(Eq. 4.5.4.7d)}$$

$$\theta_{yI} = \theta_{yI} + L*(w_J - w_I) \quad \text{(Eq. 4.5.4.7e)}$$

$$\theta_{yJ} = \theta_{yJ} + L*(w_J - w_I) \quad \text{(Eq. 4.5.4.7f)}$$

In the above equations, the same notation is used for θ's in the local twelve displacement degrees of freedom and the local six deformational displacement degrees of freedom although in the first case such parameter indicates total absolute rotation while in the second it is only a relative rotation with respect to the local x-axis of the frame superelement.

The ELM component of the UL6-to-SL6 frame superelement local stiffness matrix may be therefore treated as a summation of similar but independent contributions from the frame elements belonging to the superelement. This is not true for the SUP component since the superelement parameters EA, EY and EZ must be known ahead before the submatrix is formed.

The local six by six frame superelement stiffness matrix **Eq. 4.5.4.6** may be transformed to the global system through a reduced global displacements to local displacements transformation matrix following the same procedure described above. In fact, the standard square matrix that converts the twelve degrees of freedom, three displacements and three rotations at each node, from one system of coordinates to another is expressed by the following relation :

$$GTOL_{12} = \begin{bmatrix} [A] & [0] & [0] & [0] \\ [0] & [A] & [0] & [0] \\ [0] & [0] & [A] & [0] \\ [0] & [0] & [0] & [A] \end{bmatrix} \qquad \text{(Eq. 4.5.4.8)}$$

where [0] is a three by three zero submatrix, and [A] is the three by three submatrix transforming three translational or three rotational orthogonal degrees of freedom from one system to another. The entry A_{ij} in [A] is given by the following equation :

$$A_{ij} = COS_{ij} \qquad \text{(Eq. 4.5.4.9)}$$

where COS_{ij} is the direction cosine of vector i component j in the rotated system with respect to the reference system. The reduced transformation matrix GTOL relating the twelve nodal global displacement degrees of freedom UG12 to the six nodal local deformational degrees of of freedoms UL6 is retrieved from **Eq. 4.5.4.8** using **Eq. 4.5.4.7** :

$$UL6 = GTOL * UG12 \qquad \text{(Eq. 4.5.4.10)}$$

$$GTOL^T = \begin{bmatrix} +J_1/L & -K_1/L & -K_1/L & +J_1/L & -I_1 & 0 \\ +J_2/L & -K_2/L & -K_2/L & +J_2/L & -I_2 & 0 \\ +J_3/L & -K_3/L & -K_3/L & +J_3/L & -I_3 & 0 \\ +J_1 & +K_1 & 0 & 0 & 0 & -I_1 \\ +J_2 & +K_2 & 0 & 0 & 0 & -I_2 \\ +J_3 & +K_3 & 0 & 0 & 0 & -I_3 \\ -J_1/L & +K_1/L & +K_1/L & -J_1/L & +I_1 & 0 \\ -J_2/L & +K_2/L & +K_2/L & -J_2/L & +I_2 & 0 \\ -J_3/L & +K_3/L & +K_3/L & -J_3/L & +I_3 & 0 \\ 0 & 0 & +J_1 & +K_1 & 0 & +I_1 \\ 0 & 0 & +J_2 & +K_2 & 0 & +I_2 \\ 0 & 0 & +J_3 & +K_3 & 0 & +I_3 \end{bmatrix}$$

I_i, J_i and K_i are the direction cosines of the local unit vectors in the local x-, y- and z- axis directions respectively.

The global stiffness corresponding to the local deformational stiffness of the superelement may now be obtained by simple matrix multiplication using the reduced transformation matrix described in **Eq. 4.5.4.10** :

$$[K] = GTOL^T * [k] * GTOL \qquad \text{(Eq. 4.5.4.11)}$$

Since the local deformational frame superelement stiffness has been split into two portions, it is expected that the same concept be also applicable for the frame superelement global stiffness matrix. The first component of such matrix is assembled by forming separately similar contributions from the frame elements belonging to that superelement, as the program loops over such elements, while the second component is formed only after all such frame elements are processed. Consequently, the former contribution is added to the global stiffness of the structure each time a frame element is considered while the latter contribution is added to the same global stiffness only after treating all the frame elements belonging to the frame superelement.

Since an iterative procedure is used in computing the incremental displacements for a given time step, no exact structure global stiffness is needed. The same is true for the frame element global stiffness matrix and as a consequence the mid length location of each fiber is considered as representative in that respect.

(b) Element Internal Strain due to Nodal Displacements

As a first step towards evaluating the element's internal strains, the explicit contribution of the translational degree of freedom at the middle incompatible node is removed from the strain expression **Eq. 4.5.2.4** using **Eq. 4.5.4.5**. The resulting strain equation in terms of the nodal global displacements degrees of freedom is then reduced with the help of **Eq. 4.5.4.7** to give the following local nodal deformational displacement UL6 to internal strain $\epsilon(x)$ equation :

$$\epsilon(x) = LTOE * UL6 \qquad \text{(Eq. 4.5.4.12)}$$

where,

$$LTOE = \begin{bmatrix} -z * [6x/L^2 - 2/L] \\ -z * [6x/L^2 - 4/L] \\ -y * [6x/L^2 - 2/L] \\ -y * [6x/L^2 - 2/L] \\ 0 \\ 0 \end{bmatrix}^T + 9 * [2 * x/L^2 - 1/L] * \begin{bmatrix} -4EY \\ -4EY \\ +EZ \\ +4EZ \end{bmatrix}^T$$

The second transformation matrix component of LTOE is simply the contribution of the mid-

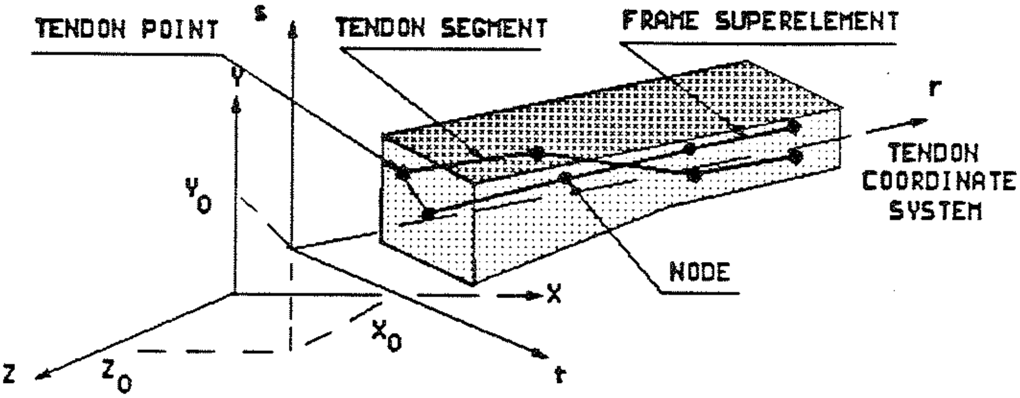
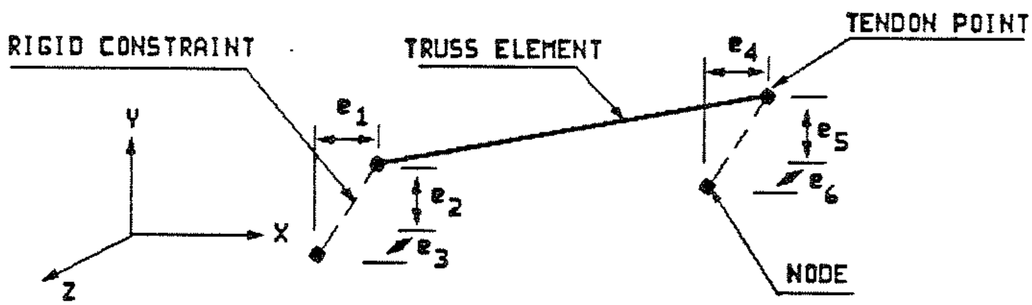
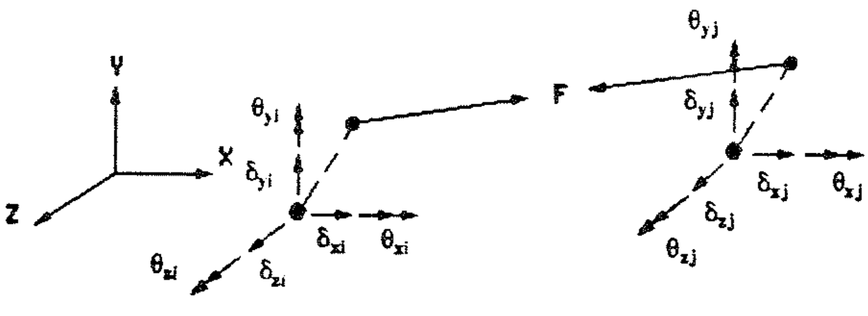


Fig. 4.6.1 TENDON COORDINATE SYSTEM AND GEOMETRY



A) TENDON SEGMENT GEOMETRY



B) TENDON SEGMENT DEGREES OF FREEDOM

Fig. 4.6.2 PRESTRESSING TENDON IDEALIZATION dle incompatible node translational degree of freedom. The distribution of the latter contribution is worthwhile noting : **Fig. 4.5.4.1**. indicates linear variation along the local x- axis with a zero value at midspan. Such a distribution is essential for the strain relationship derivation in the case of prestressing steel.

Element internal strains, consisting of the strain values at the three gauss points of each fiber belonging to the element, can be evaluated by first transforming the known global nodal displacements UG12 **Fig. 4.5.1.1b** to the local nodal deformational displacements UL6 **Fig. 4.5.2.1c** and then applying the above LTOE transformation matrix. The strain expression **Eq. 4.5.4.12** is dependent upon the frame superelement parameters which may be updated within each iteration : in fact, each time the equivalent nodal deformational displacements are computed, the corresponding strains, stresses, superelement parameters and strain-deformational displacement expression are updated and the cycle is repeated until convergence in the superelement parameters is obtained at which time the internal and unbalanced load vectors are computed and the program procedes with the next iteration.

(c) Equivalent Nodal Loads due to Initial Strains and Stresses

The equivalent local deformational forces RL6 in the local coordinate system of the frame superelement **Fig. 4.5.2.1c** due to internal resisting stress σ and initial strain ϵ vectors may be evaluated by the following integration over the volume of the frame superlement :

$$RL6 = \int_{VOL} LTOE^T * \sigma \, dV \qquad \text{(Eq. 4.5.4.13)}$$

In the case of initial strains, the equivalent nodal load vector is obtained indirectly by first updating the gauss point total stress through the stress-strain material law before the previous equation is applied to the updated total stresses. For the internal stresses only, the LTOE relationship may be updated within each iteration in the same fashion described in the previous paragraph.

The global load vector RG12 is formed by computing the local deformational vector RL6 first and then premultiplying with the local deformational forces to the global nodal forces transformation matrix. The latter matrix is simply the transpose of the GTOL transformation matrix described in **Eq. 4.5.4.10**.

When frame superelement output is requested, the gauss point stresses σ are integrated over the frame element sub-cross sections as mentioned previously. The output may includes the local deformational forces as well as the global forces for each frame element and for the superelement spanning the same end nodes. The same sign convention has been used in the output as the one described in the element formulation.

4.6 Prestressing Tendon Formulation

In the present study and similar to the procedure followed in Ketchum [27], Mari [25] and Choudhury [30], piecewise linear segments has been adopted in the geometry idealization of prestressing tendons. Such a collection of segments interconnected at tendon points **Fig. 4.6.1** is needed to allow representation of arbitrary tendon configuration in the simplest fashion. The tendon formulation explicitly assumes the tendon points displacements to be slaved to the associated nodal points displacements **Fig. 4.6.2**. Axial deformation only is included in the formulation and the tendon segments truss internal axial forces are computed accounting for instantaneous friction and anchor slip losses and effects on the force profile due to jacking forces at the end of the tendon element.

In the first time step only following the stressing operation, the tendon element is assumed unbonded. Consequently, the tendon element contribution to the global structure stiffness is disregarded in this time step and the contribution to the global nodal load vector is maintained. In all successive time steps, both contributions are considered and full displacement compatability between the tendon element and the corresponding superelements is assumed and is enforced at the end nodes I and J.

4.6.1 Definition of Tendon Geometry

In the present version, the tendon geometry input may be either provided directly by specifying the individual tendon points global coordinates, which normally requires a tremendous effort, or generated internally using an approach identical to the one adopted by Choudhury [30] and Scordelis, Ketchum, Chan and Van Der Walt [26]. The only difference that exists compared to the latter gen-

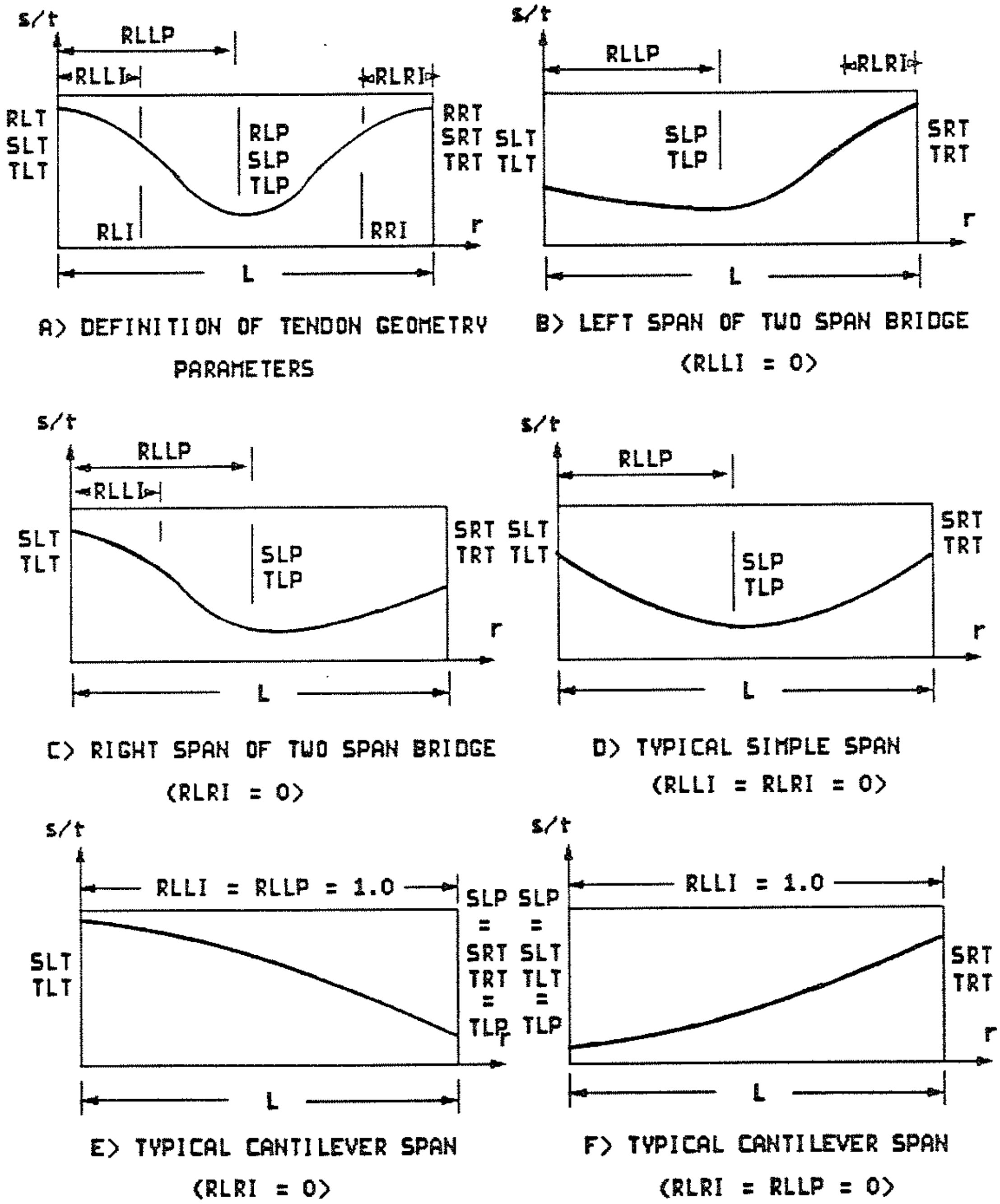


Fig. 4.6.1.1 TENDON POINT GEOMETRY GENERATION

Ketchum [27]

eration scheme is that in the current version, the tendon geometry is described relative to two projection planar surfaces while one such surface may be of a particular shape in the latter generation scheme. The reason behind such a choice in the present version is the observation that the projection of the most practical cases of prestressing steel geometry configuration on two orthogonal planes, normally those corresponding to the frame superelement local coordinate system, can be closely approximated by a simple combination of quadratic planar curves. For that purpose, the tendon element is subdivided into span sections each of which may be input or generated separately. The tendon element subdivision may be arbritrary though it was meant to facilitate the input of the actual physical subdivisions of the tendon element. In any event, the global X-, Y- and Z- coordinates of the tendon points are the primary parameters used internally to represent the tendon element geometry.

Any convenient local right handed orthonormal coordinate system (r-, s-, t-) **Fig. 4.6.1.1** may be chosen arbitrarily for each span or portion of the tendon to generate the tendon points coordinates. The right handed local coordinate system input option requires that the projected geometry of the tendon be specified in two separate orthogonal (r-, s-) and (r-, t-) planes. This input configuration has been adopted based on the observation that tendon geometry is specified in two orthogonal planes on engineering plans and drawings.

Global coordinates (X-, Y-, Z-) are retrieved from the corresponding local (r-, s-, t-) coordinates in terms of the direction cosine transformation matrix [A] of the local coordinate system with respect to the global reference system through the following relationship :

$$\begin{Bmatrix} X \\ Y \\ Z \end{Bmatrix} = \begin{Bmatrix} X_0 \\ Y_0 \\ Z_0 \end{Bmatrix} + [A] * \begin{Bmatrix} r \\ s \\ t \end{Bmatrix} \qquad \text{(Eq. 4.6.1.1)}$$

The tendon segment is modeled as a straight element spanning the two tendon end nodes I and J referred to as tendon points **Fig. 4.6.2b**. The local coordinates (r-, s-, t-) of these tendon points may be either input individually or generated internally. The tendon points' excentricities from the corresponding nodal points NI and NJ along the global directions may be expressed in terms of the tendon and nodal points global coordinates in the following manner :

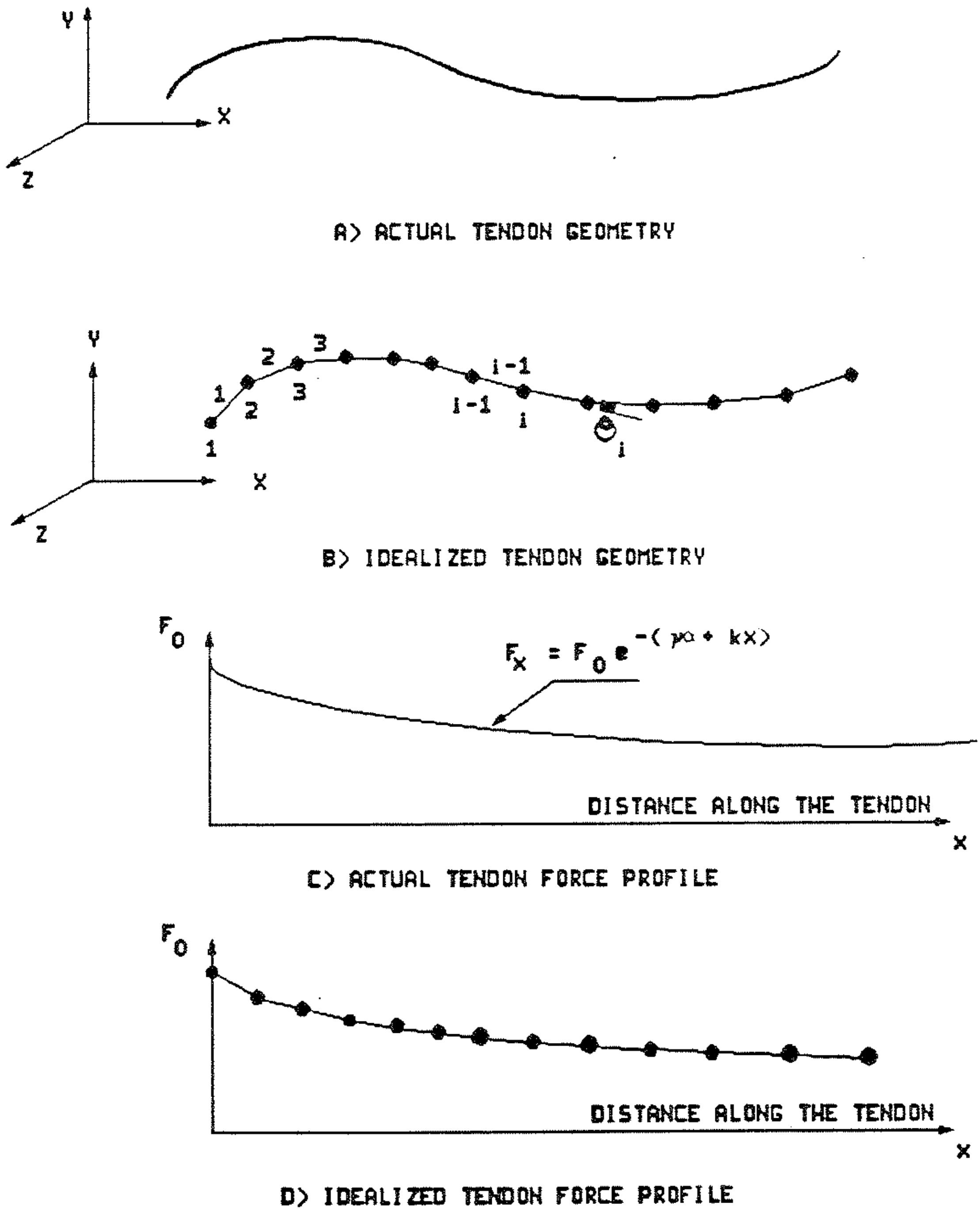


Fig. 4.6.2.1 PRESTRESSING FORCE VARIATION DUE TO FRICTION

$$
\begin{aligned}
e_{I1} &= X_I - X_{NI} \\
e_{I2} &= Y_I - Y_{NI} \\
e_{I3} &= Z_I - Z_{NI} \\
e_{J1} &= X_J - X_{NJ} \\
e_{J2} &= Y_J - Y_{NJ} \\
e_{J3} &= Z_J - Z_{NJ}
\end{aligned}
\qquad \text{(Eq. 4.6.1.2)}
$$

The internal mid-length tendon segment strain has been assumed representative in the tendon formulation. Such a local degree of freedom may be transformed to the twelve global degrees of freedom at the nodal points of the corresponding superelement **Fig. 4.6.2c** through the prestressing steel strain to displacement transformation matrix which is independent of the frame superelement material parameters.

In the event parametric generation is used for the tendon points input, the prestressing steel layout within each of the local (r- s-) and (r-, t-) planes of a given span may be assumed as a sequence of up to four quadratic curves.

The following parameters **Fig. 4.6.1.1** are needed for each of the above two planes in the event parametric generation scheme is initiated. The notation adopted by Scordelis, Ketchum, Chan and Van Der Walt [26] will be used in the following definitions :

RLT, SLT, TLT = R-, S-, T- local coordinates of Left Tendon end.

RLP, SLP, TLP = R-, S-, T- local coordinates of Low Point (having zero slope).

RRT, SRT, TRT = R-, S-, T- local coordinates of Right Tendon end.

L = (RRT - RLT) = Span Length

RLLI = (RLI - RLT) / L

RLLP = (RLP - RLT) / L

RLLI = (RRT - RRI) / L

The previous parametric generation scheme assumes displacement and slope continuity at any point along the profile. Particularly, zero slope at lt, lp and rt locations are required. The extreme locations constraint is needed to enforce horizontal slope continuity between adjacent spans.

In the event parametric generation scheme is not adequate, individual tendon points local coordinates is unavoidable. Care and proper selection of local coordinate systems may be helpful to minimize the effort.

4.6.2 Determination of Initial Forces

Friction and anchorage slip effects may be included in the computation of the initial tendon segments forces due to stressing operations. The extent of these effects along the length of the tendon is dependent on the magnitude of the stressing force applied at the jacking ends, the friction and curvature material properties.

(a) **Friction losses**

A typical variation in tendon force with the length of the tendon including frictional losses only is shown in **Fig. 4.6.2.1**. The reduction in the jacking force applied at the ends of the tendon results from the relative displacement of the tendon against the inner walls of the duct causing frictional dissipation of force. For any given portion of the tendon, the frictional force reduction may be considered as the contribution of two components. The first component is the friction loss over the straight section due to imperfection in alignment between the tendon and the duct. The second component is the friction loss over the intended curved section. In either case, the frictional force is dependent on the material frictional properties and the length of contact of the section.

Using the same notation adopted in Ketchum [27], the initial tendon force F_x is assumed to vary exponentially with the local x- distance measured from the jacking end and the following relationship has been commonly used to predict F_x in terms of the jacking force F_0, friction and wobble coefficients μ and k respectively and sections length over which the friction occurs :

$$F_x = F_0 * e^{-(\mu * \alpha + k * x)} \qquad \text{(Eq. 4.6.2.1)}$$

where α is the cumulative angle (in radians) by which the tangent to the tendon profile has changed between the jacking end and location x. Lumped mean angle changes at tendon points has been maintained in the present study. Consequently, the angle α is the cumulative sum of these

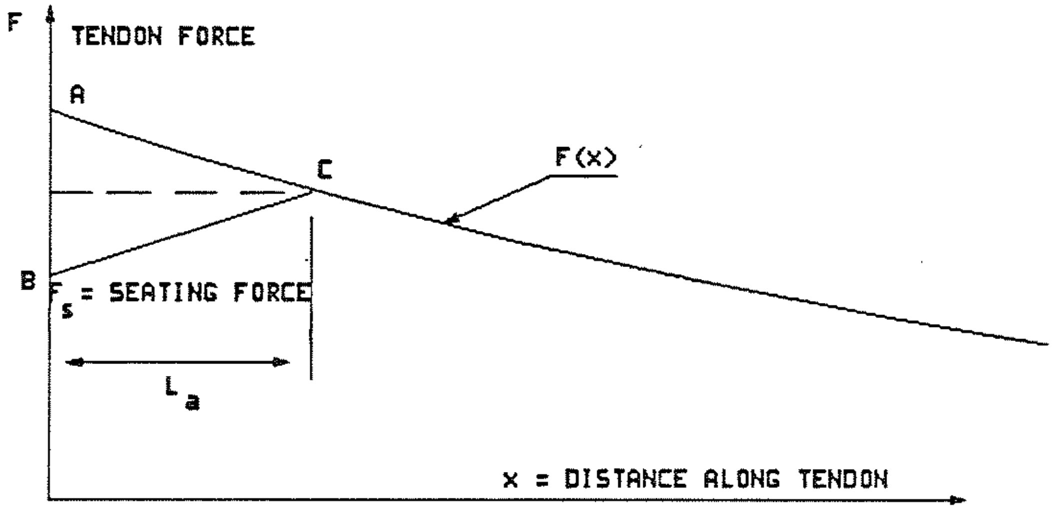


Fig. 4.6.2.2 ANCHOR SLIP LOSSES

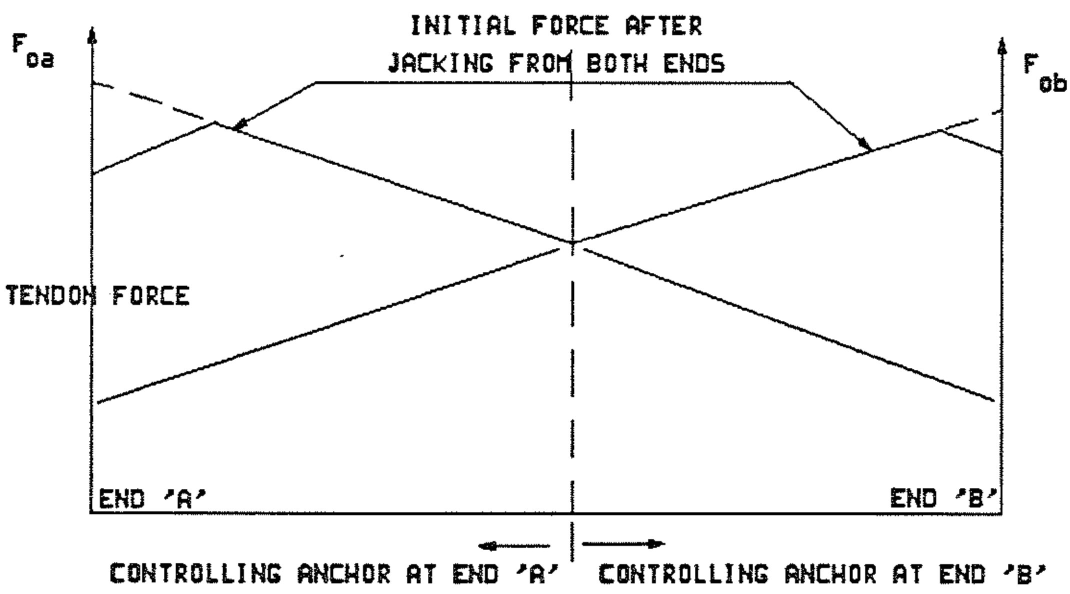


Fig. 4.6.2.3 INFLUENCE OF STRESSING PROCEDURE Ketchum [27]

lumped values occuring within the stretch of length in interest. Similarly, the parameter x in the above friction equation is the cumulative sum of the segment lengthes located within the same stretch.

Internally, the above expression is evaluated at the discrete tendon points only. Consequently, the continuous curve representing the force distribution along the tendon length is approximated by a piecewise linear curve profile **Fig. 4.6.2.1d**. The individual segment force is simply the average of the force values at the corresponding tendon nodes evaluated through the above expression.

Since the tendon material constitutive law is assumed to be nonlinear, the state of stress of each tendon segment due to jacking operation and accounting for friction losses is determined by linear search along the stress-strain virgin portion until the first occurence of the segment stress value is detected. The stressing operation is assumed complete, meaning that a state determination is performed at the end of jacking.

(b) Anchor Slip Losses

Anchor slip losses in the force profile of a given tendon is attributed to the jacking force transfer from the jacking equipment pulling the tendon to the jacking plate system incorporated into the structural member. Typical values of anchorage slip of the tendon end rangesin the vicinity of 0.5 inches.

The extent of the slip losses through the tendon force profile is normally localized in the area near the anchor at which slip losses take place **Fig. 4.6.2.2**. The extent of the slip disturbance on the tendon element may be obtained from compatability consideration in the external displacement and internal deformations. A similar approach to the one used by Ketchum [27], Mari [25] and Choudhury [30] is followed. The change in length of the tendon element over the extent L_a matches the anchor slip displacement d. The change in length of the tendon element is simply the integral over the extent L_a of the change in tendon strain $\delta\epsilon$ due to anchorage slip :

$$d = \int_0^{L_a} \delta\epsilon \, dx \qquad \text{(Eq. 4.6.2.2)}$$

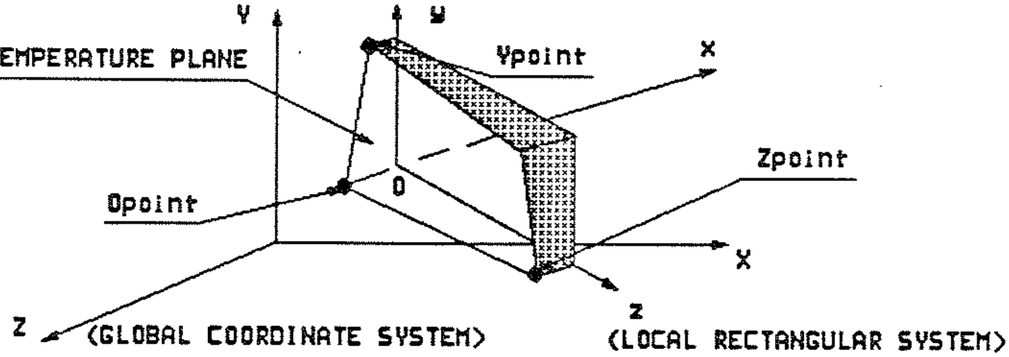


Fig. 4.3.1 TEMPERATURE POINTS DEFINING THE PLANAR DISTRIBUTION ACROSS THE FRAME ELEMENT

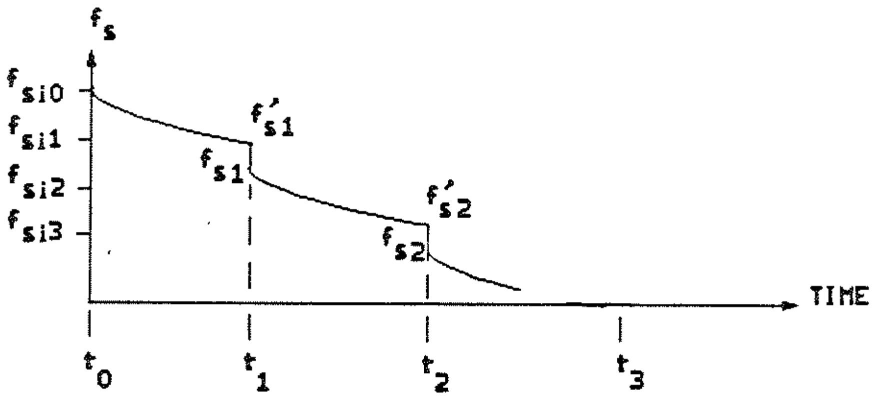


Fig 4.6.3.1 TENDON STRESS RELAXATION

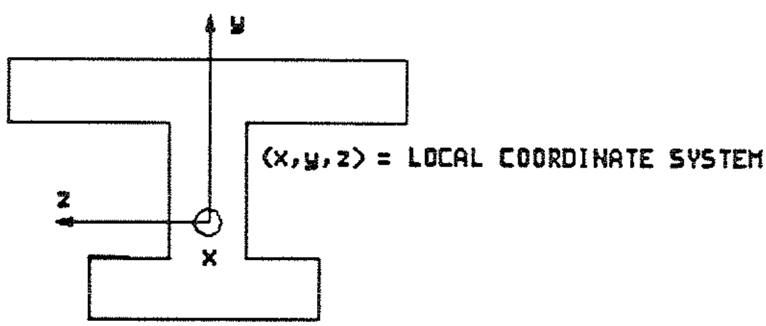


Fig. 4.7.1.1 TRAVELER CROSS-SECTION IDEALIZATION
Ketchum [27]

Internally, the above strain integral is computed from the change in force profile along the same extent L_a. In fact, the change in strain $\delta\epsilon$ in the tendon segment is proportional to the corresponding change in stress through the inverse of the elastic modulus. The latter stress change is obtained from the force change in the tendon segment through the tendon element cross sectional area proportionality constant. The change in the tendon segment force is assumed to be symmetric about the jacking force at the neutral point C in the tendon segment **Fig. 4.6.2.3**. The initial elastic modulus of the prestressing steel material relationship has been used in the strain integral computation since unloading due to anchor slip take place parallel to the initial modulus of elasticity. The assumption that the jacking operation was assumed complete has been implicitly included in the justification. Once the neutral point C is located, the segment forces along the L_a extent are reduced accordingly.

(c) Influence of Stressing Procedure

Jacking operation may be performed from either or both of the tendon element end points. In the event both ends are stressed, the above procedure should be considered for every jacking end. The final frorce profile may be then obtained from controlling profile segments retrieved from the two stressing ends individual profiles **Fig. 4.6.2.3**. Controlling profile segment is the collection of discrete force points belonging to an individual force profile the values of which exceed the corresponding values on the other individual force profile.

(d) Segment Forces

As it has been mentioned previously, the force in a given tendon segment is the average value of the segment points forces computed using the exponential frictional expression. For a smooth tendon profile, the approximation is within adequate accuracy. This is not true for abrupt changes in the tendon geometry namely abrupt changes in directions similar to the case of a harped tendon. To avoid such a poor modeling idealization, the user may consider refined discretization along such locations to capture the true response of the tendon.

4.6.3 Material Constitutive Relationships

Over a given period of time, the force in a tendon segment decreases from its original value due to the relaxation phenomena. The relaxation relationship has been discussed in the time dependent material property chapter. However, the relationship has been derived based on the primary assumption that the relationship predicts the variation in the segment force due to relaxation only. This is not true when an external disturbance is applied in which case the relationship is no longer applicable. To account for intermediate loadings, an intermediate fictitious initial segment force f_i **Fig. 4.6.3.1** at the end of loading stage number i is computed assuming that its relaxed value over the same period of time is identical to the segment force at the end of the same loading stage. The stress f_0 is the initial jacking stress at time t_0.

f_{sy} being the yield stress of prestressing steel, 0.55 * f_{sy} has been used as a threshold below which relaxation strains are neglected. If the initial stress value is less than the limit, the initial stress f_i is simply the stress at the current time step and no magnification is performed. Similarly, no reduction is applied to the current stress value to obtain the stress value at the next time step due to relaxation.

The above procedure has been suggested by Hernandez and Gamble [9] in order to incorporate force variations over time that are not due to relaxation based on the assumption that all non-relaxation changes in tendon force occur at the end of the time steps.

4.6.4 Stiffness and Load Computations

Tendon segment global stiffness and global nodal displacements to internal strains matrices are needed to form the contribution of the tendon element to the global structure stiffness matrix and the equivalent global load vector due to internal stresses and non-mechanical strains at each time step. These transformation matrices are computed once during the geometry configuration input phase and are subsequently used in all solution steps.

During the output phase, the segment end stress resultants are computed based on static equilibrium relative to the reference axis segment end nodal points I and J. The same assumption holds for the tendon formulation and the corresponding transformation matrices derivation mentioned above and described in the following sections.

(a) Segment Stiffness Matrix

The tendon segment global stiffness matrix [K] may be obtained through proper matrix operations involving the local segment stiffness matrix [k] and the global nodal displacements to internal strain transformation matrix. Since the mid length strain is assumed representative for a given tendon segment, the component of the mid length incompatible translational degree of freedom **Eq. 4.5.4.8** cancels out and the first component remains only. The latter component is the conventional displacement to strain relationship. The internal strain ϵ_x at a distance x along the local x- axis may be then related to the global nodal displacements GL12 through the following transformation matrix :

$$GTOE_p{}^T = \begin{bmatrix} -X/L \\ -Y/L \\ -Z/L \\ e_{I3}Y/L - e_{I2}Z/L \\ e_{I1}Z/L - e_{I3}X/L \\ e_{I2}X/L - e_{I1}Y/L \\ X/L \\ Y/L \\ Z/L \\ e_{J2}Z/L - e_{J3}Y/L \\ e_{J3}X/L - e_{J1}Z/L \\ e_{J1}Y/L - e_{J2}X/L \end{bmatrix} \qquad \text{(Eq. 4.6.4.1)}$$

where the segment point end excentricities e_{Ii} and e_{Ji} are as defined in **Eq. 4.6.1.2**. The global tendon segment stiffness matrix is consequently obtained from the following relationship :

$$[K_p] = GTOE_p{}^T * [k] * GTOE_p \qquad \text{(Eq. 4.6.4.2)}$$

where [k] = EA/L is the one by one tendon segment local stiffness matrix relating segment local force F to local axial deformation δ **Fig. 4.6.2b.**

(b) Segment Internal Force due to Nodal Displacements

The segment internal force computation necessitates the evaluation of the segment internal strain increment for given global nodal displacement increments. A similar approach to the one adopted in the frame element formulation is used and the global nodal displacements **Fig. 4.6.2b** are consequently transformed into corresponding local nodal deformations first then the equivalent internal strain increment is evaluated through **Eq. 4.5.4.12** evaluated at mid length. Given the incremental strains and the material stress-strain relationship, the segment internal stress and therefore internal force increment may be evaluated. The internal incremental strain ϵ_p in terms of the global nodal displacements UG12 and the global displacements to strain transformation matrix $GTOL_p$ may be expressed in the following fashion :

$$\epsilon_p = GTOL_p * UG12 \qquad \text{(Eq. 4.6.4.3)}$$

Since the tendon local deformational diplacements to internal strain relationship **Eq. 4.5.4.12** is independent of the superelement parameters, no update was needed for the above equation within each iteration for a given time step.

(c) Equivalent Nodal Loads due to Initial Forces

The reverse approach described in the previous section is required to evaluate the equivalent nodal loads FG12 of size 12x1 due to tendon segment internal force F. The transformation matrix relating these two quantities may be obtained by virtual work and is found to be the transpose of the global nodal displacements to internal strain transformation matrix GTOL described previously.

$$FG12_p = GTOL_p{}^T * F \qquad \text{(Eq. 4.6.4.4)}$$

4.7 Travelling Formwork Formulation

Segmental operations necessitate the existence of the traveler element to idealize the construction operations and provide adequate stiffness and strength for freshly cast frame element members which would otherwise fail or undergo large deformations.

The traveler element is modeled with a collection of conventional modified Bernouilli-Euler beam elements **Fig. 4.4.3.1**. Linear elastic material relationship is assumed and the traveler element cross section consists of an unlimited number of rectangular shapes. The traveler element may be moved and remove to and from any location in the structure during the solution phase. Following every movement operation, the traveler element characteristic matrices are recomputed based on the new location and are used subsequently used in all solution steps preceding the next movement operation.

Traveler element formulation allows modeling arbitrary types of moving formwork and the corresponding internal load and stiffness computations are performed automatically.

4.7.1 Traveler Geometry

Traveler element **Fig. 4.4.3.1** consists of a linear collection of frame elements. Consequently, it is sufficient to reiterate the description used previously for frame elements geometry idealization **Fig. 4.5.1.1**. Each frame element belonging to a given traveler element is defined in the global (X-, Y-, Z-) space by the two nodal points I and J. The traveler element reference x- axis the direction of which is defined by the end nodal points I and J need not coincide with the cross sectional area centroidal axis. However, the end nodal points I and J should be identical to the corresponding frame superelement and consequently the traveler reference axis is identical to the superelement reference axis.

The reference axis is the x- axis of the local coordinate system (x-, y-, z-). The origin is located at node I while the y- and z- axes constitute the traveler element cross section principal directions **Fig. 4.7.1.1**. The traveler cross section consists of an unlimited number of rectangular shapes and is assumed constant along the reference axis. Following the traveler element input phase, the cross sectional properties needed in the solution analysis are internally and automatically generated.

Arbitrary traveler elements cross sectional shapes may be used even though they might not meet the restriction of rectangular shapes collection. In such a case, the input consists of the global geometrical properties of the cross section that would have been generated internally in the event the rectangular shapes option would have been used.

4.7.2 Material Constitutive Relationship

The traveler element is assumed stress free when moved to a new location within the structure.

Time dependent and temperature strains are neglected in the traveler element formulation. Linear elastic material relationship is assumed and consequently the total stress σ at any point within the cross section may be related to the corresponding total strain ϵ in terms of the modulus of elasticity E through the following expression :

$$\sigma = E * \epsilon \qquad \text{(Eq. 4.7.2.1)}$$

4.7.3 Stiffness and Load Computation

Traveler segment global stiffness and global nodal displacements to internal strains matrices are needed to form the contribution of the traveler element to the global structure stiffness matrix and the equivalent global load vector due to internal stresses at each time step. These transformation matrices are computed once during the geometry configuration input phase and are subsequently used in all solution steps.

(a) Element Stiffness Matrix

The traveler element stiffness matrix $[K_t]$ in the global coordinate system is evaluated by transforming the corresponding six by six local deformational stiffness matrix $[k_t]$ through the global nodal displacements to local nodal deformations relationship. The local deformational stiffness matrix given in **Eq. 4.5.4.6** is used. However, no distinction is made between the two components forming such relationship for the traveler element since no compatability in the displacement fields between the traveler and super elements spanning the same two end nodes I and J is considered. In other words, the traveler element is assumed to be the only element in his own superelement. The six by twelve global nodal displacements **Fig. 4.5.1.1c** to local nodal deformations **Fig. 4.5.2.1c** relationship is simply the GTOL transformation matrix given in **Eq. 4.5.4.10**. The transformation operation may be expressed in the following manner :

$$[K_t] = GTOL^T * [k_t] * GTOL \qquad \text{(Eq. 4.7.2.2)}$$

(b) Element Internal Forces due to Nodal Displacements

The traveler element internal actions computations is a two phase operation. First the global nodal displacements UG12 **Fig. 4.5.1.1c** are transformed to local nodal deformations UL6 **Fig. 4.5.2.1c** which are subsequently transformed into the sought internal forces SL6 (four bending moments, one axial force and one torsional moment) through the local element deformational stiffness $[k_t]$. The expression relating the local deformational actions to the global nodal displacements (three translations and three rotations) may be written in the following manner :

$$SL6_t = [k_t] * GTOL * UG12 \qquad \text{(Eq. 4.7.2.3)}$$

(c) **Equivalent Nodal Loads due to Initial Strains and Forces**

The transformation matrix relating the element internal forces $SL6_t$ to the equivalent nodal global actions $SG12_t$ **Fig. 4.5.1.1c** may be obtained from virtual work considerations. Again, such transformation matrix is found to be the transpose of the GTOL matrix described in **Eq. 4.5.4.10** and the relationship expressing the equivalent global nodal loads in terms of the local nodal deformational forces may be expressed in the following manner :

$$SG12_t = GTOL^T * SL6_t \qquad \text{(Eq. 4.7.2.4)}$$

4.8 Computer Program Organization

A short root module and approximately 260 fortran subroutines are in charge of the input and output functions, database management functions, and the numerical computations required for the time dependent solution. The structure and organization of the computer program **SPCF3D** developed for this study is similar to the one adopted in the implementation of **SFRAME** program [7]. Consequently, the different subroutines have been grouped into modules each having a dictinctive function in the execution. The current implementation is subdivided in a Root Module that controls the flow of execution in addition to the Concrete Parameters, Mesh Input, Change, Solve, Output and the Library modules.

Except for the Concrete Parameter and Mesh Input Modules, segemental erection operations necessitates repetitive calls to the other modules depending on the user request. When the solution phase is terminated, the in-core data base is saved to a file and the solution phase may be restarted accordingly by restoring the data base file.

The input required for any analysis consists of the geometry of the structure, the cross-sectional shapes and material types, the material properties of concrete, mild steel and prestressing steel, the time dependent properties of concrete, the environmental conditions, the loading configurations and the sequence of construction. The output consists of the internal actions of each frame element, traveler element, tendon element and superelement. In addition, the user may request the values of stress, strain and modulus of elasticity at each gauss point belonging to any given filament within any specified frame element. All of the output can be requested at any time step during the solution up to failure.

4.9 Comparison of Computer Program Capabilities

A detailed comparison of capabilities among some of the computer programs developed by different reseachers at the University of California, Berkeley is tabulated in **Fig. 4.9.1** thru **Fig. 4.9.6**. The capabilities of the computer program **SPCF3D** developed for this study as well as those of **SFRAME** [27], **SPCFRAME** [34], **PCF3D** [25] and **NAPBOX** [30] are listed and compared in the following categories : (1) Types of structures; (2) Types of analysis; (3) Construction operations; (4) Types of loading; (5) Material property models; (6) Concrete creep model; (7) Element formulation; (8) Equation solver; (9) Data input format; (10) Automatic generation; (11) Units; (12) Program organization; (13) Program execution mode; (14) Torque-twist relation; (15) Cross-section; and (16) Date of completion, reference report no. and author. **SFRAME** may be used for the linear time dependent analysis of segmentally erected planar prestressed concrete frames, **SPCFRAME** for the nonlinear geometric, material and time dependent segmental analysis of planar prestressed concrete frames, **PCF3D** for the nonlinear, time dependent analysis of three dimensional reinforced and prestressed concrete frames while **NAPBOX** may be used for the nonlinear material analysis of curved non-prismatic reinforced and prestressed concrete box girder bridges.

The main additional features **SPCF3D** possesses over the other computer programs listed are the segmental erection capability across the depth of the cross section, the three dimensional generation capabilities of the prestressing tendons and the arbitrary nonlinear material stress-strain relationships that may be used for concrete, mild steel and prestressing steel. Thus **SPCF3D** utilizes and extends many of the capabilities of previously developed computer programs at the University of California at Berkeley.

Item	Category	SFRAME	SPCFRAME
1	**Types of structures** Concrete Structures	* Plain reinforced and prestressed concrete 2D frames	* Plain,reinforced and prestressed concrete 2D frames
	prestressed concrete structures	* pretensioned frames * post-tensioned bonded and unbonded	* post-tensioned bonded
2	**Types of analysis** nonlinearity combinations considered	* **Material/Geometry** linear/linear linear/nonlinear Nonlinear/Linear Nonlinear/Nonlinear	**Material/Geometry** linear/linear
	Time dependent analysis	* Instantaneous analysis * Time step analysis	* Instantaneous analysis * Time step analysis
3	**Construction operation** Installation and removal	Frames, tendons and traveling formworks	Frames, tendons and traveling formworks
	Boundary Conditions	* Restraining and releasing	* Restraining and releasing
	Type of operation	Cantilever, span-by-span, incremental launching	Cantilever, span-by-span, incremental launching
4	**Types of loading**	* Joint and uniform loads * imposed displacements	* Joint and uniform loads * imposed displacements
	Internal loadings	* Creep, shrinkage	* Creep, shrinkage
	Environmental loading	* Temperature change	* Temperature change
	Cross-sectional variation of temperature and shrink and shrinkage	* Linear	* Arbitrary
5	**Material property model** Conc sig-eps model Stl. sig-eps model Pres sig-eps model	* Linear * Linear * Linear	* Linear, Parabolic-linear Parabolic-linear-linear * Linear, Bilinear * Linear, Bilinear
	Nonlinear effects Concrete Mild steel Pres steel	None None None	* Crack, yield, crush * Load reversal, tens stif * Yield, fail, ld reversal * Yield, fail, ld reversal
	Time dependent effects Concrete Pres stl	* Creep, shrinkage,aging * Relaxation	* Creep, shrinkage, aging * Relaxation

Table 4.9.1 COMPARISON OF PROGRAM CAPABILITIES

Item	SPCF3D	PCF3D	NAPBOX
1	* Plain, reinforced concrete structures	* Plain reinforced and prestressed concrete 2D frames	* Plain, reinforced and prestressed non-prism 3D box girder bridges
	* pretensioned frames * post-tensioned bonded	* pretensioned frames * post-tensioned bonded and unbonded	* post-tensioned bonded
2	*** Material/Geometry** Linear/Linear Nonlinear/Linear	*** Material/Geometry** Linear/Linear Linear/Nonlinear Nonlinear/Linear Nonlinear/Nonlinear	*** Material/Geometry** Linear/Linear Nonlinear/Linear
	* Instantaneous Analysis * Time step analysis	* Instantaneous analysis * Time step analysis	* Instantaneous analysis
3	* Frames, tendons and traveling formworks	* Installation only Frames, tendons	* Box girder frame elements, tendons
	* Restraining and releasing	* Restraining and releasing	* Restraining and releasing
	* Cantilever, span-by-span incremental launching and composite	* Whole structure built in one stage	* Whole structure built in one stage
4	* Joint & unif. loads * Imposed displacements	* Joint loads * Imposed displacements	* Joint and Z-uniform * Imposed displacements
	* Creep, shrinkage	* Creep, shrinkage	* None
	* Temperature change	* Temperature change	* None
	* Piecewise planar	* Planar	* None
5	* Piecewise linear (multi-linear) * Piecewise linear * Piecewise linear	* Parabolic-Linear * Linear, bilinear * Linear, multilinear	* Parabolic-Linear * Linear, bilinear * Linear, multilinear
	* Crack, yield, crush * Load reversal * Yield, fail, ld reversal * Yield, fail, ld reversal	* Crack, yield, crush * Load reversal * Yield, fail, ld reverse * Yield, fail, ld reverse	* Crack, yield, crush * ld reversal, tens stiffening * Yield, fail, ld reversal * Yield, fail, ld reversal
	* Creep, shrinkage, aging * Relaxation	* Creep, shrinkage, aging * Relaxation	None

Table 4.9.2 COMPARISON OF PROGRAM CAPABILITIES

Item	Category	SFRAME	SPCFRAME
6	**Concrete creep model** Analysis model	* **Stress/Mat Param** Constant/Constant Linear/Constant Linear/Linear	* **Stress/Mat Param** Constant/Constant Linear/Constant Linear/Linear
	Creep data	* Laboratory, ACI, CEB	* Laboratory, ACI, CEB
	Nonlinear effects	None	* Creep increase for high stress level
7	**Element Formulation** Cross-section idealization	* Concrete and smeared reinforced mild steel	* Concrete and smeared reinforced mild steel
	Tendon idealization	* Linear segments within frame elements	* Linear segments within frame elements
	Displacements functions	* Cubic deflection and linear curvature * Constant axial strain on reference axis	* Cubic deflection and linear curvature * Linear axial strain on reference axis
	Element degree of freedom	**End rotation/Axial Disp** 2/1	**End rotation/Axial Disp** 2/1
	Integration for stiffness and internal resisting ld Over cross-section Over element length	* Closed form * Closed form	* Layer integration * 3-point gauss quadrature
	State determination	* Closed form	* Path independent
	Nonlinear strategy Instantaneous analysis	* Load control with single load step	* Load control with single load step * Disp control with multiple load steps
	Time step analysis	* Load control with single load step and multiple time steps	* Load control with single load step and multiple time steps
	Iteration scheme	* Constant stiffness	* Constant stiffness * Variable stiffness
	Convergence criteria	* Unbalanced load norm * stress norm * stress difference	* Displacement ratio * Max unbalanced load * stress ratio
8	Equation solver	* COLSOL column profile solver	* COLSOL column profile solver
9	Data input format	* Free field format	* Free field format
	Execution sequence	* Controlled by input command lines	* Controlled by input command lines

Table 4.9.3 COMPARISON OF PROGRAM CAPABILITIES

Item	SPCF3D	PCF3D	NAPBOX
6	* **Stress/Mat Param** Constant/Constant Linear/Constant Linear/Linear	* **Stress/Mat Param** Constant/Constant Linear/Constant Linear/Linear	None
	* Laboratory, ACI, CEB	* Laboratory, ACI, CEB	None
	Creep increase for high stress level	Creep increase for high stress level	None
7	* Discrete concrete and steel filaments	* Discrete concrete and steel filaments	* Discrete concrete and steel filaments
	* Linear segments within frame elements	* Linear segments within frame elements	* Linear segments within frame elements
	* Cubic deflection and linear curvature * Linear axial strain on reference axis	* Cubic deflection and linear curvature * Linear axial strain on reference axis	* 3 node isoparametric box girder element including warping
	Rot./Axial/Twist/Warp 4/1/1	**Rot./Axial/Twist/Warp** 4/1/1	**Rot./Axial/Twist/Warp** 4/1/1/1/Distort : 1
	* Filament integration * 1-point gauss (stif) 3-point gauss (load)	* Filament integration * 1-point gauss (stif) 2-point gauss (load)	* Filament integration * 2-point gauss (stif) 2-point gauss (load)
	* Path independent	* Path independent	* Path independent
	* Load control with single load step	* Load control with multi load step	* Load control with multi load step * Disp control with multiple load steps
	* Load control with single load step and multiple time steps	* Load control with multi load step and multiple time steps	None
	* Constant stiffness * Variable stiffness	* Constant stiffness * Variable stiffness	* Constant stiffness * Variable stiffness
	* Displacement ratio * Max unbalanced load * Stress and stress ratio	* Displacement ratio * Max unbalanced load * Max displacements	* Max unbalanced load * Max displacements
8	* COLSOL column profile solver	* SYMSOL symmetric active solver	* SYMSOL symmetric active solver
9	* Free field format	* Free field format	* Fixed format
	* Controlled by input command lines	* Fixed	* Fixed

Table 4.9.4 COMPARISON OF PROGRAM CAPABILITIES

Item	Category	SFRAME	SPCFRAME
10	**Automatic data generation** Node, element & load	 * Generated	 * Generated
	Tendon geometry	* Parabolic tendons generated	* Parabolic tendons generated
11	Units	* lb-in	* lb-in
12	**Program organization** Modularity # of subprograms # of instruction lines	 * High * 123 * 7700	 * High * 130 * 9600
	Core storage allocation	* Dynamic	* Dynamic
	Data base manage management Core memory Direct access files Sequential files	 * Manages incore arrays * Frame, tendon * Camber, Incore arrays	 * Manages incore arrays * Frame, tendon * Camber, Incore arrays
13	Program execution mode	Batch, terminal, restart	Batch, terminal, restart
14	Torsion-twist relation	None	None
15	Cross-section distortion	None	None
16	**Date of completion** **Report No.**	May 1986 UCB/SESM-86/07 by M. Ketchum	February 1989 UCB/SEMM-89/07 by Y. Kang

Table 4.9.5 COMPARISON OF PROGRAM CAPABILITIES

Item	SPCF3D	PCF3D	NAPBOX
10	* Generated	None	* Generated
	* Parabolic tendons generated on 2-2D planes	None	* Generated
11	* lb-in	* Any consistent unit	* Any consistent units
12	* High * ~ 260 To be determined	* Low * ~ 8 To be determined	* Low To be determined To be determined
	* Dynamic	* Static	* Static
	* Manages incore arrays * Frame, tendon, traveler conc. & stl filaments * Camber, Incore arrays	None Incore arrays None	None Incore arrays None
13	Batch, terminal, restart	Batch, terminal	Batch, terminal
14	Linear	Bilinear-plastic	Bilinear-plastic
15	None	None	None
16	May 1990 UCB/SEMM-90/03 by F. Kasti	June 1984 UCB/SESM-84/12 by A. Mari	December 1986 UCB/SEMM-86/13 by D. Choudhury

Table 4.9.6 COMPARISON OF PROGRAM

CAPABILITIES

CHAPTER 5 : NUMERICAL EXAMPLES

Introduction

The computer program **SPCF3D** developed for this study has been used to analyse several numerical examples discussed in the current chapter. The results obtained with **SPCF3D** are compared with analyses performed by other researchers for the same examples in order to check the formulation, show the accuracy of the results, underline the limitations of the current practice, study the effects of time dependent effects and demonstrate the capability of the current implementation to analyse large and complex three dimensional structures built segmentally along the longitudinal length or across the depth of the cross section. These examples may be also used to verify the proper installation of the computer program on a new computer system.

Complete input data file listings of all the examples analysed in this report are provided in Appendix B. The first four examples are analyses using **SPCF3D** for numerical examples used by other researchers to illustrate the application of their analytical models and computer programs. Each of these four examples is a special case of the more general problem that can be analysed by **SPCF3D** and each is used to check the development of the analytical model developed herein and its implementation in the program **SPCF3D**. The fifth example illustrates an application of **SPCF3D** to the analysis of a segmentally erected composite reinforced and prestressed continuous beam structure, which cannot be analyzed by the other programs.

Example 5.1 a is three span straight prestressed concrete box girder bridge that has been analysed using the computer program **NAPBOX** developed by Choudhury [30]. The bridge is assumed built nonsegmentally in one stage and is analysed for a 28 days concrete age. A P13 truck loading is applied incrementally and the nonlinear material response of the structure is traced through the maximum overload capacity. No time dependent effects are included. The results obtained using **SPCF3D** are compared with those found with **NAPBOX**.

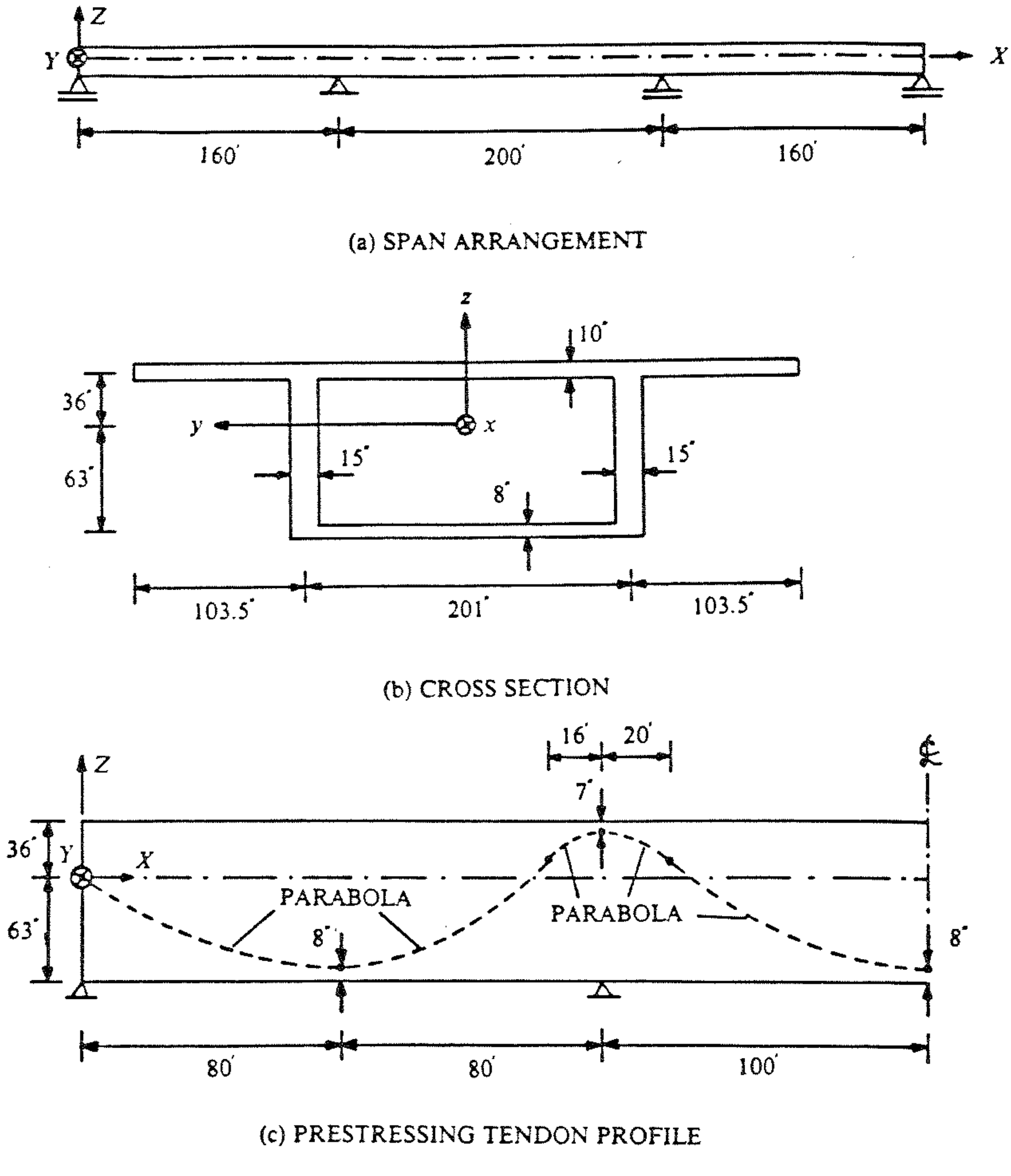


Fig. 5.1.1.1 EXAMPLE 5.1 - THREE-SPAN STRAIGHT PRESTRESSED CONCRETE BRIDGE
Choudhury [30]

Example 5.2 is a rerun of Example 5.1 with time dependent effects due to creep and shrinkage of concrete and relaxation of prestressing steel included. The results obtained using **SPCF3D** for this example are compared with similar results obtained by Kang [34] using his computer program **SPCFRAME**.

Example 5.3 is a three span curved prestressed concrete box girder bridge which is identical in shape, properties and loadings to the previous two examples except for the circular curvature in the third dimension. Nonlinear material only is accounted for in the analysis. Once more, the results are compared to Choudhury's values [30] obtained using **NAPBOX**..

Example 5.4 is a three span, straight, segmentally erected, prestressed concrete box girder bridge analysed including time dependent effects of creep, shrinkage and relaxation. The bridge was designed and analysed by Ketchum [27] using his computer program **SFRAME**. Results obtained using **SPCF3D** are compared with those found by **SFRAME**.

Example 5.5 is a two span simply supported girders made continuous with a top deck slab. The difference in the concrete ages of the girders and the in-situ deck due to transverse segmental erection, as well as the time dependent effects of creep, shrinkage and relaxation are included in the analysis.

5.1 Example 5.1-Three-Span Continuous Prestressed Concrete Straight Bridge

This example has been previously investigated by Choudhury [30] and Kang [34] as part of the on going research undertaken at the University of California, Berkeley to improve the analytical techniques for analysing box girder and segmentally erected structures. The objective of the current example is to demonstrate the capabilities of the computer program **SPCF3D** developed for the present study in tracing the response of a three span, continuous prestressed concrete bridge through its linear, inelastic and ultimate load capacity ranges. The intention is to check also the accuracy of the results compared to the results obtained by Choudhury (**NAPBOX**) and Kang (**SPCFRAME**) mentioned previously.

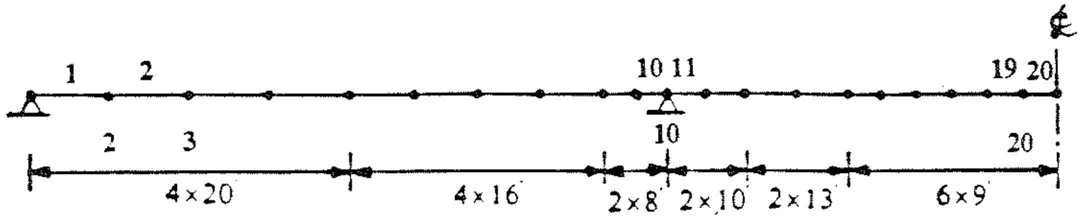
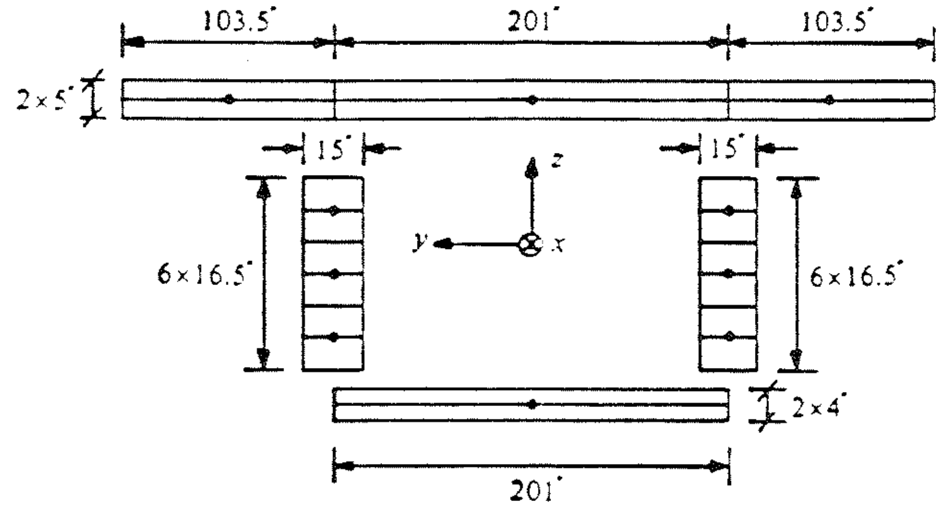
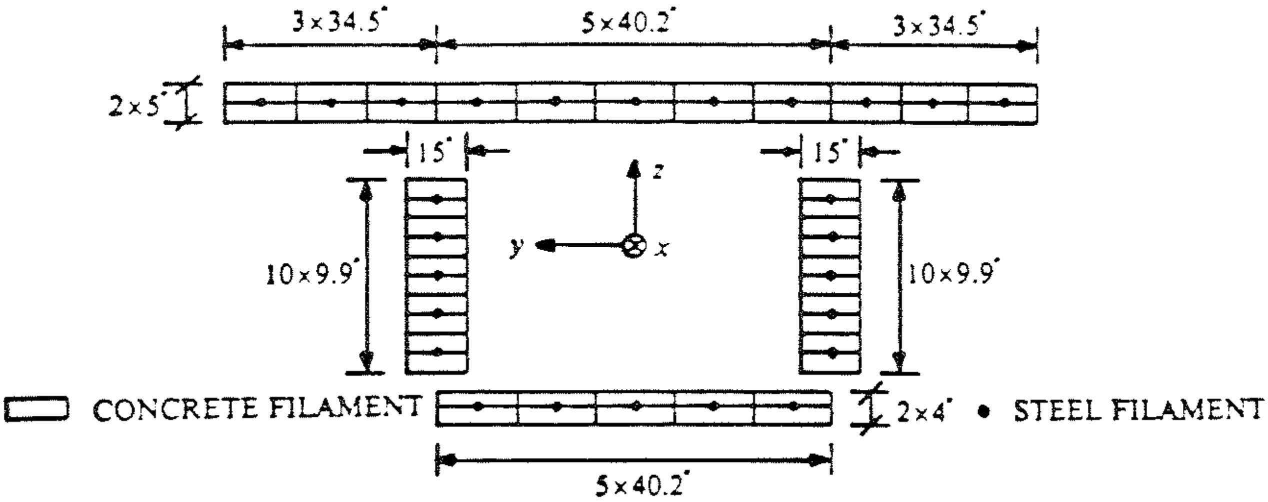


Fig. 5.1.1.1d EXAMPLE 5.1 - ELEMENT SUBDIVISION



(a) EXAMPLE 5.1



(b) EXAMPLE 5.2

Fig. 5.1.1.2 EXAMPLE 5.1 & 5.2 - DISCRETIZATION OF CROSS-SECTION INTO CONCRETE AND LONGITUDINAL STEEL FILAMENTS Choudhury [30]

5.1.1 Structural Modeling

The structural configuration of the straight three span continuous prestressed concrete box girder bridge is shown in **Fig. 5.1.1.1**. Due to structural symmetry of frame element geometries, material properties, support boundary conditions, mild and prestressing steel distributions about the mid length of the center span, half of the actual structural length only need to be considered in the analysis **Fig. 5.1.1.1c** and the corresponding structural modeling is given in **Fig. 5.1.1.1d**. A total of twenty longitudinal frame elements was used in the model. Boundary conditions are imposed at the bridge centerline. The structure is fully restrained in the vertical translational direction at all supports. The midspan node is restrained in the horizontal and rotational directions.

The two lane bridge cross section depicted in **Fig. 5.1.1.1b** is assumed constant over the whole length. Standard design practice might utilize a variable depth depth but this is not considered in the example. The corresponding cross section discretization is given in **Fig. 5.1.1.2b**. No transverse fiber discretization was needed since the structure and its response is two dimensional.

A summary of material and other properties for concrete, prestressing and mild steel is given in **Fig. 5.1.1.5**.

Post-tensioned prestressing tendons are used and the stressing operation is performed symmmetrically in both vertical webs and over both **Fig. 5.1.1.1c** longitudinal halves of the bridge. The tendon is grouted after prestressing and dead load is applied forcing full compatability in the displacments between the tendon and the corresponding frame superelement. The jacking force required in the stressing operation for each web is 2660 kips.

A total of 172, 0.5 in. diameter, 7-wire strand, 270 ksi prestressing tendons (total area 26.32 $in.^2$) divided symmetrically between the two webs, with the material properties given in **Fig. 5.1.1.5**.

0.3% mild steel reinforcement in the longitudinal direction is provided. The discretization and distribution of mild steel is given in **Fig. 5.1.1.2b**.

Transverse reinforcement in the form of stirrups in the webs and flanges of the box girder cross section has been provided. These consist of 0.155 $in.^2$/in. area/length in each web and 0.062 $in.^2$/in. area/length in each flange.

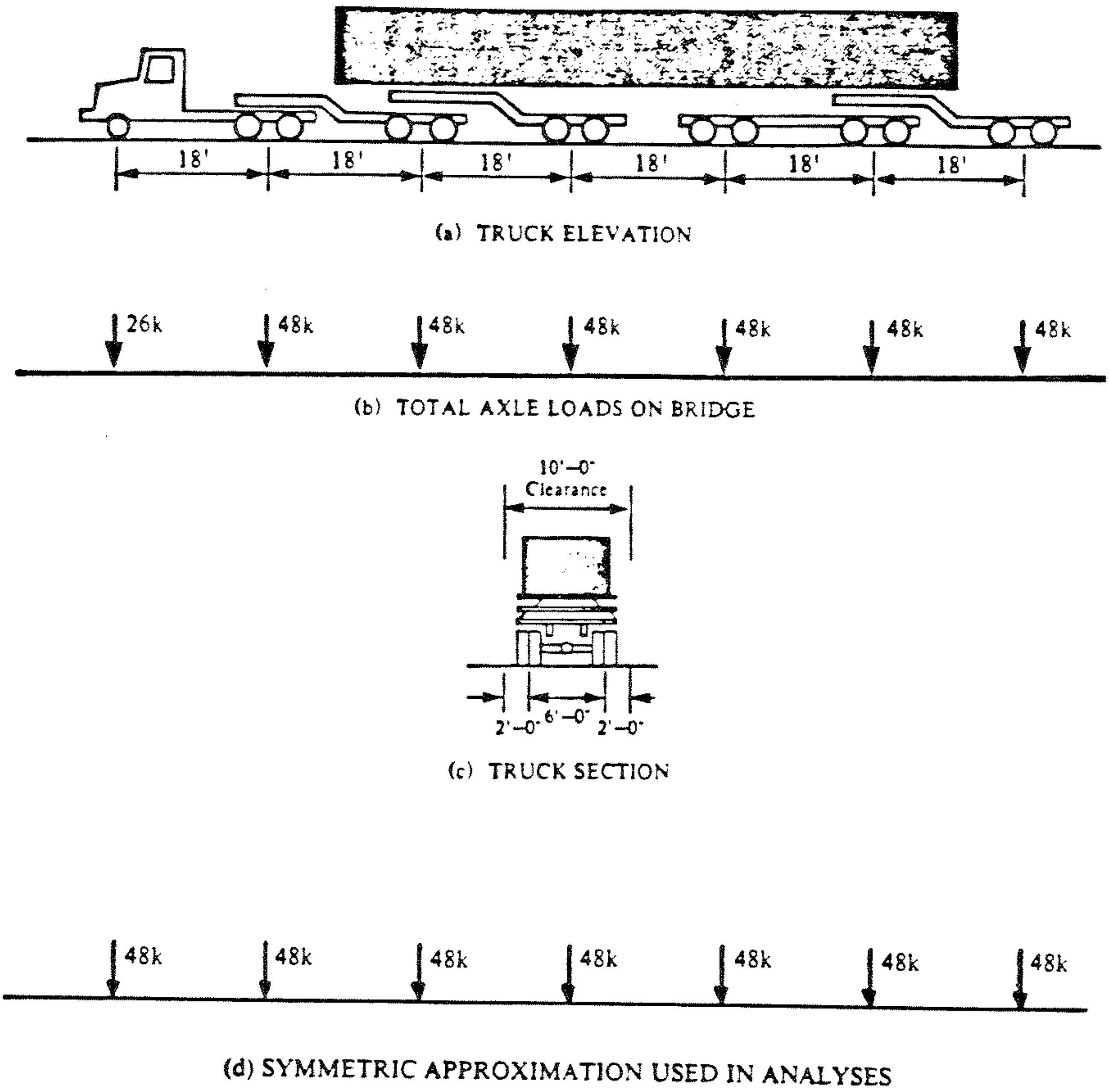
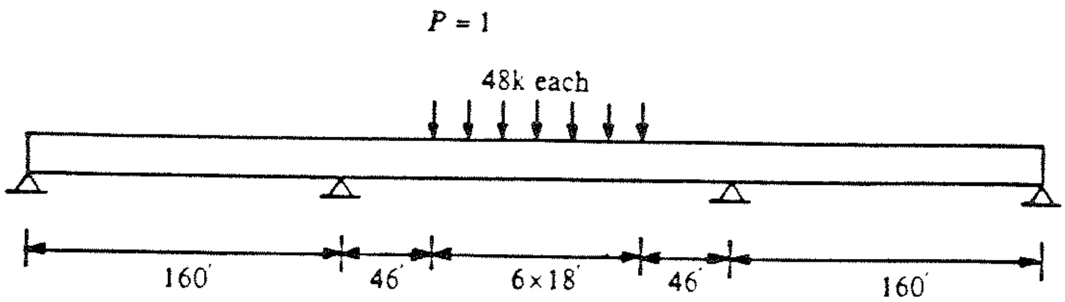
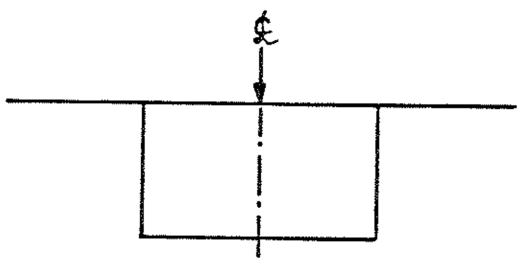


Fig. 5.1.1.3 P13 TRUCK LOADING Choudhury [30]



(a) LONGITUDINAL POSITION OF TRUCK LOAD



(b) TRANSVERSE POSITION OF TRUCK LOAD

Fig. 5.1.1.4 EXAMPLE 5.1 - POSITION OF TRUCK LOAD
Choudhury [30]

Concrete :

$f_c' = 4000$ PSI

$f_t = 500$ PSI

$\epsilon_{cu} = 0.004$

LONGITUDINAL REINFORCING STEEL : (ELASTIC PERFECTLY PLASTIC)

$f_y = 60$ KSI

$E = 29000$ KSI

$\epsilon_{su} = 0.003$

PRESTRESSING STEEL :

$\epsilon_1 = 0.00715$	$\sigma_1 = 196.6$ KSI
$\epsilon_2 = 0.00900$	$\sigma_2 = 220.0$ KSI
$\epsilon_3 = 0.01150$	$\sigma_3 = 240.0$ KSI
$\epsilon_4 = 0.01350$	$\sigma_4 = 245.0$ KSI
$\epsilon_5 = 0.05800$	$\sigma_5 = 270.0$ KSI

ANCHORAGE SLIP AT EACH JACKING END = 0.25 IN.

WOBBLE FRICTION COEFFICIENT = 0.0002/FT.

CURVATURE FRICTION COEFFICIENT = 0.25/RADIAN

UNIT WEIGHT OF COMPOSITE STRUCTURE = 155 PCF

Fig. 5.1.1.5 EXAMPLES 5.1 & 5.2 MATERIAL PROPERTIES
Choudhury [30]

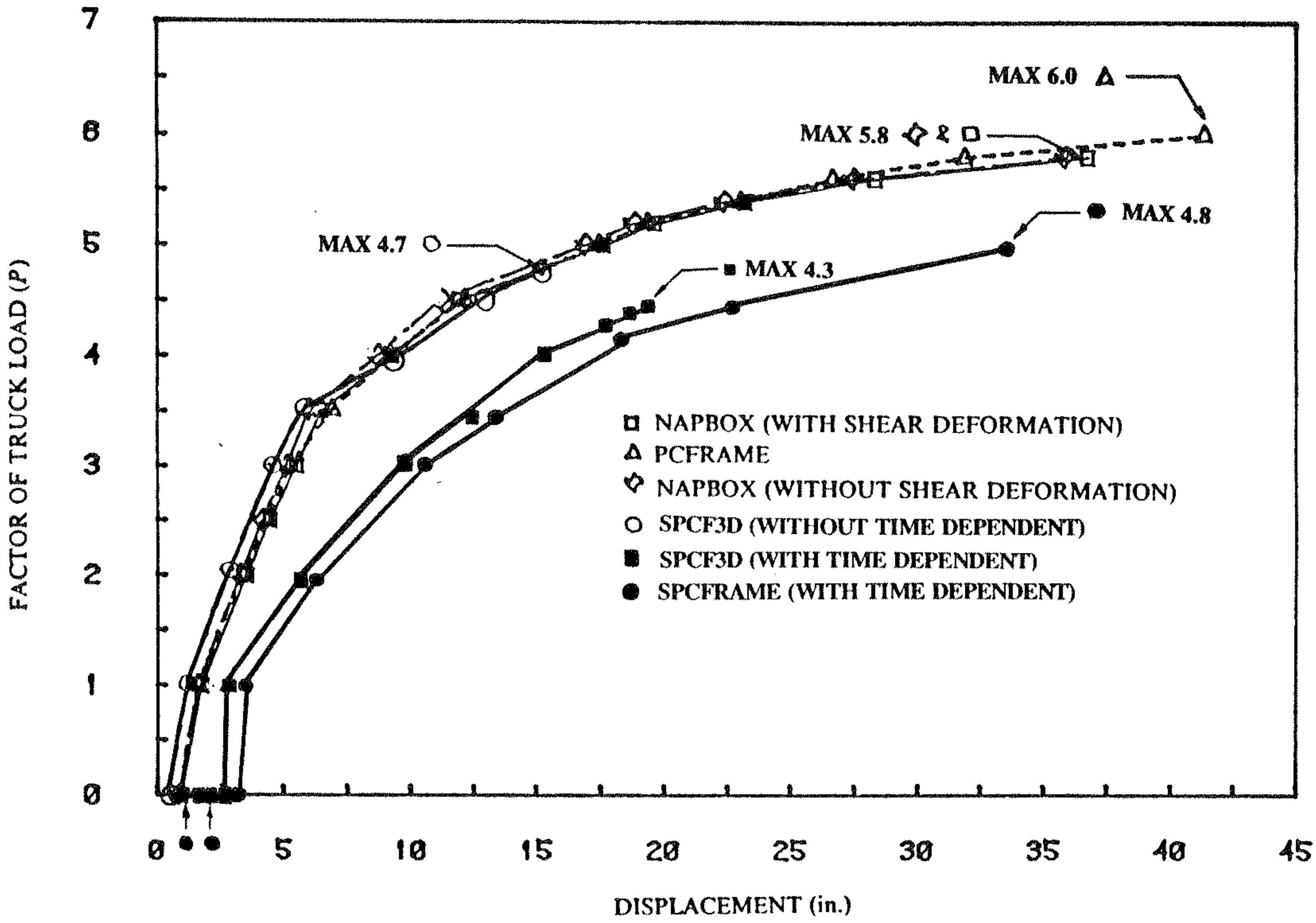


Fig. 5.1.2.1 EXAMPLES 5.1 & 5.2 - LOAD vs. VERTICAL DISPLACEMENT AT MIDDLE OF CENTER SPAN Choudhury [30]

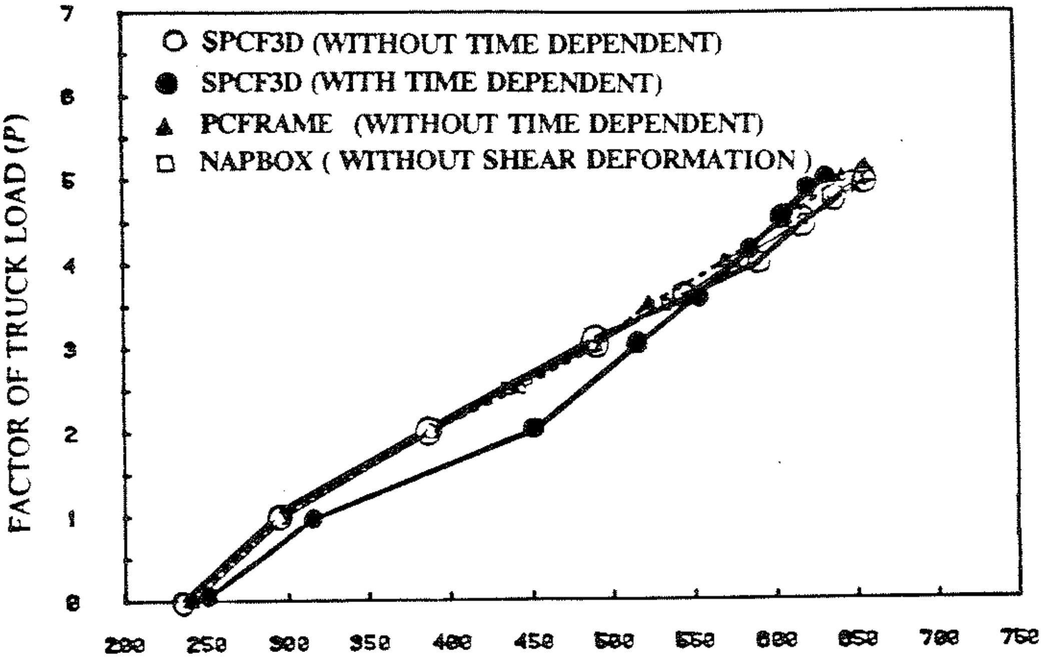
State of California standard practice has been applied in the structural design of the present bridge. Two lanes of HS20 loading with the corresponding impact factors and the material allowable stresses were considered in the calculations of the required prestressing steel reinforcement. The three span continuous bridge is overloaded with the heaviest vehicle found on California highway system. The P13 truck **Fig. 5.1.1.3a** is modeled as a collection of concentrated loads at the wheel locations **Fig. 5.1.1.3b** and is positioned half way in the center span with a symmetric load distribution **Fig. 5.1.1.3c** to take advantage of structural symmetry. The truck loading is located over the bridge centerline **Fig. 5.1.1.4** and the concentrated load collection is incrementally scaled during the analysis to reach the ultimate load the structure can resist. Only bending deformations are considered.

5.1.2 Observations and Discussion

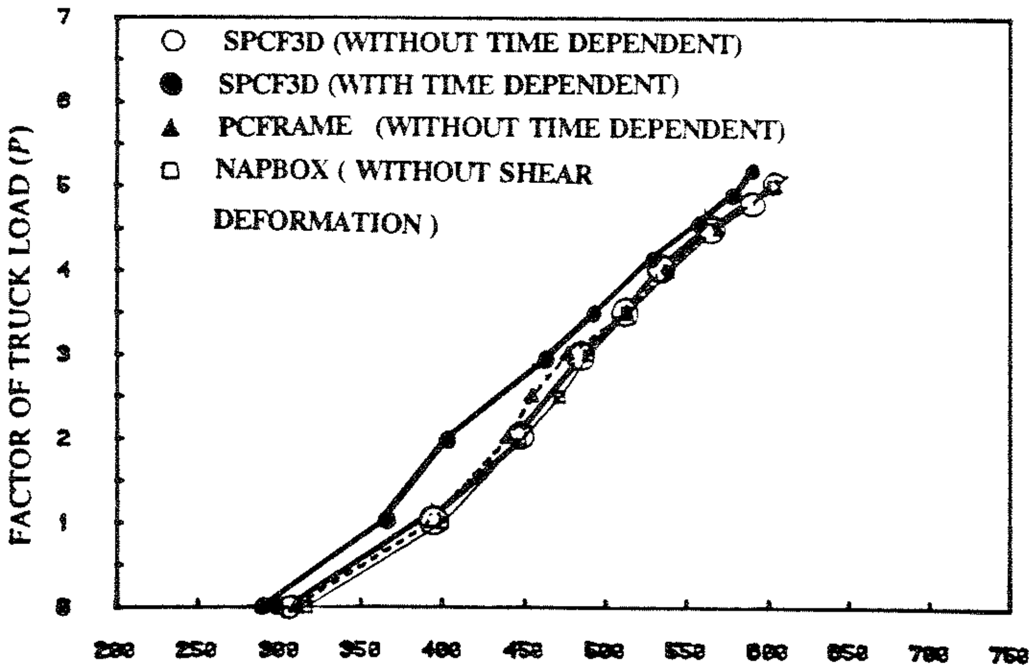
The results of the investigation obtained using the computer program **SPCF3D** developed for the present study have been compared to the results of similar analyses performed by Choudhury [30] and Kang [34]. The overall results show excellent agreement even for the case in which shear deformations have been considered. Such a behaviour indicates that the shear deformation contribution to the response of the structure, neglected in **SPCF3D** may be disregarded for similar situations.

The vertical displacement at mid length of the center span has been plotted in **Fig. 5.1.2.1** in terms of the incremental load vector applied to the center span equivalent to a P13 truck. The different plots representing different analyses using different analytical tools is a good check on the accuracy of the formulation and implementation of **SPCF3D**.

Another aspect that should be mentioned is that the maximum capacity of the bridge was found to be close to 5.0 times the P13 truck load used in **SPCF3D**. This figure was closer to 6.0 in the other investigations (such as **SPCFRAME**). The reason for such a discrepancy is that the higher figure was obtained with the computer program **SPCFRAME** which, contrary to **SPCF3D**, is equipped with a displacement control strategy allowing the incremental analysis to be undertaken at relatively small and even negative structural stiffness. The design of the bridge was based on servi-



(a) NEGATIVE MOMENT AT INTERIOR SUPPORT



(b) POSITIVE MOMENT AT MIDDLE OF CENTER SPAN

Fig. 5.1.2.2 EXAMPLES 5.1 & 5.2 - LOAD vs. INTERNAL MOMENTS AT INTERIOR SUPPORT AND MIDDLE OF CENTER SPAN Choudhury [30]

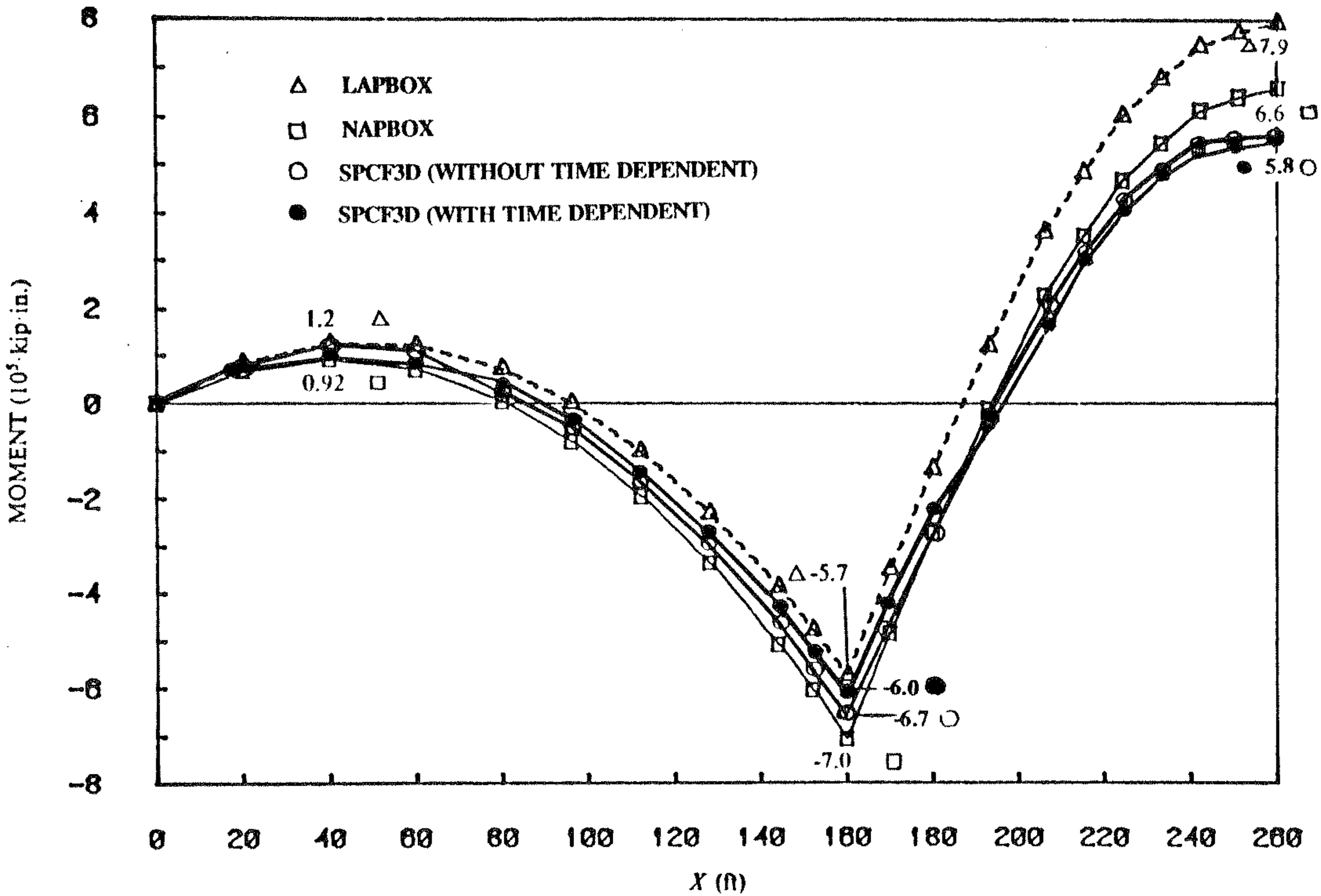


Fig. 5.1.2.3 EXAMPLES 5.1 & 5.2 - LONGITUDINAL DISTRIBUTIONS OF BENDING MOMENTS AT ULTIMATE LOAD Choudhury [30]

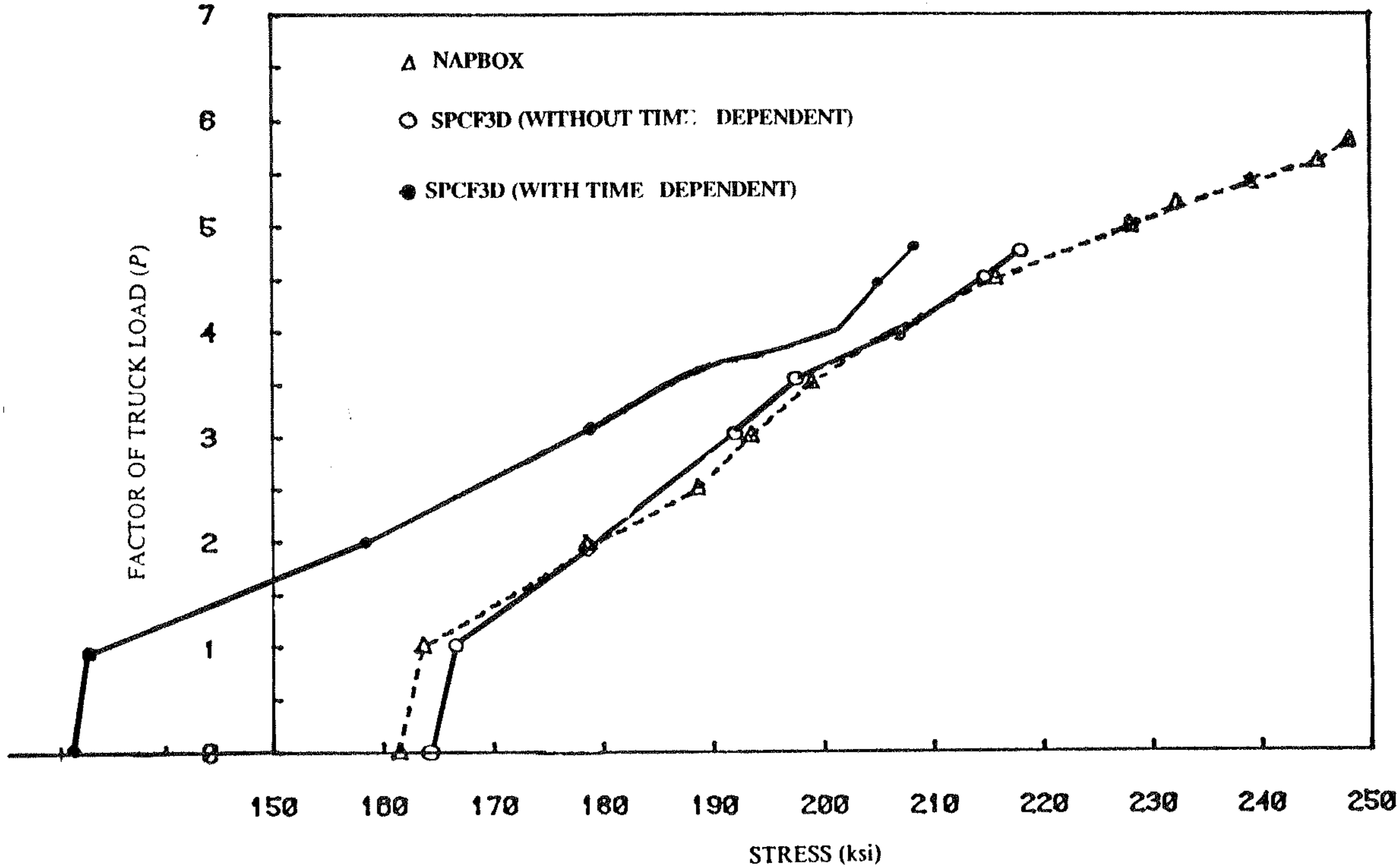


Fig. 5.1.2.4a EXAMPLE 5.1 - LOAD vs. PRESTRESSING STEEL SEGMENT STRESSES NEAR MIDDLE OF CENTER SPAN Choudhury [30]

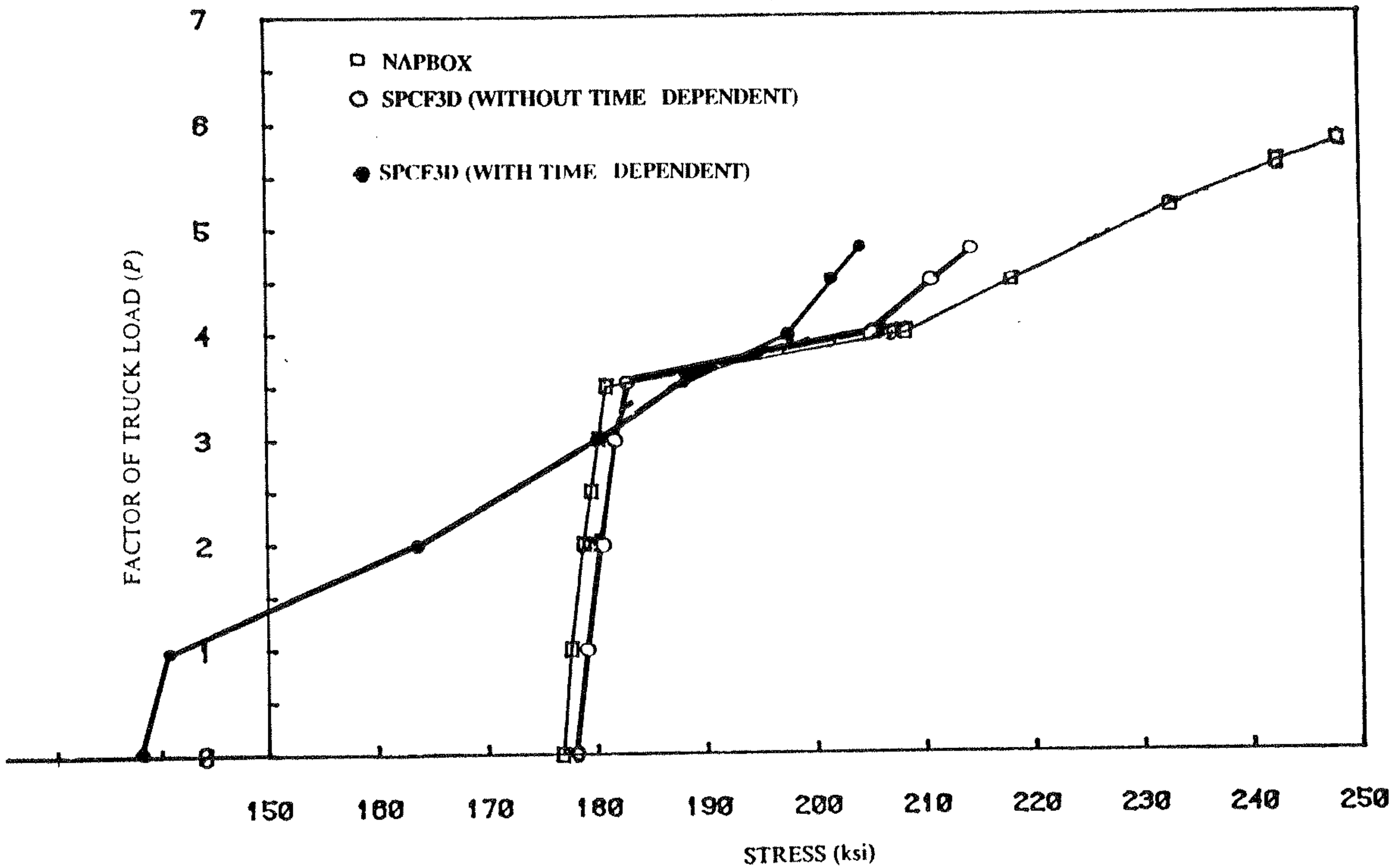


Fig. 5.1.2.4b EXAMPLE 5.1 - LOAD vs. PRESTRESSING STEEL SEGMENT STRESSES

NEAR INTERIOR SUPPORT Choudhury [30]

ceability criteria of a smaller load than the equivalent P13 truck vector. Even then, the ultimate load capacity was almost 5.0 to 6.0 times the truck equivalent load meaning that bridges designed based on the current State of California standard practice possess a significant overstrength capacity for vertical live load.

Center span mid length positive and interior span negative resisting moments have been plotted versus the incremental overload in **Fig. 5.1.2.2**. Shear deformations results obtained from Choudhury [30] have been omitted because they are quite similar to the results obtained without shear deformations. Good correlation is observed and the redistribution of the resisting internal bending moments due to cracking at the mid span location for a factor 1.0 overload is noted. The structure resistance at the interior support following such cracking is weakened and less stiffness and lower bending moment slope is observed beyond that point.

Redistribution due to nonlinear material behaviour may be more striking when comparing the bending moment profile obtained from the present study at the maximum load of dead load plus five times the P13 load and the bending moment obtained from linear and nonlinear ultimate load analyses using Chooudhury's [9] program LAPBOX and NAPBOX respectively **Fig. 5.1.2.3**. In either case, the P13 truck load is superposed to the existing structure dead load. Significant shift in the bending moment values from the center of the middle span which was reduced by 16.0% to the interior support which increased by 23.0% is noted. Such a reditribution in the internal resisting bending moment profile necessitates nonlinear ultimate load analyses of such classes of structure otherwise large deviations from the true response may result.

The prestressing steel internal forces at the same two locations are presented in **Fig. 5.1.2.4**. Smaller slope is observed at load levels ˜ 1.0 and 3.5 for middle of center span and interior support respectively. Such an increase in slope indicates that formation of crack is in progress at that particular location. The prestressing force in both segments reaches a common value at the maximum overload capacity.

5.2 Example 5.2-Time Dependent Three Span Continuous Prestressed Concrete Bridge

The analysis of the previous three span continuous prestressed continuous bridge has been repeated with time dependent effects due to creep and shrinkage of the concrete as well as relaxation of the prestressing steel has been included. The results are compared with the results obtained using the computer program **SPCFRAME** developed by Kang [34]. No comparison was made with Choudhury's [30] results since no time dependent effects could be accounted for in his study. The structural geometry and the material properties described in the previous example are still applicable. A creep coefficient of 2.35 and an ultimate shrinkage strain of 0.0008 were used for the current analysis. The bridge was assumed completely built in one stage at an age of 28 days similar to Example 5.1 and was subjected to its own dead load and prestressing forces at this time. This loading was left on the structure through 10000 days (about 27 years) at which time the bridge was incrementally overloaded with a P13 truck factor similar to the previous analysis until the maximum possible overload factor was reached.

The various quantities plotted in the figures shown in the previous section have been compared for this example with the results obtained using Kang's program **SPCFRAME**. Excellent correlation in the solutions is observed. For the sake of illustration, the variation of the vertical displacement at the mid length of the center span obtained using **SPCFRAME** has been plotted in **Fig. 5.1.2.1**. The curve obtained using **SPCF3D** from the present study is strikingly close to the one obtained by Kang. Comparison with the plot for no time dependent effects on the same graph, it may be concluded that the inclusion of the time dependent effects results in a less stiff structure that undergoes larger deformations under the same applied increment of load. The strength of the structure appears to be also affected and the maximum overload factor reached with time dependent analysis is reduced to a value in the vicinity of 4.3 for **SPCF3D** and 4.8 for **SFRAME**. Again, the reason for such a discrepancy is that the higher figure was obtained with the computer program **SPCFRAME** which, contrary to **SPCF3D**, is equipped with a displacement control strategy allowing the incremental analysis to be undertaken at relatively small and even negative structural stiffness.

The bending moments variation at the mid length of the center span and the interior support have been plotted in **Fig. 5.1.2.2** against the results obtained for Example 5.1 without time dependent effects. The variation of these internal resisting moments from the values obtained in Example 5.1 are within expectations. The equivalent nodal loads due to time dependent effects loads both side and half central spans downward leading to higher curvature and therefore a higher negative bending moment at the interior support. Consequently, a lower positive bending moment at mid central span is expected for static equilibrium requirements. Finally, the resisting bending moments at the previous two locations reaches asymptotically the same bending moment value as the one obtained for no time dependent analysis at maximum overload capacity. The same is true for prestressing segments forces at the same two locations shown in **Fig. 5.1.2.4a&b**.

5.3 Example 5.3-Three-Span Prestressed Concrete Continuous Curved Bridge

A three span prestressed concrete continuous curved bridge is considered next as part of a series of analytical analyses intended to demonstrate the ability of the computer program **SPCF3D** developed for this study and to check the accuracy of the results obtained. The results obtained by Choudhury [30] using **NAPBOX** will be used in comparing the results obtained with **SPCF3D**. A linear torsion-twist relationship is used in the present study compared to the trilinear model adopted by Choudhury [30] since the current implementation assumes the origin of the reference axis for each frame superelement to be the shear center. However, good correlation was still observed (for a factored load up to 3.5 between **SPCF3D** and **NAPBOX**). No comparison is made with the NAPBOX results of the cross sectional twist and transverse distortion as was reported by Choudhury. For the transverse distortion of the cross section, the current formulation of **SPCF3D** neglects any such deformation. Three load configurations have been considered in the investigation of the overload capacity and ultimate strength of the curved continuous bridge in the present study as well.

5.3.1 Structural Modeling

The geometry of the three span curved continuous bridge with respect to the (X-, Y-, Z-) global coordinate system is shown in **Fig. 5.3.1.1a**. Due to structural and loading symmetry, only half of the three span (160ft, 200ft, 160ft) structure measured along the centerline is considered in the

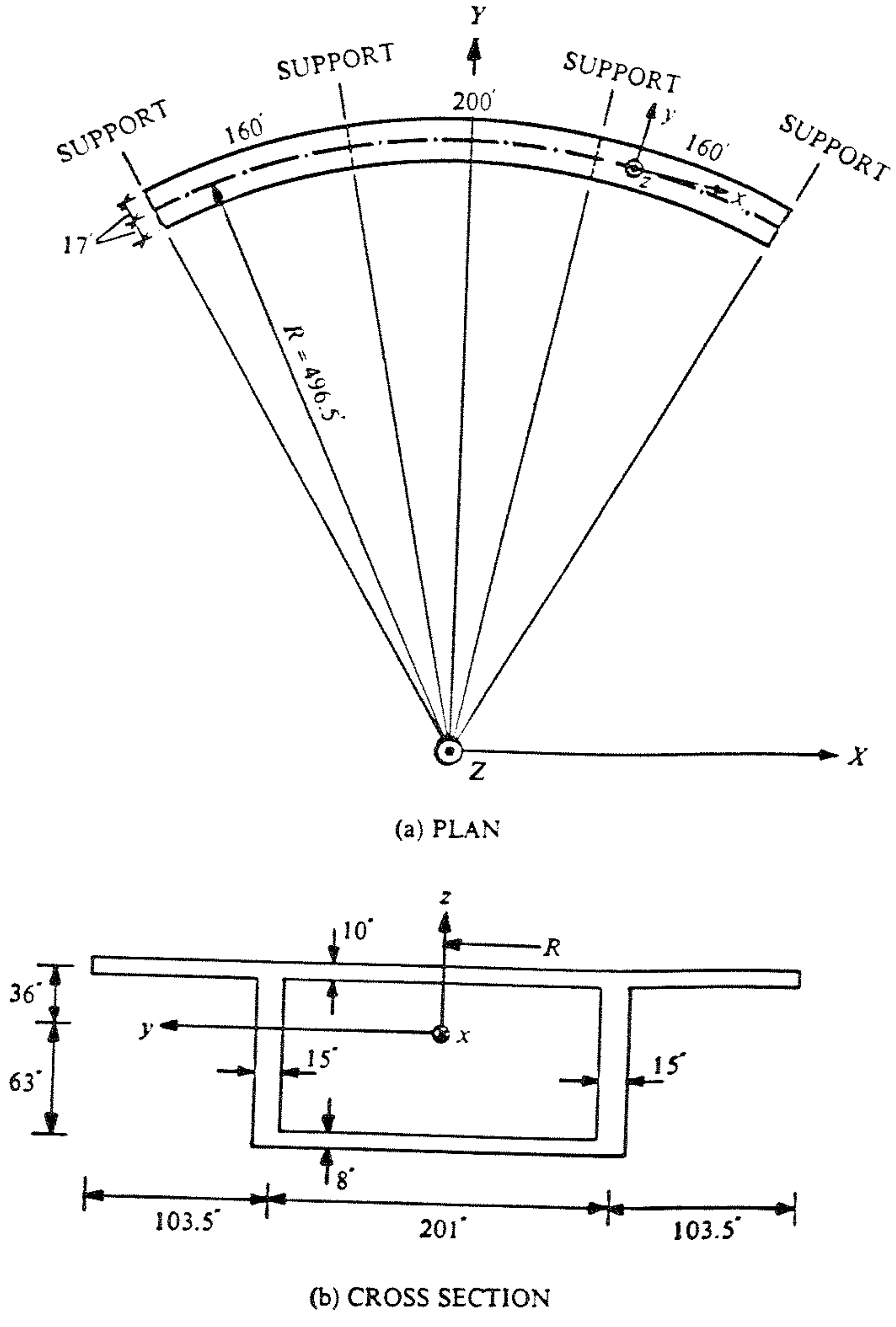
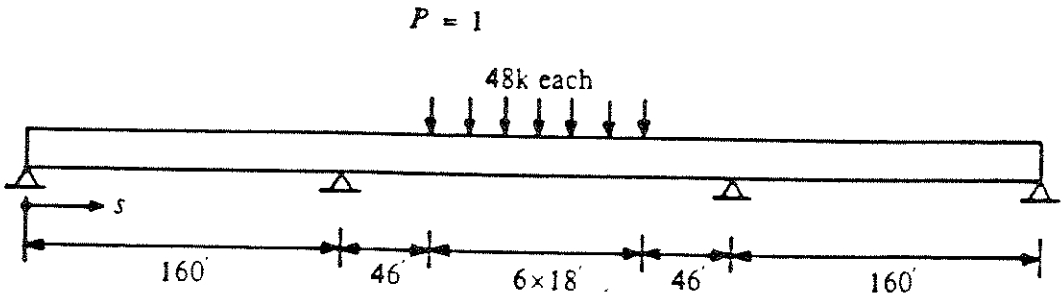
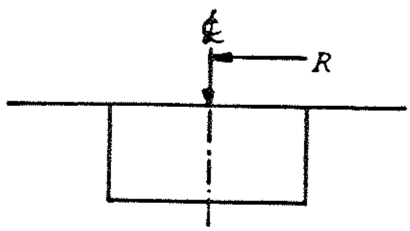


Fig. 5.3.1.1 EXAMPLE 5.3 - THREE-SPAN CURVED PRESTRESSED CONCRETE BRIDGE

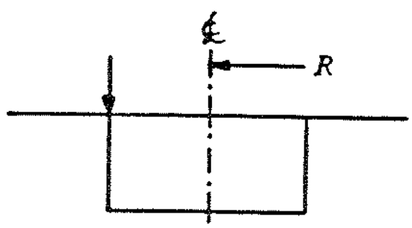
Choudhury [30]



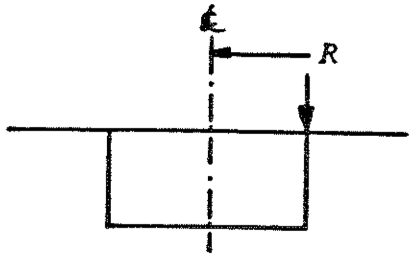
(a) LONGITUDINAL POSITION OF TRUCK LOAD FOR LOAD CASES 1, 2 AND 3



(b) TRANSVERSE POSITION OF TRUCK LOAD FOR LOAD CASE 1



(c) TRANSVERSE POSITION OF TRUCK LOAD FOR LOAD CASE 2



(d) TRANSVERSE POSITION OF TRUCK LOAD FOR LOAD CASE 3

Fig. 5.3.1.2 EXAMPLE 5.3 - POSITION OF TRUCK LOAD FOR DIFFERENT LOAD CASES

Choudhury [30]

analysis **Fig. 5.3.1.2**. Material and cross sectional shape used in the current example are identical to the ones adopted in the previous analysis. The cross sectional shape is shown again in **Fig. 5.3.1.1b** and the corresponding discretization in **Fig. 5.1.1.2b**. The discretization is carried out in the local y and z directions since three dimensional behaviour of the cross section is expected. The centerline of the bridge is circular in shape in the (X-, Y-) plane with a 496.5 ft radius. Choudhury has provided vertical, transverse and torsional rertraints at all supports. The current implementation of **SPCF3D** can not support such boundary conditions. However, a similar response has been obtained for different combinations (full fixity versus hinged condition) of support restraints considered.

The longitudinal frame element discretization is similar to the one adopted in the first two examples **Fig. 5.1.1.1** except for the fact that in the present example the elements are curved rather than straight and that each frame element in the previous two cases is further subdivided into two equal length elements. Thus a total of forty longitudinal frame elements is used in the analytical model.

Prestressing tendons are assumed identical in geometry and property to the tendons used in Examples 5.1 and 5.2. However, in the present study the tendons are curved (circular) in the (X-, Y-) plane measured along the webs. Consequently, parametric generation of tendon segments local geometry was not used and individual tendon points global coordinates input was adopted instead.

A P13 truck load vector was used in the present investigation **Fig. 5.3.1.2** . The equivalent load vector was located at mid span of the center span and incremented subsequently until the maximum overload capacity was reached. Three separate transverse P13 truck load positions have been analysed. The difference in the configuration was due to the load positioning relative to the curved bridge centerline. In load case number 1, the P13 truck load was positioned over the longitudinal centerline of the bridge meaning no externally applied torsional effects **Fig. 5.3.1.2b**. The second load case consisted in an excentrically loaded bridge, the P13 truck load being applied over the outer web **Fig. 5.3.1.2c**. The third load case is similar to the second case except for the fact that the equivalent truck load is positioned over the inner web **Fig. 5.3.1.2d**.

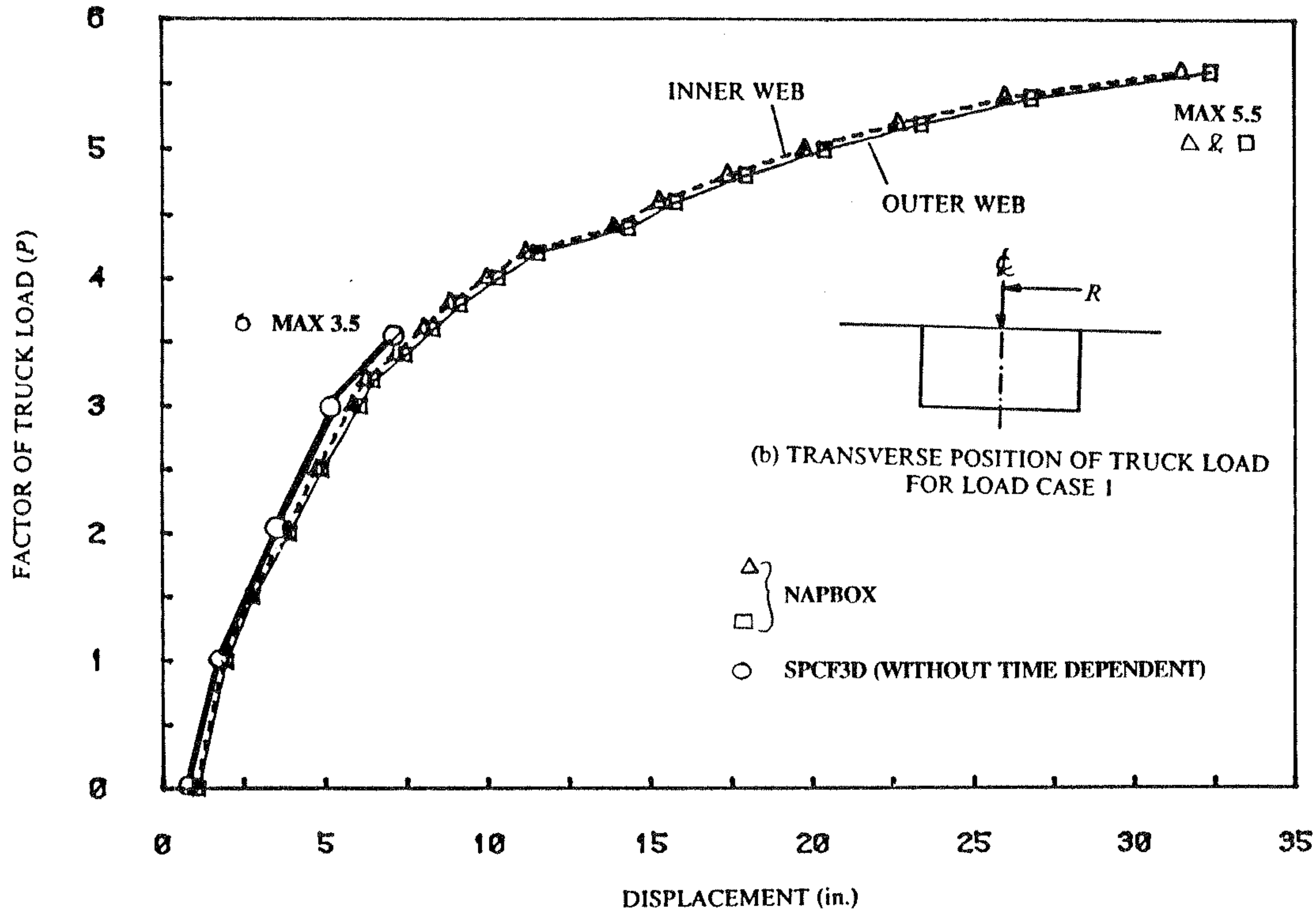


Fig. 5.3.2.1 EXAMPLE 5.3 - LOAD vs. VERTICAL WEB DISPLACEMENTS AT MIDDLE OF CENTER SPAN (LOAD CASE 1) Choudhury [30]

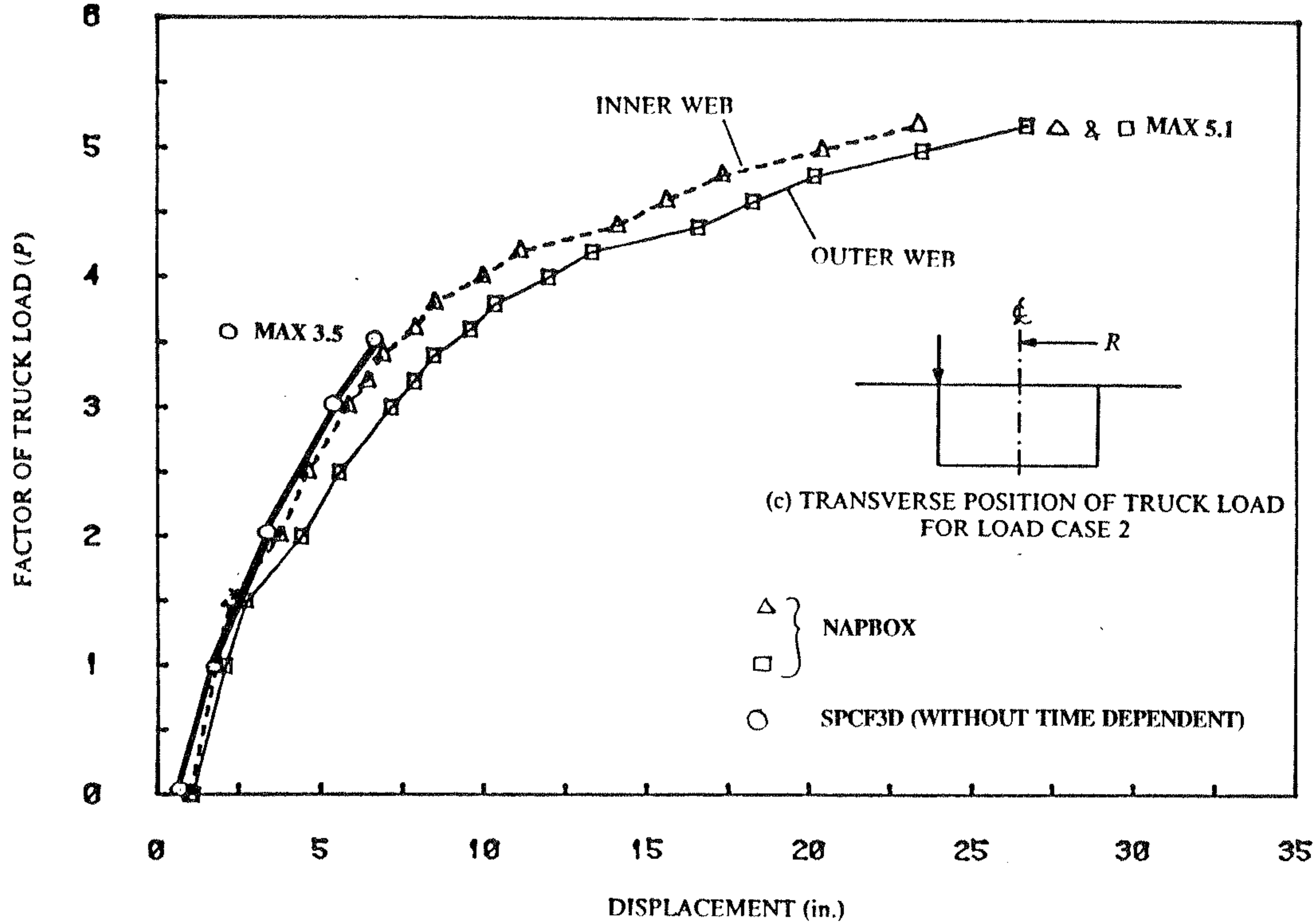


Fig. 5.3.2.2 EXAMPLE 5.3 - LOAD vs. VERTICAL WEB DISPLACEMENTS AT MIDDLE OF CENTER SPAN (LOAD CASE 2) Choudhury [30]

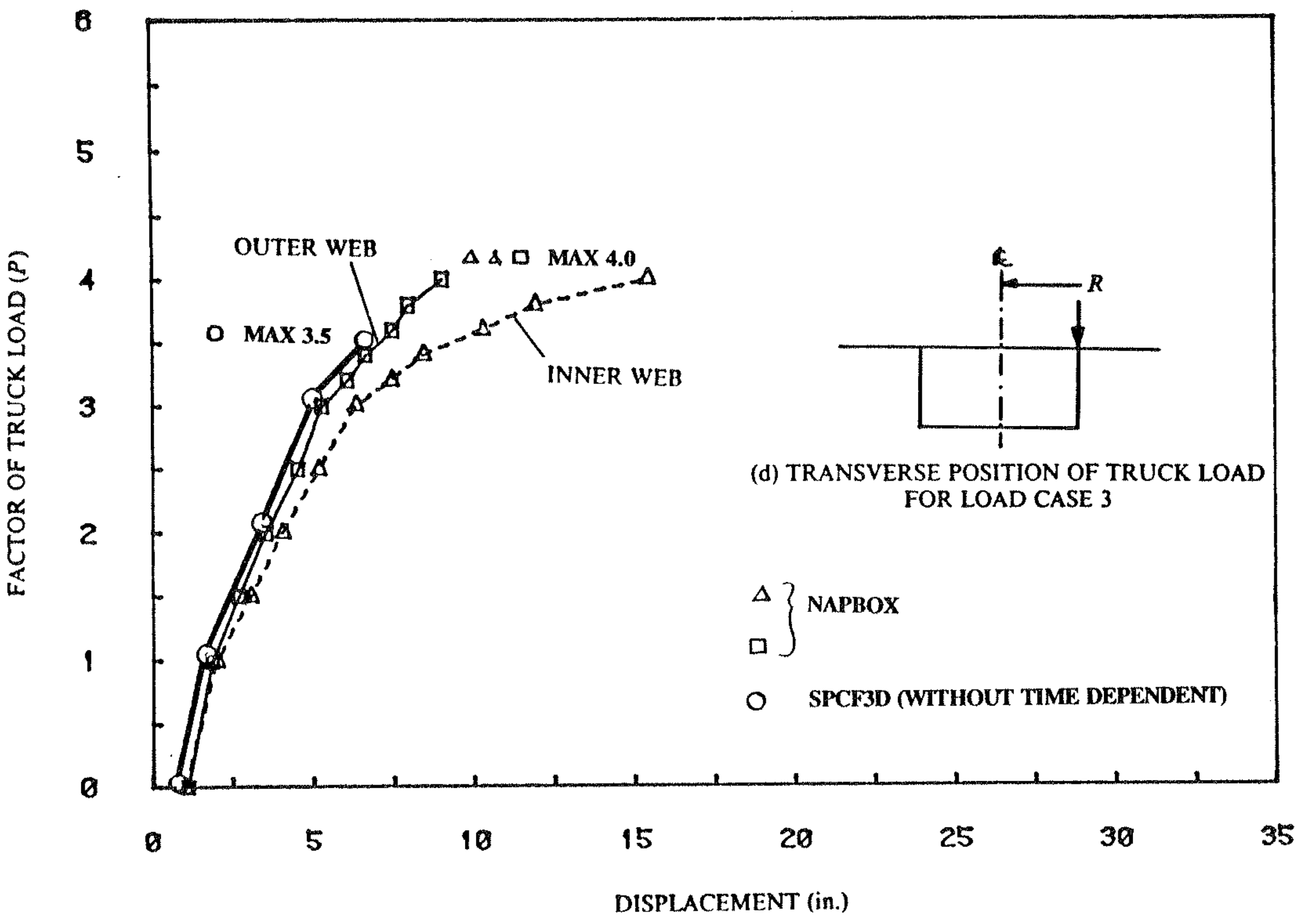


Fig. 5.3.2.3 EXAMPLE 5.3 - LOAD vs. VERTICAL WEB DISPLACEMENTS AT MIDDLE OF CENTER SPAN (LOAD CASE 3) Choudhury [30]

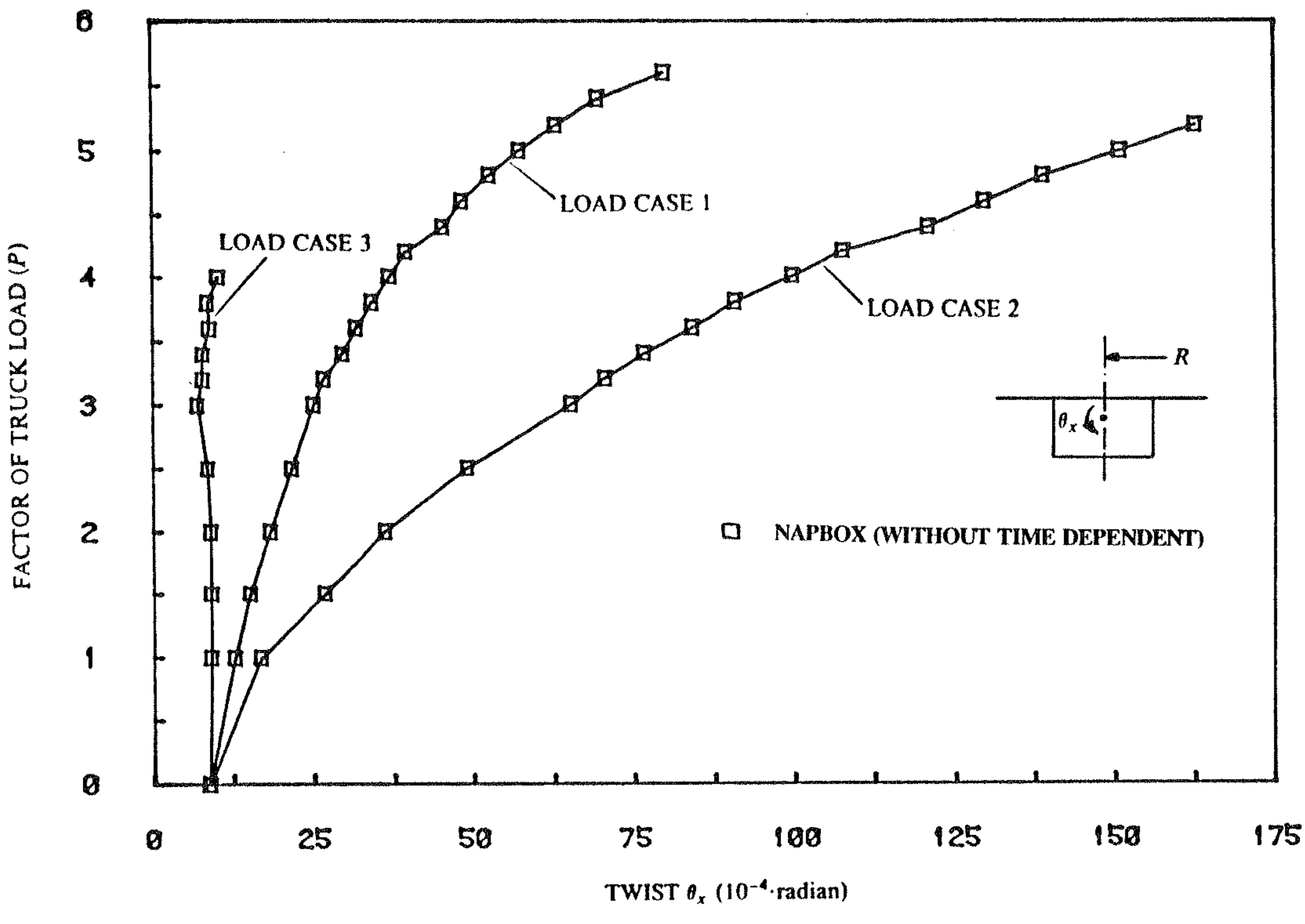


Fig. 5.3.2.4 EXAMPLE 5.3 - LOAD vs. CROSS-SECTION TWIST AT MIDDLE OF CENTER SPAN (LOAD CASES 1,2 & 3) Choudhury [30]

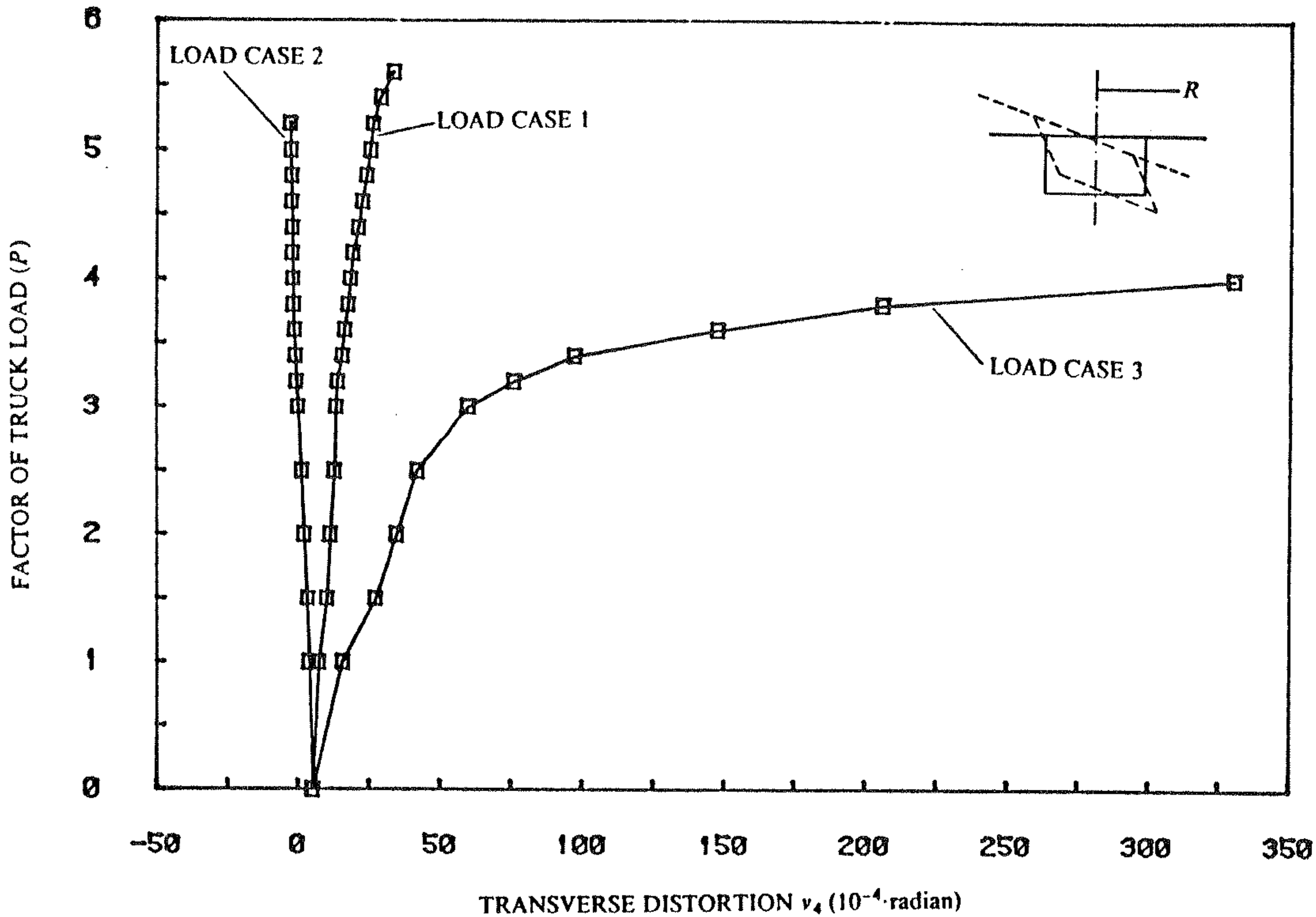


Fig. 5.3.2.5 EXAMPLE 5.3 - LOAD vs. TRANSVERSE DISTORTION OF CROSS SECTION AT MIDDLE OF CENTER SPAN (LOAD CASES 1,2 & 3) Choudhury [30]

5.3.2 Observations and Discussions

The vertical displacement at the mid span of the center span is plotted in **Fig. 5.3.2.1**, **Fig. 5.3.2.2** and **Fig. 5.3.2.3** for load cases 1 through 3 respectively. Up to a load factor of 3.5, good agreement between **SPCF3D** and **NAPBOX** is obtained, however, the maximum load and displacement reached by **SPCF3D** are well below similar results obtained by **NAPBOX**.

The maximum overload capacity reached by **SPCF3D** in all three load cases was close to 3.5 compared to a value of 4.5 obtained for the straight bridge previously. Choudhury reported values of 5.8, 5.2 and 4.0 for load cases 1 thru 3 respectively, thus large differences in the overload capacity exist between **SPCF3D** and **NAPBOX**. Many factors might have contributed to such a difference namely, the stress-strain relationship for torque-twist which is linear in **SPCF3D** compared to the tri-linear model used in **NAPBOX**, the fact that **SPCF3D** does not have a controlled displacement option that traces the response of the structure at low and negative global structure stiffness and finally the fact that transverse distortion and bending of the cross section is neglected in the current **SPCF3D** formulation. For the purpose of the following discussion, the variation of the angle of twist and the variation of the transverse distortion of the cross section at mid span of the center span as obtained by Choudhury with **NAPBOX** is plotted in **Fig. 5.3.2.4** and **Fig. 5.3.2.5** respectively.

The investigation carried out by Choudhury included the effects of transverse distortion and transverse flexural deformation of the cross section in the formulation. Based on **Fig. 5.3.2.1**, **Fig. 5.3.2.4** and **Fig. 5.3.2.5**, the conclusion was that the modes of failure in the three loading cases may be attributed to longitudinal flexural failure at the interior support, rapid deterioration in the torsional rigidy initiated by shear cracking at the interior support and rapid deterioration in the transverse flexural rigidity respectively.

Table 5.3 summarizes a comparison of cracking loads at the middle of center span and at the interior support between **SPCF3D** and **NAPBOX**.

In relation to prestressing steel, the prestressing force variation with truck load increments at the mid span of the center span and at the interior support have been plotted in **Fig 5.3.2.6a&b**, **Fig. 5.3.2.7a&b** and **Fig. 5.3.2.8a&b** for the three load cases respectively. The prestressing force

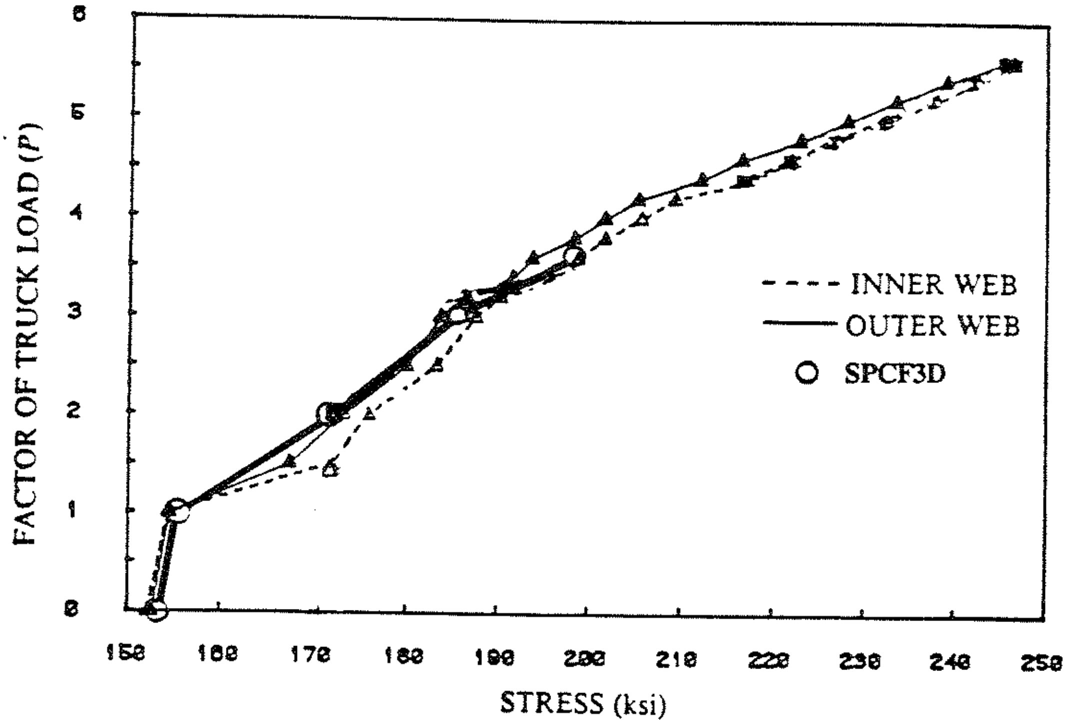


Fig. 5.3.2.6a EXAMPLE 5.3 - LOAD vs. PRESTRESSING NEAR MIDDLE OF CENTER SPAN (LOAD CASE 1)

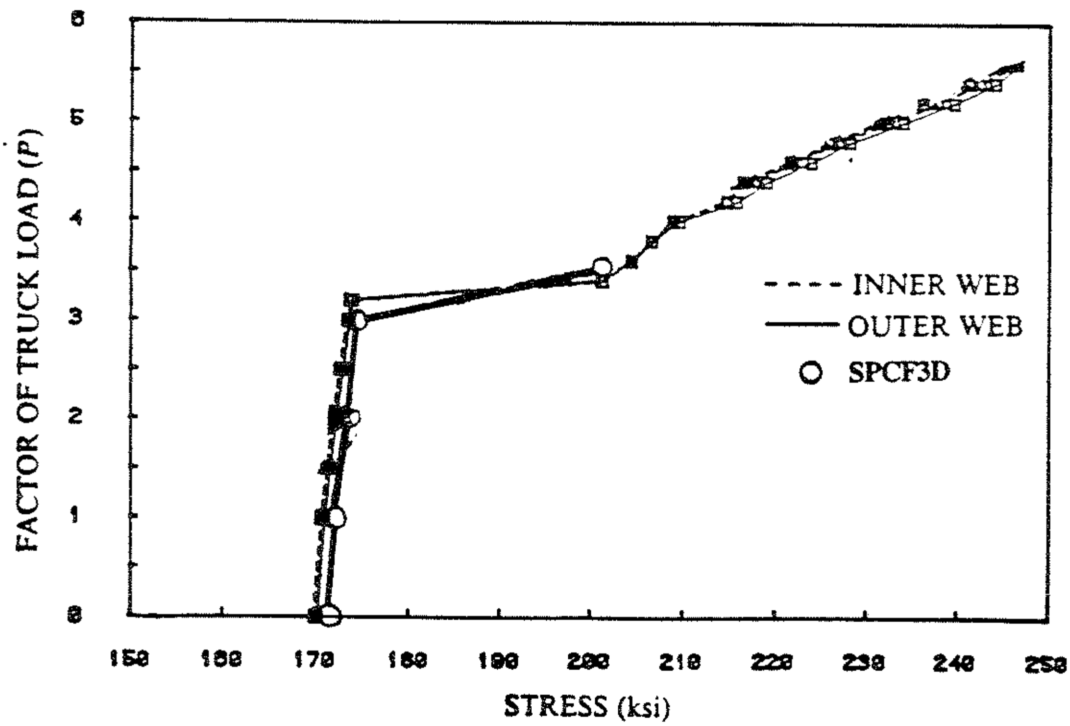


Fig. 5.3.2.6b EXAMPLE 5.3 - LOAD vs. PRESTRESSING NEAR INTERIOR SUPPORT SPAN (LOAD CASE 1) Choudhury [30]

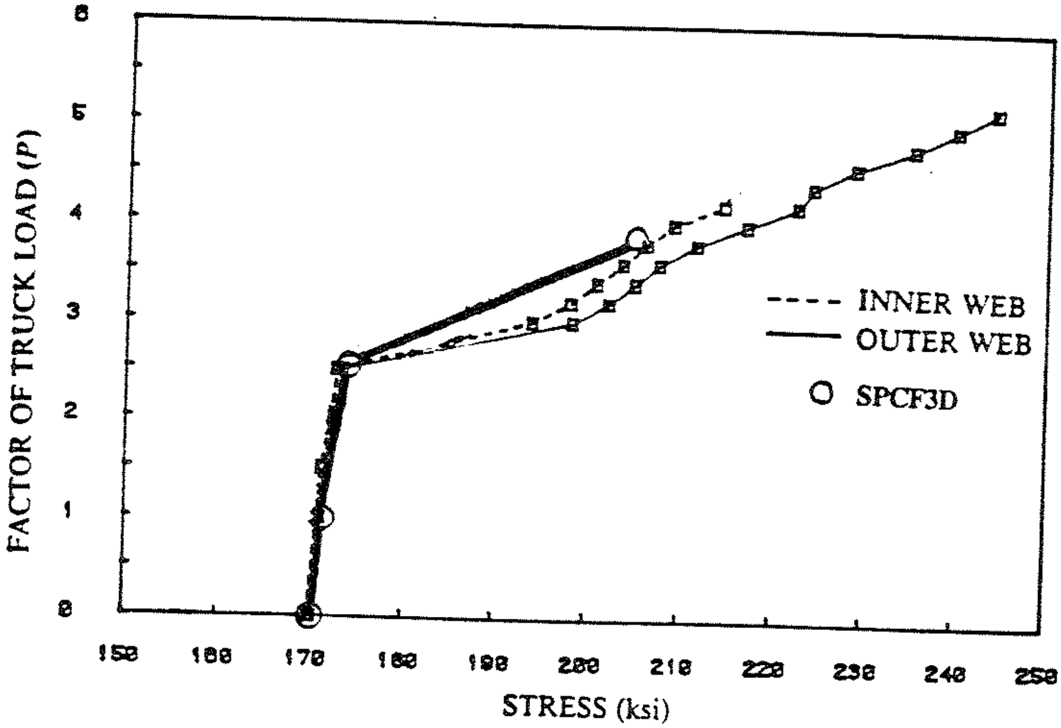


Fig. 5.3.2.7a EXAMPLE 5.3 - LOAD vs. PRESTRESSING NEAR MIDDLE OF CENTER SPAN (LOAD CASE 2) Choudhury [30]

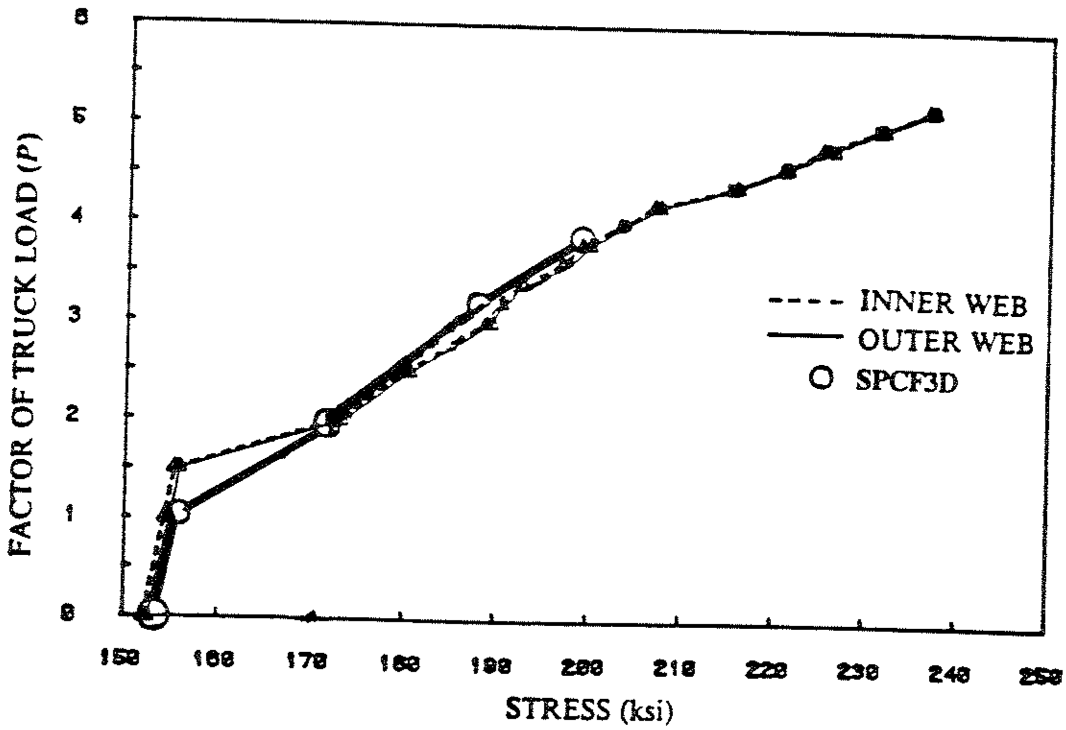


Fig. 5.3.2.7b EXAMPLE 5.3 - LOAD vs. PRESTRESSING NEAR INTERIOR SUPPORT SPAN (LOAD CASE 2) Choudhury [30]

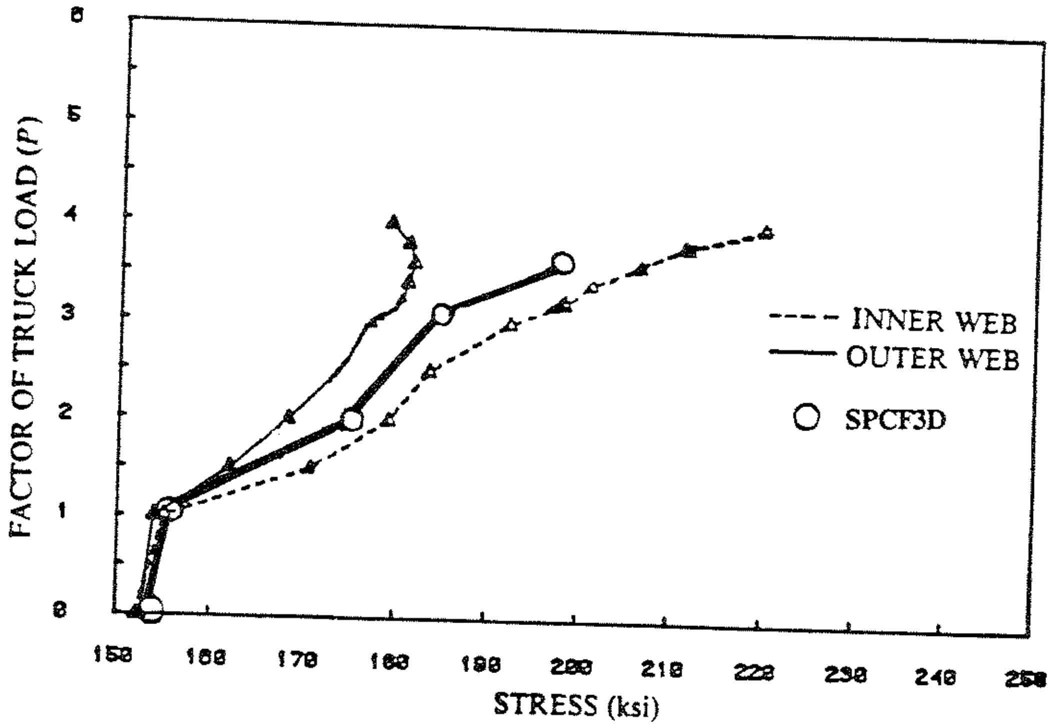


Fig. 5.3.2.8a EXAMPLE 5.3 - LOAD vs. PRESTRESSING NEAR MIDDLE OF CENTER SPAN (LOAD CASE 3) Choudhury [30]

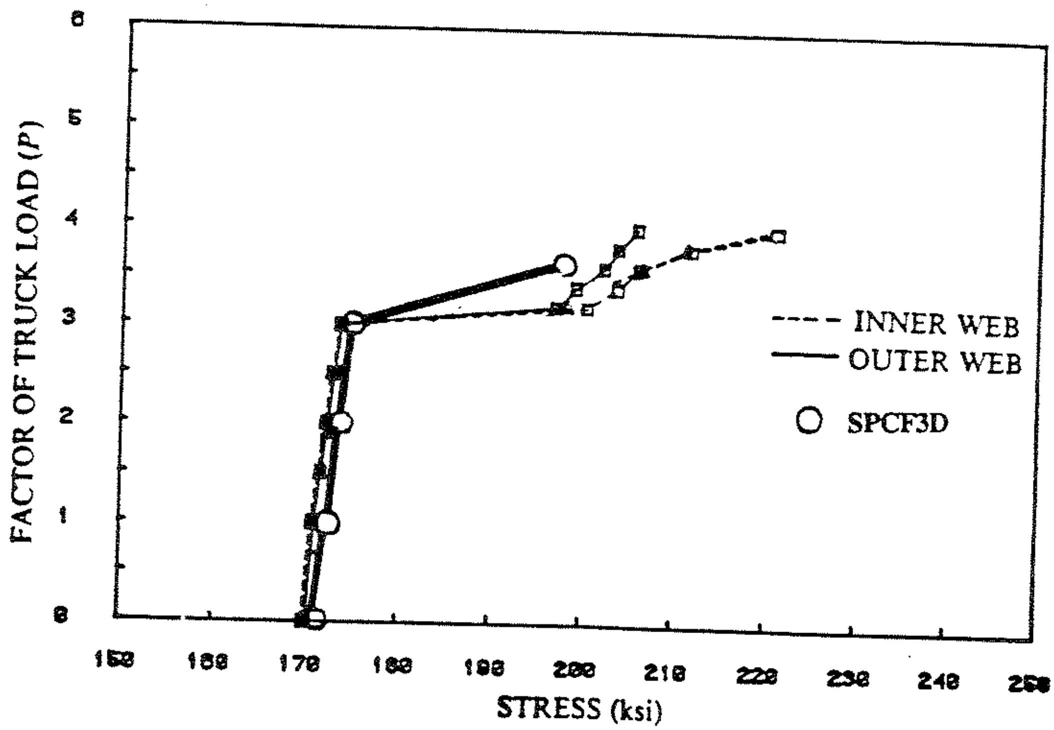


Fig. 5.3.2.8b EXAMPLE 5.3 - LOAD vs. PRESTRESSING NEAR INTERIOR SUPPORT SPAN (LOAD CASE 3) Choudhury [30]

LOAD CASE	INITIAL CRACKING AT MIDDLE OF CENTER SPAN		INITIAL CRACKING AT INTERIOR SUPPORT	
	SPCF3D	NAPBOX	SPCF3D	NAPBOX
CENTERLINE LOAD	P = 1.5	P = 1.5	P = 3.1	P = 3.4
OUTERWEB LOAD	P = 2.0	P = 2.0	P = 3.0	P = 3.0
INNERWEB LOAD	P = 1.5	P = 1.5	P = 3.0	P = 3.2

TABLE 5.3 SUMMARY OF CRACKING LOADS

due to stressing operations are lower compared to the corresponding forces for the case of the straight bridge analysed in the previous example. The change in the prestressing forces in the two tendons with load increments reflects the extent of the tendon contribution in carrying the load. For load cases 1 and 2, the externally applied load is practically carried out uniformly by the two webs. The deviation from a common prestressing force value may be related to the concrete cracking that is initiated. For load case 3, the large difference in the two web prestressing force is attributed to the transverse flexural deterioration in the ability of the box section to distribute eccentrically applied loads. Since the current implementation **SPCF3D** neglects any transverse deformation or distortion, it cannot model this response as is clearly emphasized in **Fig. 5.3.2.8a&b**.

5.4 Example-5.4-Three-Span Prestressed Cantilever Concrete Box Girder Bridge

A three span prestressed concrete box girder bridge built segmentally following a cantilever approach in the erection is treated in the current discussion. The results obtained for displacements, internal bending actions and prestressing force variations using the computer program **SPCF3D** are compared to similar output obtained by Ketchum from **SFRAME** analyses [27] for the same structure. Cast in place elements as well conventionally erected segments are considered in the erection sequence and the response of the structure under its own weight and additional construction and prestressing loads is traced throughout the life of the structure.

5.4.1 Bridge Design

A straight, three span segmentally erected box girder single cell haunched bridge designed by Ketchum [27] and analysed by him using **SFRAME** [27] as depicted in **Fig. 5.4.1.1** has been chosen for the current discussion to demonstrate the capability of **SPCF3D** in analysing this two dimensional frame structure as a special case of the more general classes of three dimensional frame structures and to check the accuracy of the results obtained. Haunched cast in place girder segements are used in the cantilevering operation and are post tensioned to the previous girder segment through Closure segments at mid span and abutments are installed following conventional erection methods and once

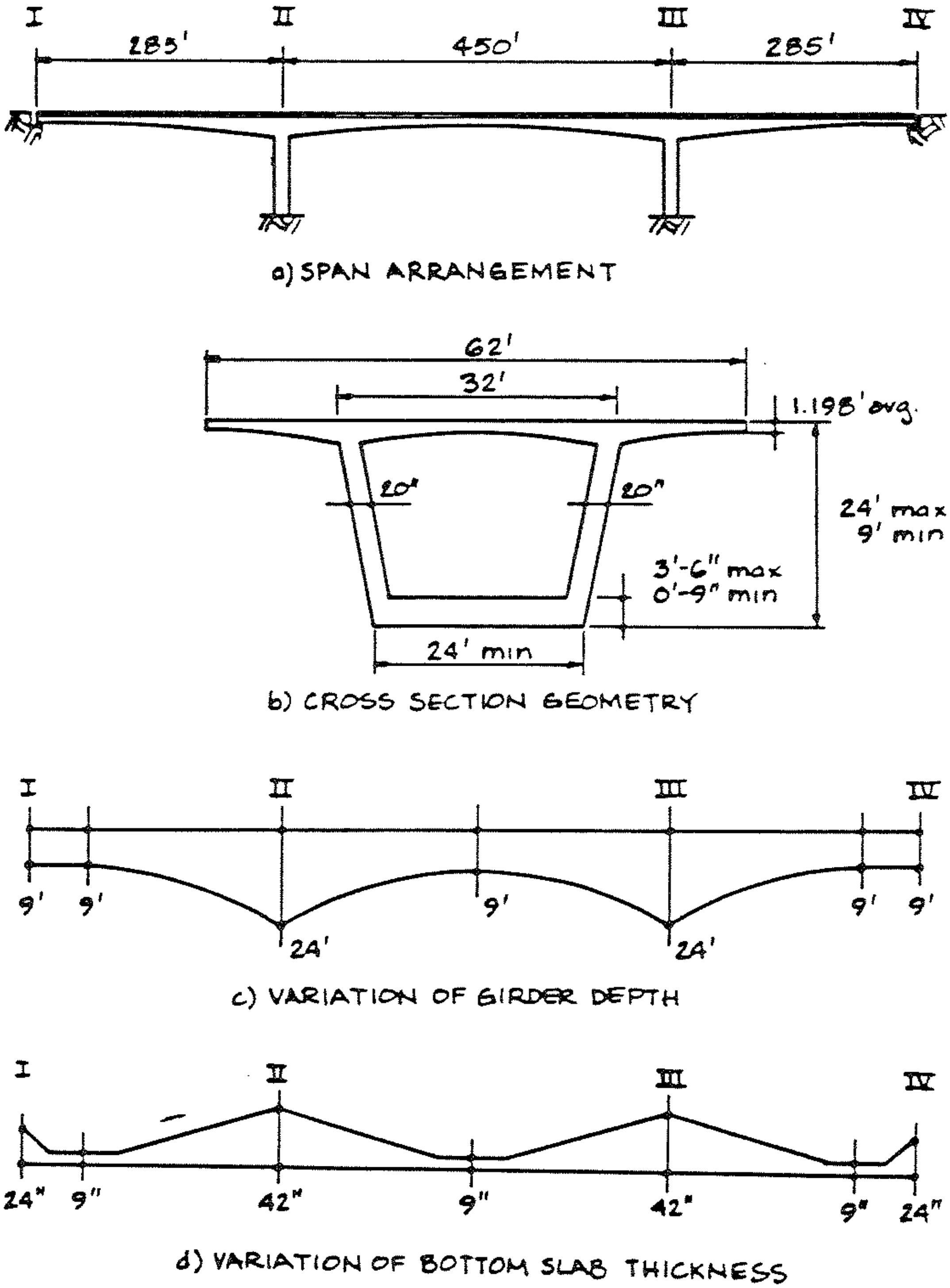


Fig. 5.4.1.1 THREE SPAN CANTILEVER CONSTRUCTION BRIDGE
Ketchum [27]

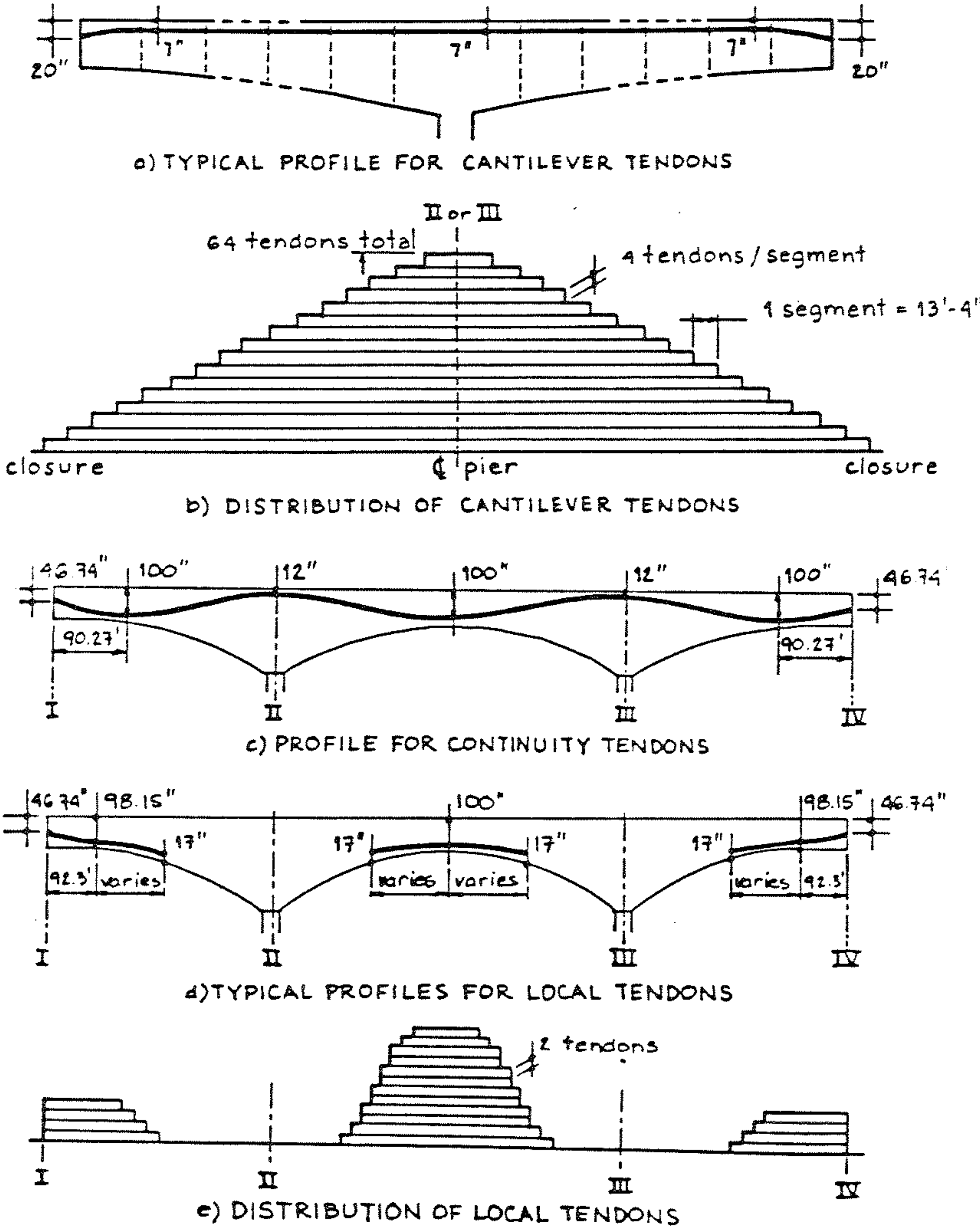


Fig. 5.4.1.2 PRESTRESSING TENDON LAYOUT

Ketchum [27]

these closure segments are in place additional continuity prestressing tendons extending over the whole length of the box girder bridge are installed. Local tendons at high positive moment locations have been added as well. A detailed description of the segmental construction of this structure with time is given by Ketchum [27].

A symmetrical span configuration with two 285 foot side spans and a center span of 450 feet **Fig. 5.4.1.1a** span as well as a symmetric cantilevering operation about each pier is used. Such an arrangement improves on the efficiency in the cantilevering operation and the structural behaviour.

The single cell box girder typical cross-section is shown in **Fig. 5.4.1.1b** and is subdivided into three sub-cross sections for the analysis. The subdivisions correspond to the physical variations in the thickness of the cross section meaning that the upper flange, web and lower flange are the three sub-cross sections. The depth of the girder from the upper edge of the upper flange to the lower edge of the lower flange varies gradually though not linearly from 24 feet at the inner piers to 9 feet at mid span and abutments. Consistently, the lower flange thickness varies gradually from 3.5 feet at the inner piers to 0.75 feet at midspan and its width from 29 feet at midspan and abutments to 24 feet at the inner piers. All remaining cross sectional dimension are assumed to be constant over the length of the structure.

Three different types of prestressing tendons are considered in the bridge configuration. The classification of these tendons has been based on location, purpose and continuity. For the cantilevering operation during the initial phases of construction, four "cantilever" prestressing tendons located within the top slab of each added segment and stressed from both ends have been used bringing the total number of such tendons to 64 over the inner piers. The distribution and geometry of these tendons is shown in **Fig. 5.4.1.2a** and **Fig. 5.4.1.2b** respectively. For continuity, eight "continuity" tendons were installed in the webs of the box girder cross section over the full length of the structure. As the name indicates, the bridge is made continuous with the installation of the closure segments **Fig. 5.4.1.2c**. The stressing operation is performed separately for the inner span from both sides. The center span tendons are then coupled at the pier locations to the side span tendons which are subsequently stressed from the abutments only.

"Local tendons" are installed in the center span (24 in total) and the side spans (8 in total) as shown in **Fig. 5.4.1.2d** and **Fig. 5.4.1.2e**.

All tendons consist of 21, 2 1/2 inches diameter strands.

The design loads for the aforementioned bridge structure consist of the structure own weight based on a 155 pcf concrete unit weight, an additional 2.5 kips/foot dead load and an HS20-44 lane loading according to the AASHTO Code recommendations.

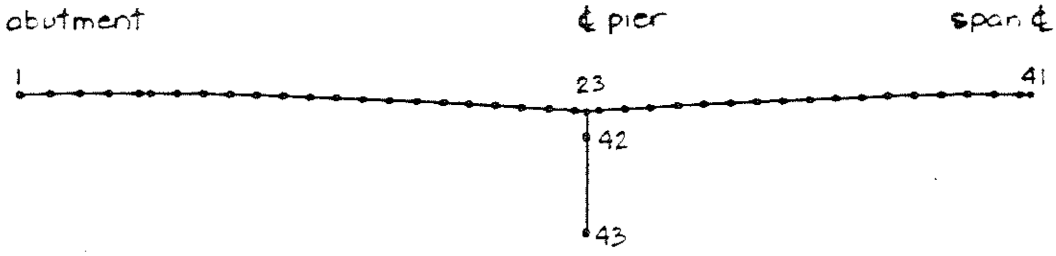
An ultimate strength analysis was performed by Ketchum [27] to proportion the box girder cross section and top slab prestressing based on the design loads mentioned in the previous paragraph applied to the statically determined cantilever structure existing during the initial segmental operations and prior to the closure-continuity stage. Proportioning of the local and continuity tendons was based on an allowable stress approach with the full design loads applied to the complete structure.

The design of the tendons was updated to allow for full redistribution under full design loads.

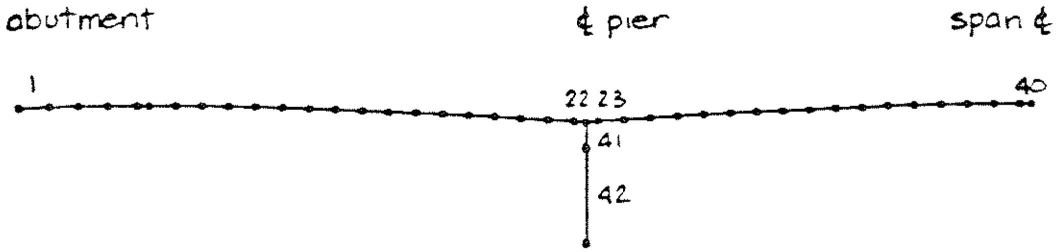
5.4.2 Analytical Model

Due to symmetry in the bridge configuration and load distributions, half of the bridge structure need to be analysed only. The lb-in English system of units is used in the analysis.

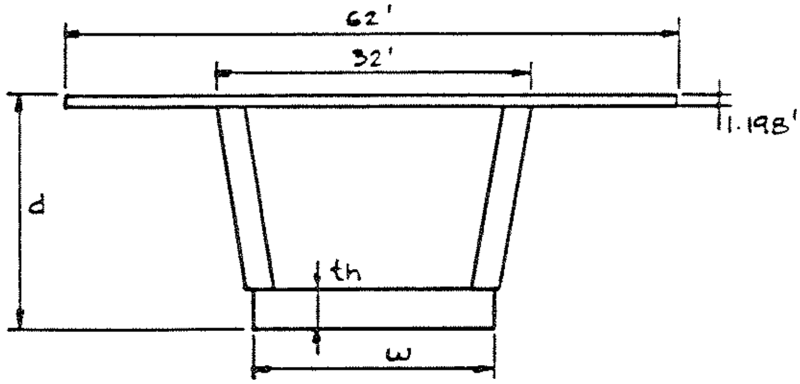
Despite the fact that the beam element formulation used in **SPCF3D** allows arbitrary locations of the nodes, the 43 nodes modeling the bridge structure are located at the centroidal level of the segment intersections to be consistent with the model used by Ketchum [27]. A node generation scheme in **SFRAME** and **SPCF3D** is behind the node numbering adopted and shown in **Fig. 5.4.2.1a**. The bandwidth of the structure global stiffness matrix is consequently reduced by specifying the sequence of nodes to be followed with the use of the SEQUENCE command. Except for boundary nodes 1, 41 and 43, the degrees of freedoms for all other nodes are set for free displacement restraint conditions. Vertical restraint is assumed at node 1 while horizontal and rotational restraints are enforced at node 41, center span mid length, for symmetry considerations. Node 43 is completely fixed assuming bedrock boundary conditions. Nodes 2 thru 5 near the abutments are temporarily restrained in the vertical direction in order to model the temporary shoring used wich is



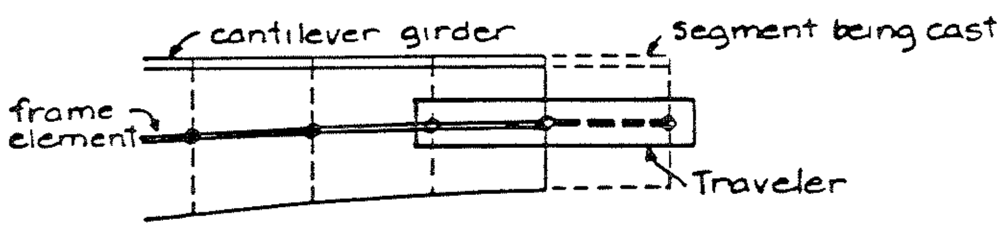
a) NODE NUMBERING



b) FRAME SUPERELEMENT NUMBERING



c) CROSS SECTION IDEALIZATION



d) TRAVELER IDEALIZATION

Fig. 5.4.2.1 ANALYTICAL MODEL

Ketchum [27]

removed upon installation of the continuity segments.

The 42 frame superelements **Fig. 5.4.2.1b** used to model the piers and the girders are assumed to be prismatic over their own length with the actual mid length cross section assumed representative. For superelement cross section properties and dead load generation, the cross section is subdivided into three frame element sub-cross sections as shown in **Fig. 5.4.2.1c**. Mild steel proportioning was based on a uniform distribution of 0.02 % in the girder and 0.1 % in the pier for all superlements.

The traveler element consist of a two standard adjacent frame elements **Fig. 5.4.2.1d**. The front frame element is used to support the new concrete segment. The rear frame element provides support for the traveling formwork. Total weight of the traveler is assumed to be 150 kips.

Tendon elements are used in modeling the prestressing steel. 30 tendon elements subdivided into 442 tendon segments were needed to idealize the cantilever, local and continuity prestressing steel.

Cantilever tendons are numbered from 1 to 16. The profile of these cantilever tendons consist of a straight curve over most of the length except for a short portion near the anchorage. Cantilever tendons are jacked from both ends using a 2550 kips stressing force.

The eight continuity tendons in the side and center spans are modeled by tendon numbers 17 and 18 respectively. The profile of these continuity tendons is parabolic and hence may be internally generated through parametric generation. These tendons are jacked from the left end only with a 5100 kips stressing force.

Local tendons in the side and center spans are numbered from 19 through 30. Similar to the continuity tendons, the local tendons are located in the webs of the box girder sections. The geometry is parametrically generated and the stressing force varies from 1275 kips for tendons number 19 through 22 which are stressed from both ends to 2550 kips for tendons 25 through 28 which are stressed from the left end only to 1275 for the remaining tendons (23, 24 and 30) which are stressed from the left end only as well.

5000 psi concrete ultimate strength, 2.5 ultimate creep factor, 0.0008 ultimate shrinkage strain, 155 pcf concrete unit weight, 29000 ksi mild steel elastic modulus, 28000 ksi prestressing steel elastic modulus, 270 ksi prestressing steel ultimate strength, 10 relaxation coefficient, 0.25 / radian curvature friction coefficient, 0.0004 / foot wobble coefficient and 1/4 inch anchorage slip have been adopted for the material specifications for the current numerical study. Time dependent parameters has been obtained according to the ACI recommendations.

5.4.3 Loadings and Construction Sequence for the Time Dependent Analysis

A 7-Day cycle segmental construction is assumed during which a cast in place concrete segment is added and post tensioned to the existing structure. Continuity tendons are installed to the complete structure after the closure segments are put in place. Additional dead load is considered at that stage and the full structure is analysed through a 27 years service period.

The construction sequence of the box girder three span bridge is initiated with the erection of the inner pier column and the adjacent girder, followed by the segmental addition of the individual cast in place concrete segments which are post tensioned to the existing structure through cantilever prestressing tendons. Subsequently, the approach segments at the abutments are build and the side spans closure segments are installed and side spans local tendons stressed. The construction operations are terminated by installing the center span closure segments, stressing the center span local tendons, stressing the structure continuity tendons which extend over its full length and the addition of the superimposed dead load.

During the cantilever segmental construction, the traveling formwork is moved into position before the cast in place segment is added. A three day period of idle time, normally falling on a weekend, is needed here before the newly cast segment is post tensioned to the existing structure. The formwork and construction is then moved to the new location requiring a four day period of time. Consequently, a total of 105 days are needed for the cantilever construction. Internally, the newly cast segment and post tensioning prestressing are added simultaneously to economize on the analysis since the time dependent parameters extracted from the big table assume a minimum of three day period of time. Hence, a seven day period of time exist between consecutive added segments requiring a total of 48 solution steps. The segments dead load addition and prestressing operation are performed in a two separate zero time length analysis steps. The time period existing between the end of the construction at day 196 and time 27 years is subdivided into two 10 analysis time steps obtained based on a logaritnmic scale subdivision. Again, a detailed description of the brief summaries given in sections **5.4.1** to **5.4.3** has been previously presented by Ketchum [27].

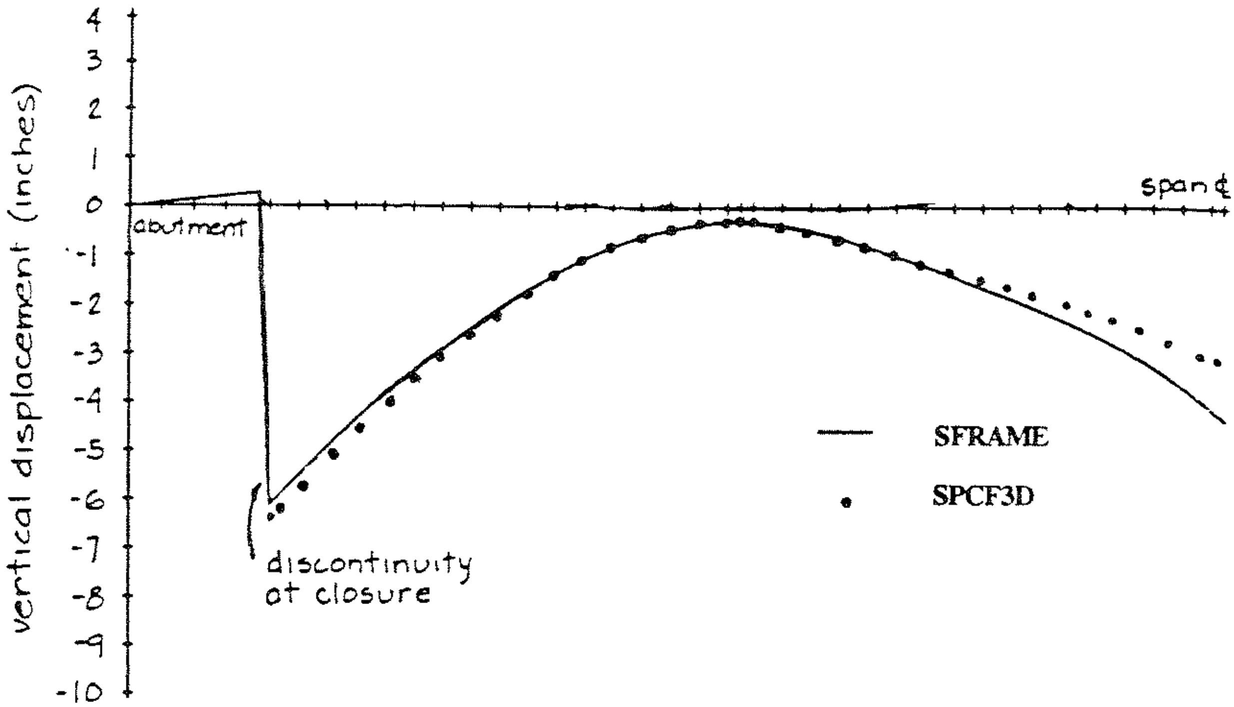
5.4.4 Discussion of Results

In the present section, the displacements, total static shear, total static superelement bending moments and the prestressing tendons force variations with respect to time through year 27 from start of construction as obtained with **SPCF3D** wich has been developed for the present study are presented , discussed and compared to the results Ketchum [27] obtained for the analysis he performed on the same structure using the computer program **SFRAME** he developed for his study.

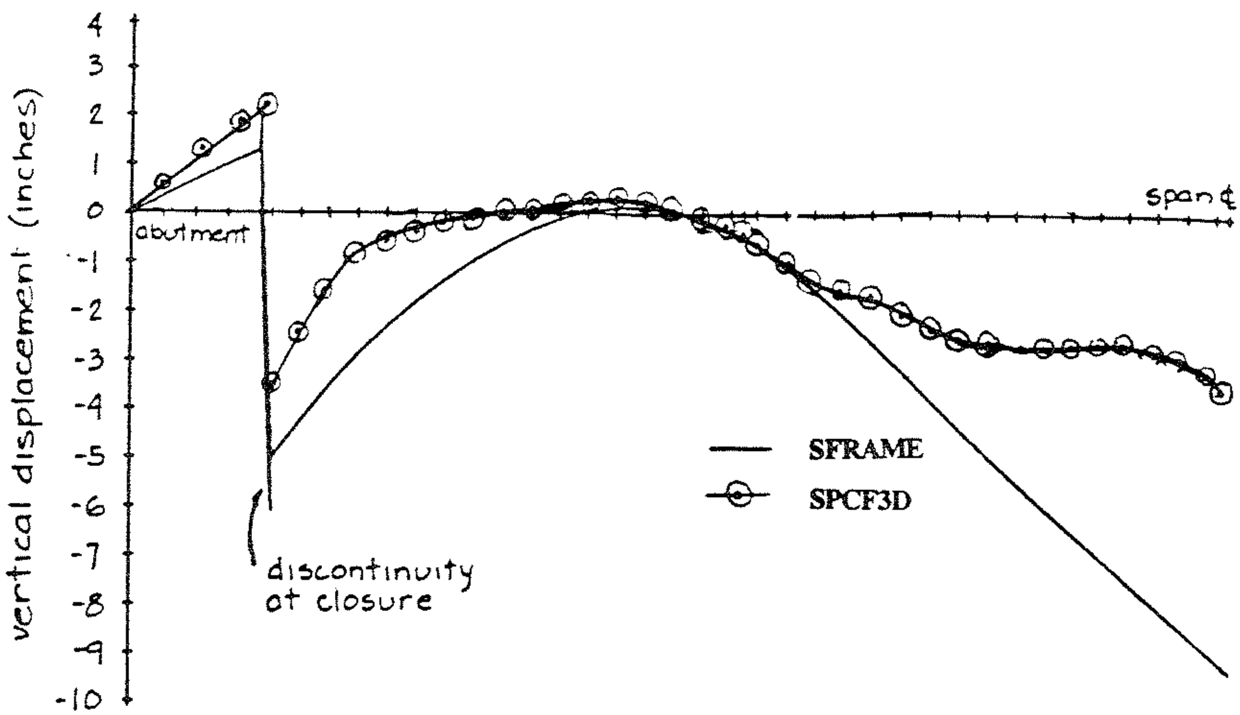
5.4.4.1 Displacements

The displacement profile of the full analysed structure at the end of the cantilevering construction phase before closure and at 27 years through the service life of the structure is shown in **Fig. 5.4.4.1.1**. The plotted profiles are the actual displacements configurations of the structure at these observation times assuming that no cambering has been performed during construction. Consequently, large discontinuities in the displacement profiles are observed at the closure segment near the abutment which may be eliminated by appropriate cambering during cantilevering operations. The amount of cambering needed at one end of a newly cast segment is the relative displacement of that end with respect to the displacement of the other end as obtained from **Fig. 5.4.4.1.1**. Consequently, the cantilever tips at the closure segment will possess zero elevation when continuity is enforced.

During cantilevering operations, the vertical displacement profile is closely symmetrical about the inner pier column. Such a behaviour would have been expected due o the symmetry in the geometry and loading. The symmetry would have been complete had the friction loss distribution for prestressing been symmetrical too. After the closure segment is installed, the symmetry disappears and the center span displacements becomes larger than the side span's due to the larger center span



a) DISPLACEMENTS AT COMPLETION OF CONSTRUCTION



b) DISPLACEMENTS AT 27 YEARS

Fig. 5.4.4.1.1 VERTICAL DISPLACEMENT PROFILES
Ketchum [27]

dimension. Comparing the results obtained from the two computer programs, perfect correlation between the two plots is observed at the end of the cantilevering phase. However, at 27 years through the service life of the structure a large discrepancy in the two responses is noted and both spans long term displacements are upward for **SPCF3D** while only the side span behaves in that manner for **SFRAME**. The discrepancy may be attributed to the choice of the modulus of elasticity for load and time dependent computations adopted in either program. For **SPCF3D**, the elastic modulus at different observation times for creep computations is obtained from the ACI relationship but for load calculations is consistently obtained from the material stress-strain diagram. In the case of **SFRAME** the elastic modulus is computed from the ACI relationship for both cases.

The displacement profiles shown in the figure are lower bounds for the actual behaviour of the structure since shearing deformations have been neglected in the frame superelement formulation. Such an assumption is adequate for all practical purposes.

5.4.4.2 Distribution of Bending Moments

The statical moments profile in the box girder structure is shown in **Fig. 5.4.4.2.1** for observation times corresponding to completion of construction stage following continuity and 27 years service period. The results are compared to similar results obtained by Ketchum [27].

The statical superelement moment is the sum of bending moment contributions from all concrete and mild steel sub-cross sections belonging to those frame elements within the superelement considered. These statical moments are in equilibrium with the total dead loads and reactions and consequently illustrate the moment redistribution of the structure statical system.

Some redistribution of the statical moments across the bridge structure is observed for **SPCF3D** but very little for **SFRAME** but both of the profiles at the end of the construction stage and year 27 are within acceptable comparison with the results obtained by Ketchum [27] for the same structure. A significant portion of this redistribution may be attributed to the losses in the prestressing forces where losses have been observed in the 15-25 percent range. Time dependent redistribution due to creep and shrinkage should be also considered as a contributing factor.

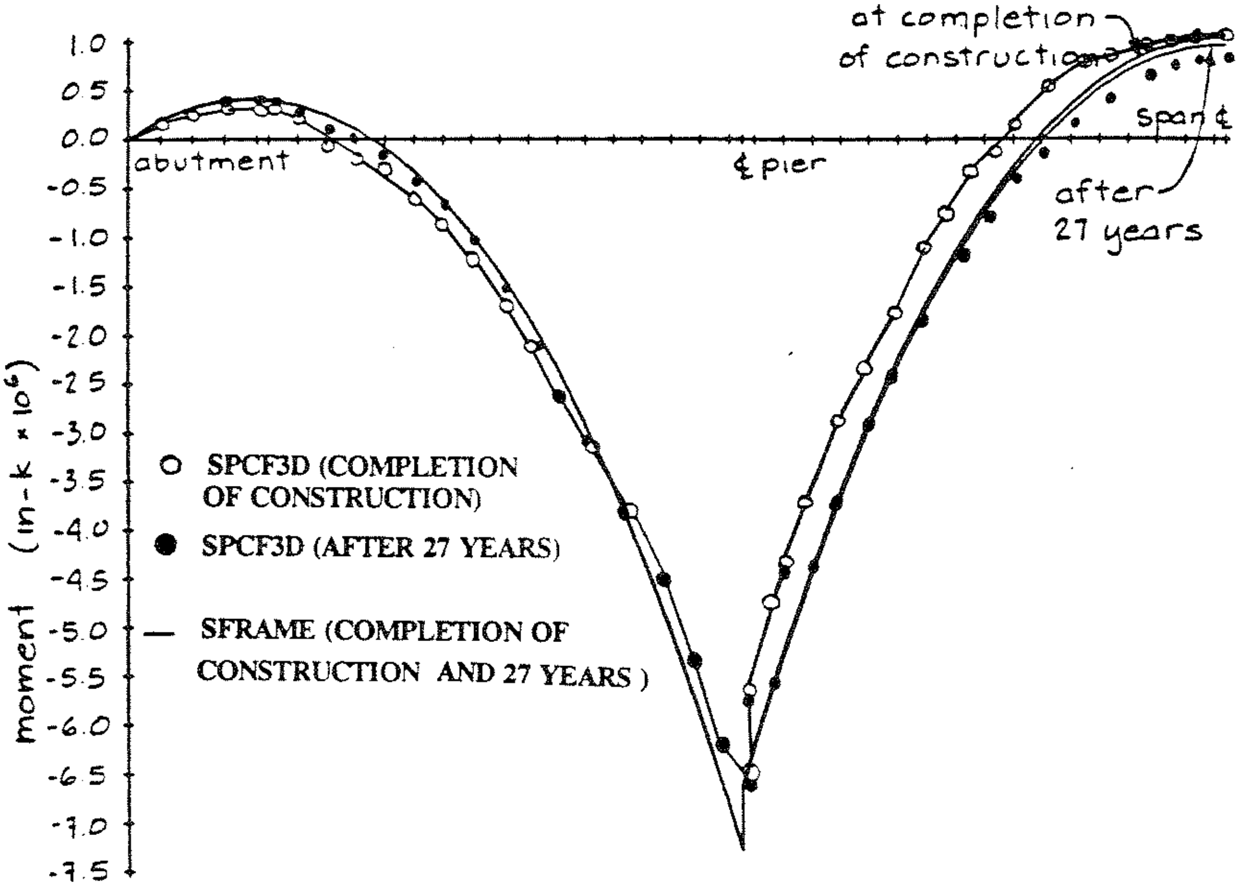


Fig. 5.4.4.2.1 REDISTRIBUTION OF BENDING MOMENTS

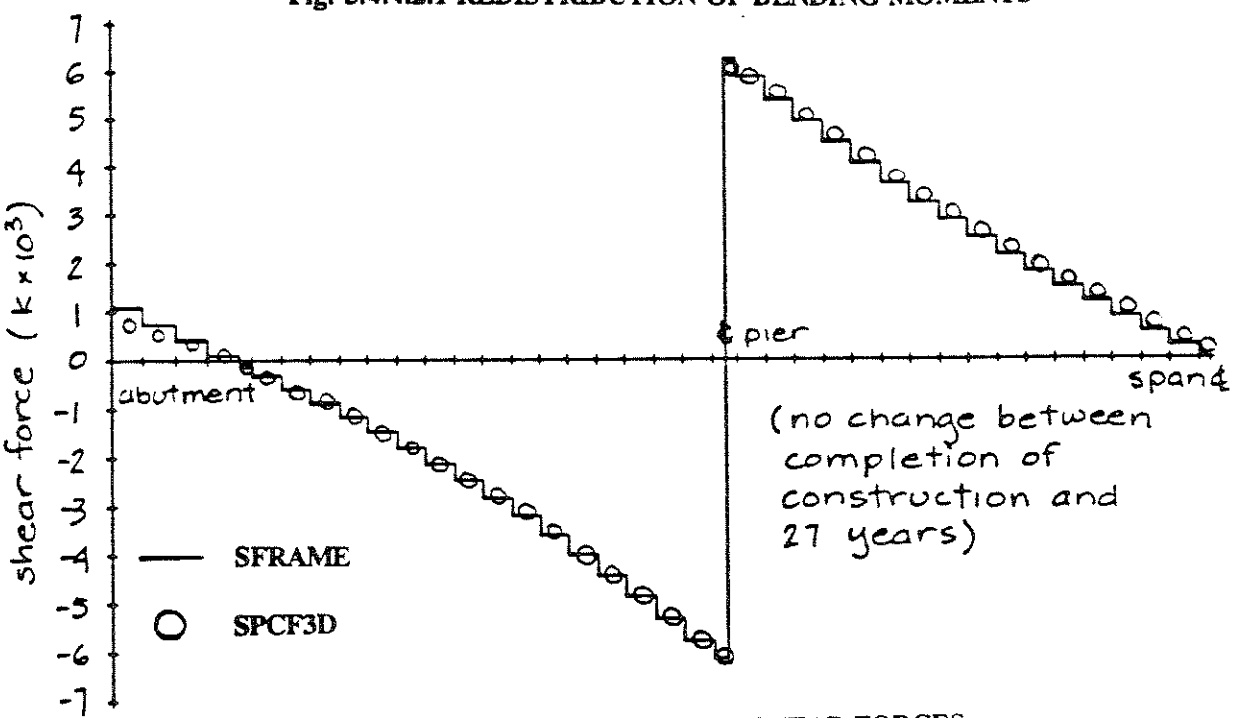


Fig. 5.4.4.3.1 REDISTRIBUTION OF SHEAR FORCES

Ketchum [27]

5.4.4.3 Redistribution of Shear Forces

The **SPCF3D** statical shear forces profile in the box girder structure are shown in **Fig. 5.4.4.3.1** for observation times corresponding to completion of construction stage following continuity and 27 years service period. The results are compared to similar results obtained by **SFRAME** [27].

The statical superelement shear forces are the sum of similar contributions from all concrete and mild steel sub-cross sections belonging to those frame elements within the superelement considered. These statical shear forces are in equilibrium with the total dead loads and reactions and consequently illustrate the shear force redistribution of the structure statical system.

Very little redistribution of the statical shear forces across the bridge structure is observed and an excellent correlation in the results obtained from **SPCF3D** and **SFRAME** [27] is noted. A significant portion of this redistribution may be attributed to the losses in the prestressing forces where losses have been observed in the 15-25 percent range. Time dependent redistribution due to creep and shrinkage should be also considered as a contributing factor.

5.4.4.4 Percentage Loss of Force in Prestressing Tendons

A summary of the percentage prestressing steel force loss over the service life period of the structure is shown in **Fig. 5.4.4.4.1**. The percentage loss value is simply the relative change in the force value between the force at 27 years and the initial force at stressing. The results obtained using **SPCF3D** developed for the present study are compared with similar results obtained by **SFRAME** [27]. Excellent correlation between the two plots is noted and the variations for tendons 1 through 5 and 26 through 30 are compared in the same figure for the sake of illustration. It may be observed that the loss in the prestressing force ranges between 15 and 25 % for all tendons.

Tendon numbers 1 through 16 exhibit non-uniform variation in the prestressing force over their lengths. This distribution may be attributed to the non-uniform age of the concrete segment along the tendon length and the increase in the negative moment in the cantilever substructure especially at the inner pier leading to higher tensile strains at the top of the box girder section. Consequently, larger prestresing force loss is expected closer to the inner pier than the stressing ends of the tendons.

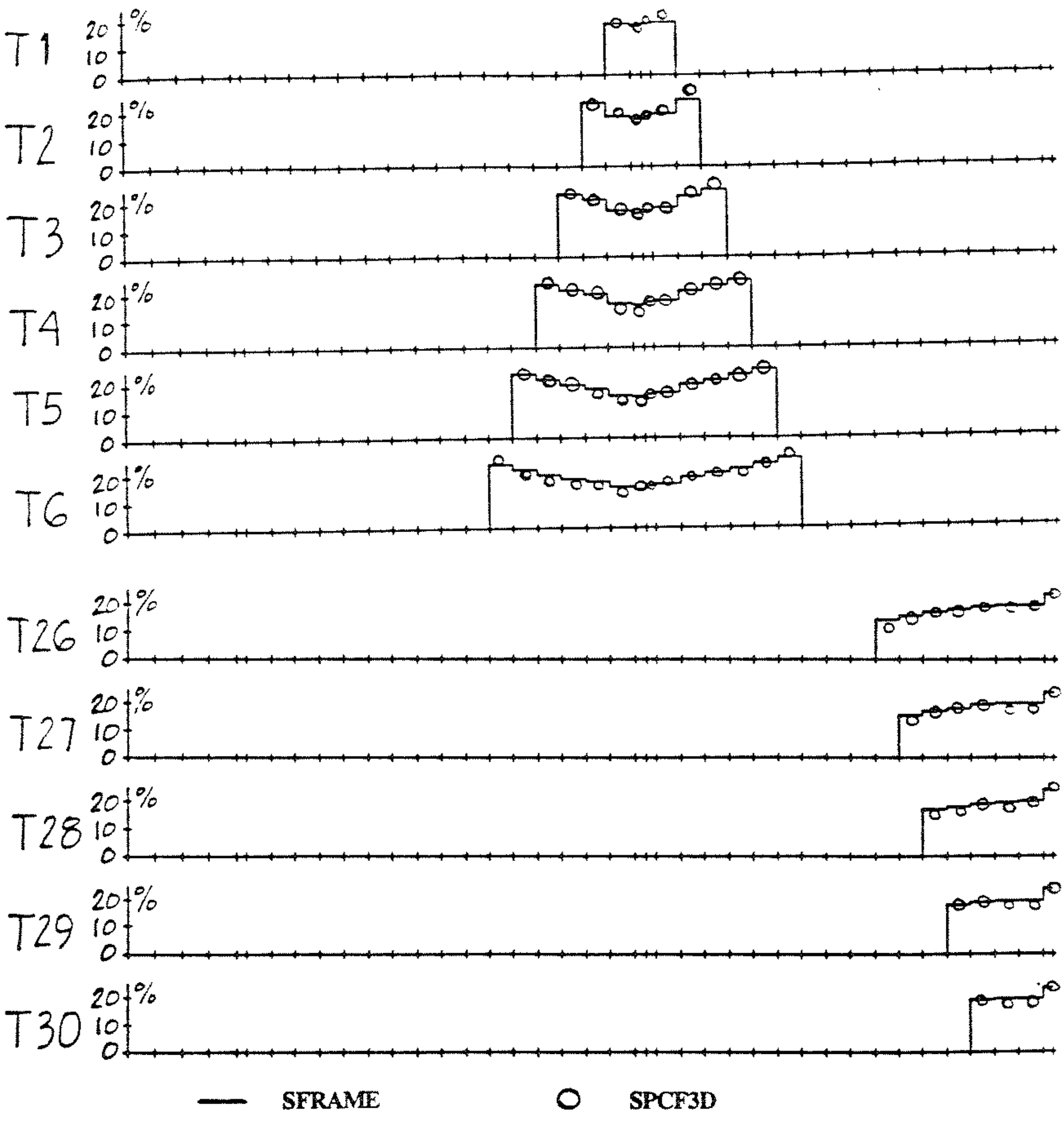


Fig. 5.4.4.4.1 PERCENTAGE LOSS OF FORCE IN PRESTRESSING TENDONS
Ketchum [27]

The prestressing force variation for the local and continuity tendons, numbered 17 through 30, is almost similar. Again, the non-uniform variation in the concrete segments age is probably the main factor causing such force profile.

In summary, the assumption of a uniform loss in the prestressing force along the tendon length as followed in the present design and analysis practice may lead to large inaccuracy and a more precise approach is needed.

5.5 Example 5.5-Two Span Simply Supported Precast Prestressed Girders Made Continuous and Composite

A two span composite post-tensioned prestressed concrete box I-girder overpass bridge built segmentally over the depth of the cross section is analysed in the present discussion. The structure bears great similarities to the simply supported multi-span girders presented in a recent NCHRP report 322 by Oesterle, Glikin and Larson [36] and made continuous through a cast in place deck. However, the material properties used in the latter presentation were not available. Post tensioned prestressing was considered for the example in the present study although pretensioning operations may be modeled equally through proper construction sequence and material properties assignments in the computer program **SPCF3D**.

5.5.1 Structure Geometry and Analytical Model

A straight, two span simply supported girder bridge made continuous and composite is analysed using the computer program **SPC3D** developed for this study in order to demonstrate the capabilities of the program as a useful tool to predict accurately the analytical behaviour of similar classes of structures.

Inverted T-shape cast in place girders built at day 01 and postensioned in situ at day 07 were used and simply supported at the end nodes. Each 40 ft span girder was subdivided into four equal length incremental beam superelements as shown in **Fig. 5.5.1.1b**. The structure modeling, frame elements, superelement and node numbering is also shown in the same figure. The dead load of the

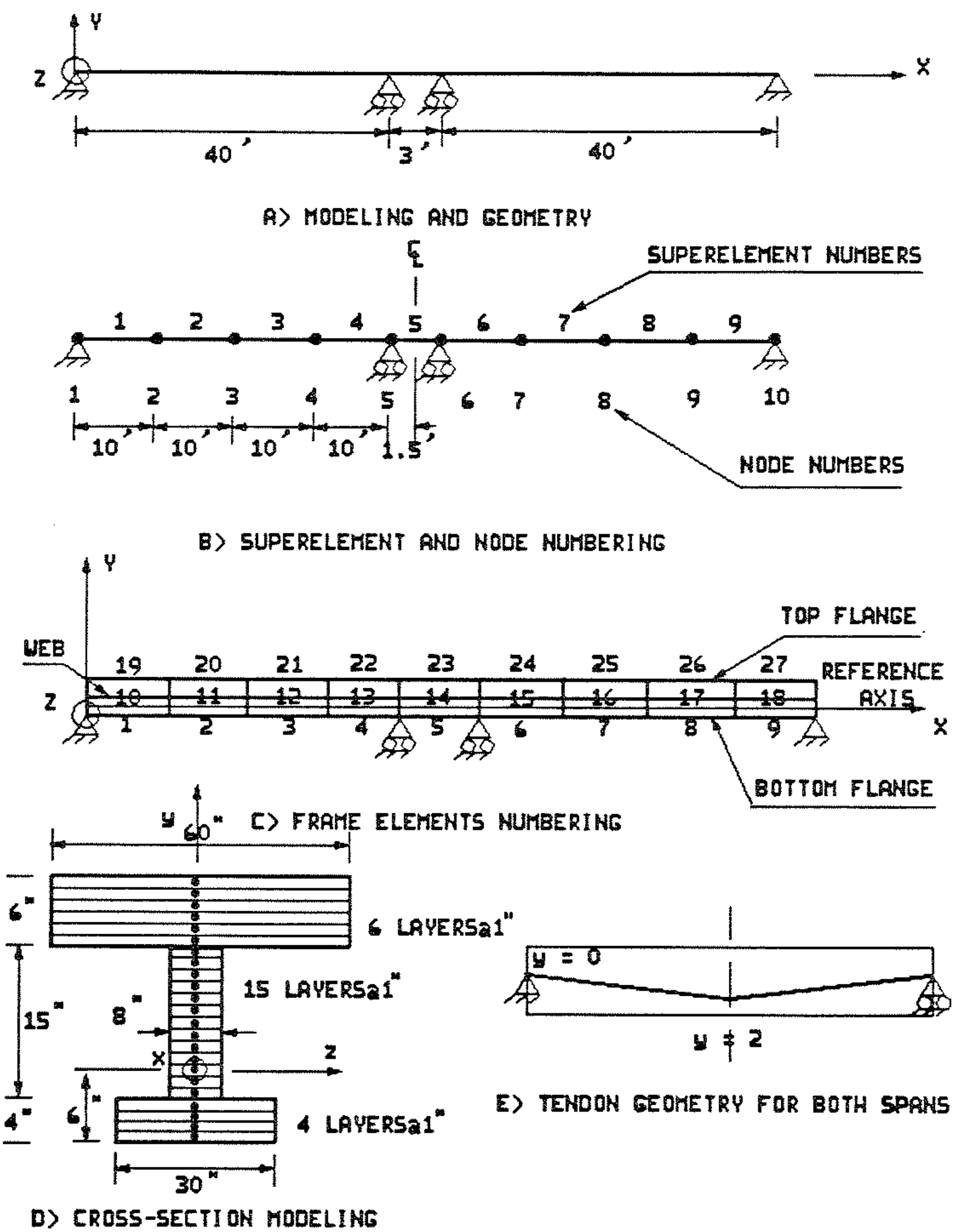
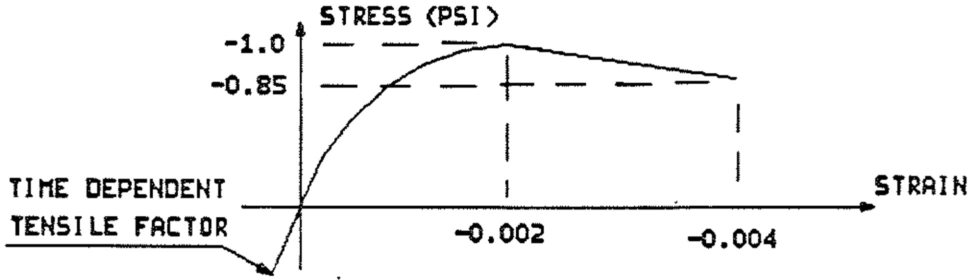
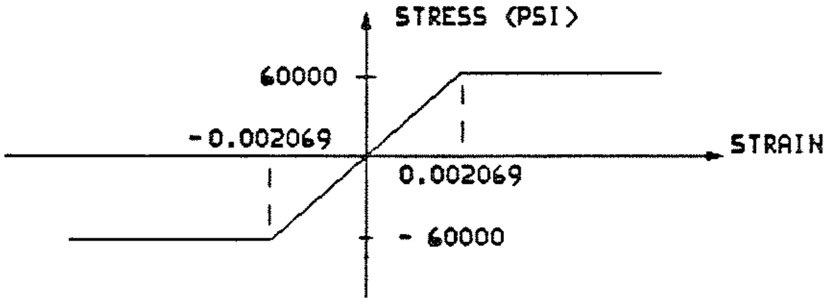


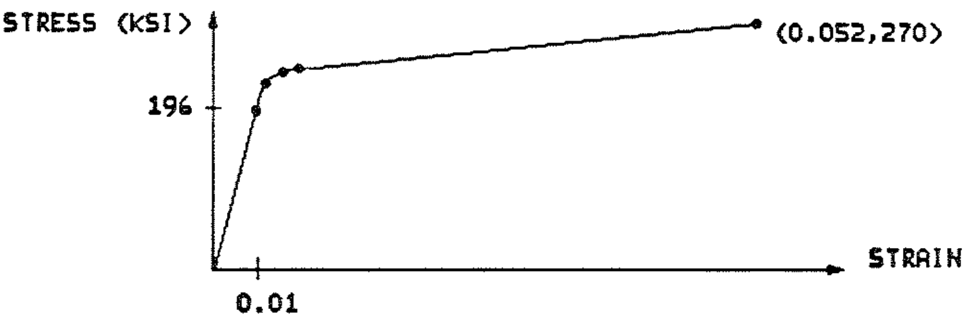
Fig. 5.5.1.1 GEOMETRY AND STRUCTURAL MODELING



A) NORMALIZED CONCRETE STRESS-STRAIN DIAGRAM



B) MILD STEEL STRESS-STRAIN DIAGRAM



C) PRESTRESSING STEEL STRESS-STRAIN DIAGRAM

ULTIMATE CREEP COEFFICIENT = 2.5 $\quad f'_c$ (28 DAYS) = 5000 PSI

ULTIMATE SHRINKAGE = 0.006 $\quad f_y$ = 60000 PSI

RELAXATION COEFFICIENT = 10.0 $\quad f_p$ max = 270000 PSI

Fig. 5.5.1.2 MATERIAL PROPERTIES

cast in place deck is carried completely by the girders already in position. The cast in place deck and the continuity connection is assumed effective at day 56 despite the fact that their corresponding dead load is transferred to the simply supported girders at day 28. Such an assumption allows the slab to develop adequate tensile and compressive strength needed in resisting the incremental live load.

The cross section discretization is shown in **Fig. 5.5.1.1d**. A uniform layer depth for the full section has been adopted. Consequently, 4 layers are used to model the lower flange, 15 for the web and 6 for the top flange or deck slab. No discretization in the local z direction was needed since the structure, loading and behaviour is two dimensional. The I- shape cross section has been subdivided into three sub cross-sections. The subdivision was made based on construction sequence and width representation requirements. For the latter reasoning, all the filaments of any frame element cross section must have the same width dimension. Consequently, the superelement cross section had to be subdivided into top flange, web and bottom flange subcross sections. For the former requirement, the top slab had to be independently represented in order to model the segmental operations across the depth of the cross section.

The connection joint located between the two girders is needed to provide continuity between the two spans. These types of connections, as has been mentioned in the introduction, provide an attractive solution particularly for the ease of construction and reduction in strength and deformation requirements. The controversy however, is related to the degree of continuity that these connections provide in resisting the loads applied to the structure during its service life.

The distribution through the I-girder cross-section of the mild steel reinforcement which is located in the mid-height of each layer is shown in **Fig. 5.5.1.1d**. A 1.0 % area reinforcement ratio has been chosen for the top and bottom flanges while a higher percentage value (2.0 %) was given to the web to resist the tensile stresses located in the web of the inverted T- section of the simply supported girders during the initial stages of construction. The choice of the reinforcement ratio was based on a linear analysis of the inverted T-section under total dead load and presstressing loads and of the continuous composite structure under dead,live and prestressing loads.

Referring to **Fig. 5.5.1.1e**, post tensioned tendons were used to model the prestressing steel

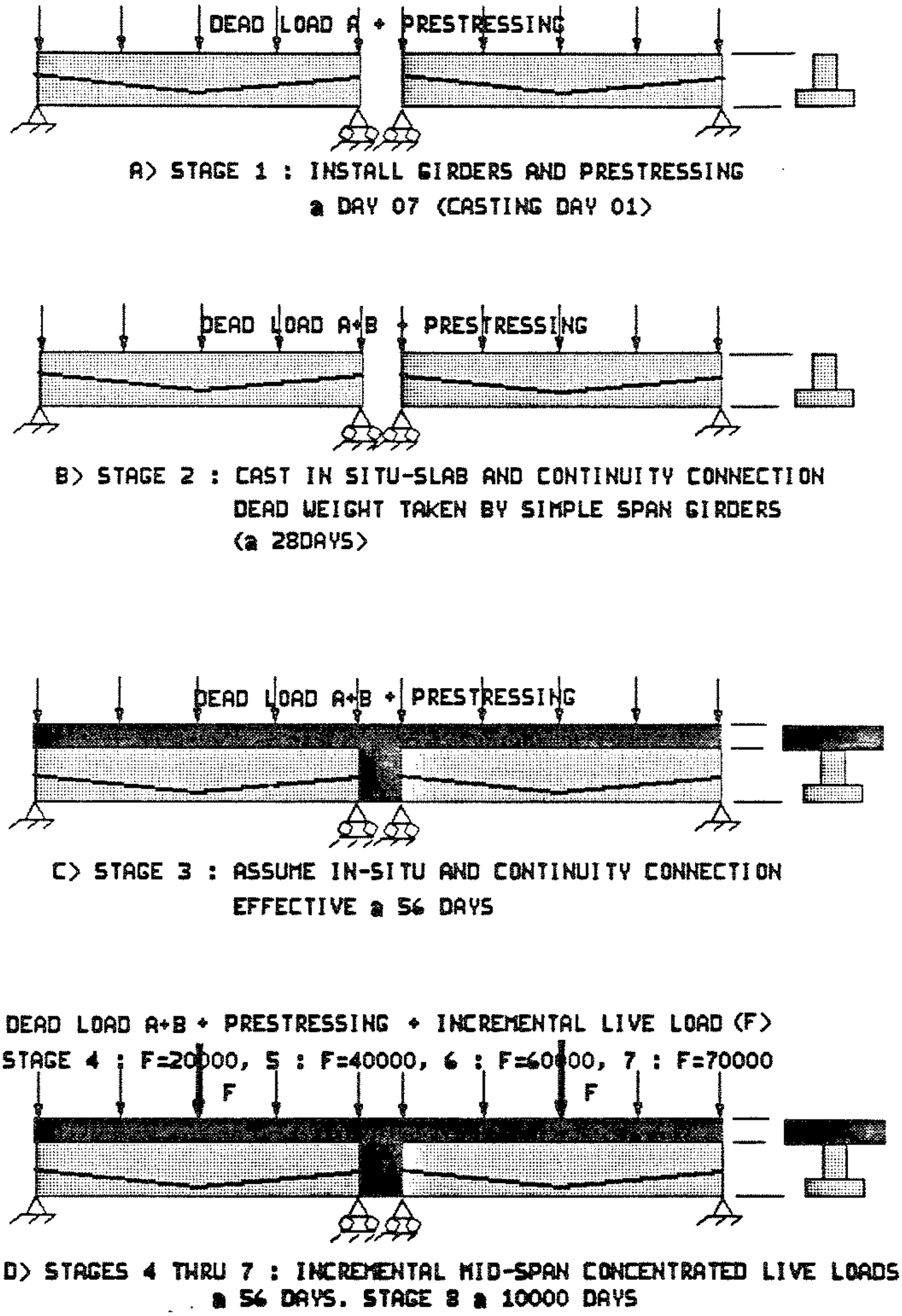


Fig. 5.5.1.2 CONSTRUCTION AND EXTERNAL LOADINGS HISTORY

installed in both spans. The design of these tendons was based on a linear elastic behaviour assumed for the simply supported girders subjected to its own dead loads and subsequently to the dead load of the added slab. The design was updated and checked for ultimate conditions with the ultimate live loads applied and the proper boundary and continuity conditions considered. Consequently, the required tendon cross sectional area was in the vicinity of 3.5 in^2. The bilinear tendon element profile and location in the global and local (X-, Y-) and (x-, y-) system was adopted to offset as closely as possible the multi-linear bending moment distribution due the simply supported girders and top slab dead weight then provide additional strength in resisting the aaplied incremental live loads.

A summary of material properties and material stress-strain diagrams for concrete, mild steel and prestressing steel is shown in **Fig. 5.5.1.2**. Values of 2.5, 0.0006 and 10.0 have been assumed for the time dependent material properties of creep and shrinkage in concrete as well as relaxation in prestressing steel respectively. The concrete stress-strain material model is a discretization of Kent and Park relationship commonly used for concrete analyses. Concrete strength at 28 days age is 5000 psi. Mild steel is assumed elastic perfectly plastic allowing for possible loading and unloading in tension and compression. The strength of mild and prestressing steel were assumed 60000 and 270000 psi respectively.

Construction phases and loading history of the simply supported structure made continuous and composite under live load is shown in **Fig. 5.5.1.3**.

5.5.2 Discussion

The vertical displacement at midspan through all construction phases and loading history, consisting of the structure dead load, prestressing, time dependent effects and incremental mid-spans concentrated live loads, has been plotted in **Fig. 5.5.2.1**. The simply supported structure cambers upward under its own dead load, prestressing forces and time dependent effects and the stiffness is subsequently increased with the addition of the in-situ deck slab. The response is almost linear for all these increments of the external live load. The displacement decreases from a maximum value of 1.36 inch to a value of 0.28 inch at maximum total live load at 27 years through the service life of

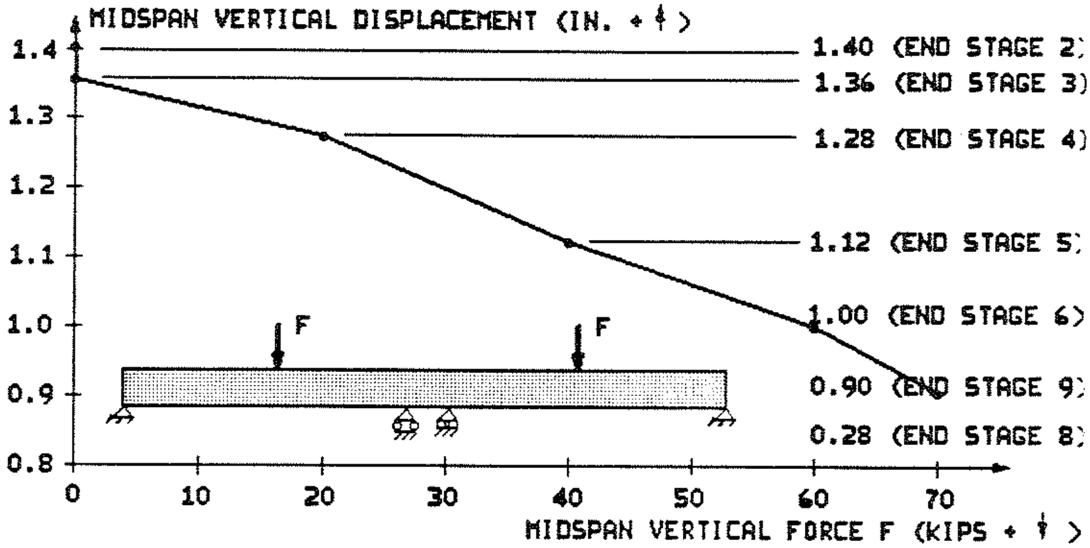


Fig. 5.5.2.1 MIDSPAN DISPLACEMENT THROUGH ALL PHASES OF CONSTRUCTION AND LOADING

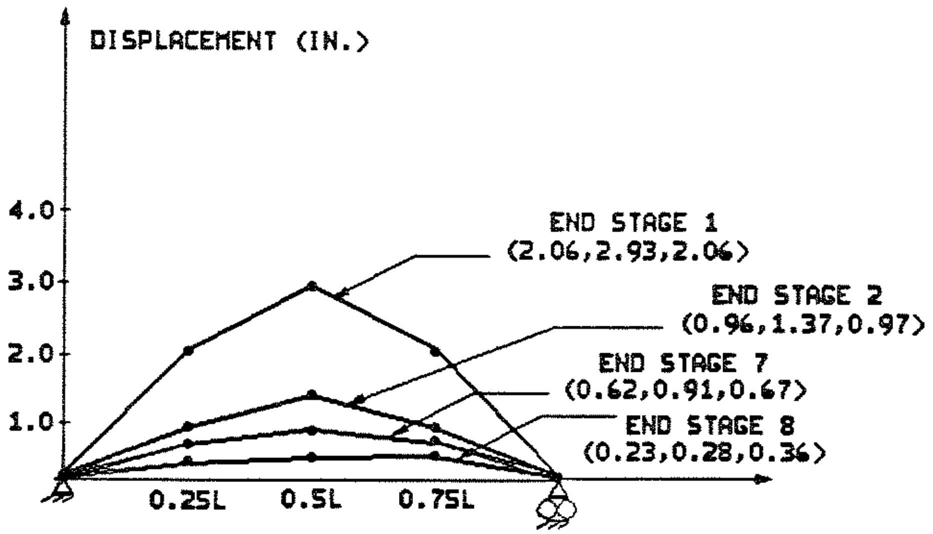


Fig. 5.5.2.2 DISPLACEMENT PROFILES

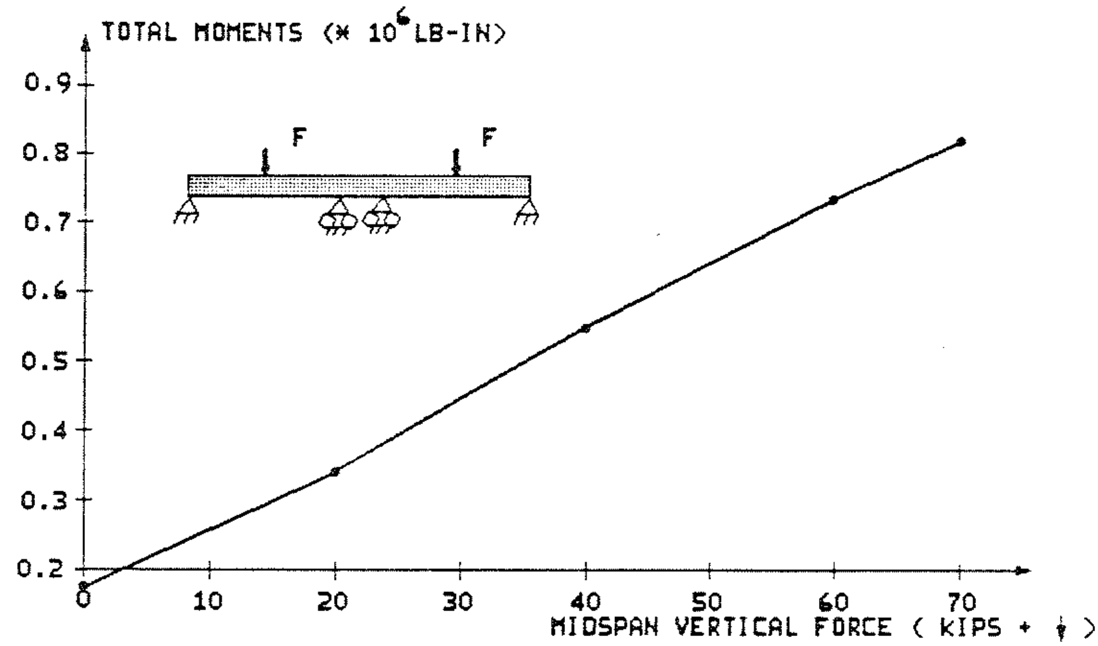


Fig. 5.5.2.3 MIDSPAN TOTAL MOMENTS STAGES 4 THRU 7

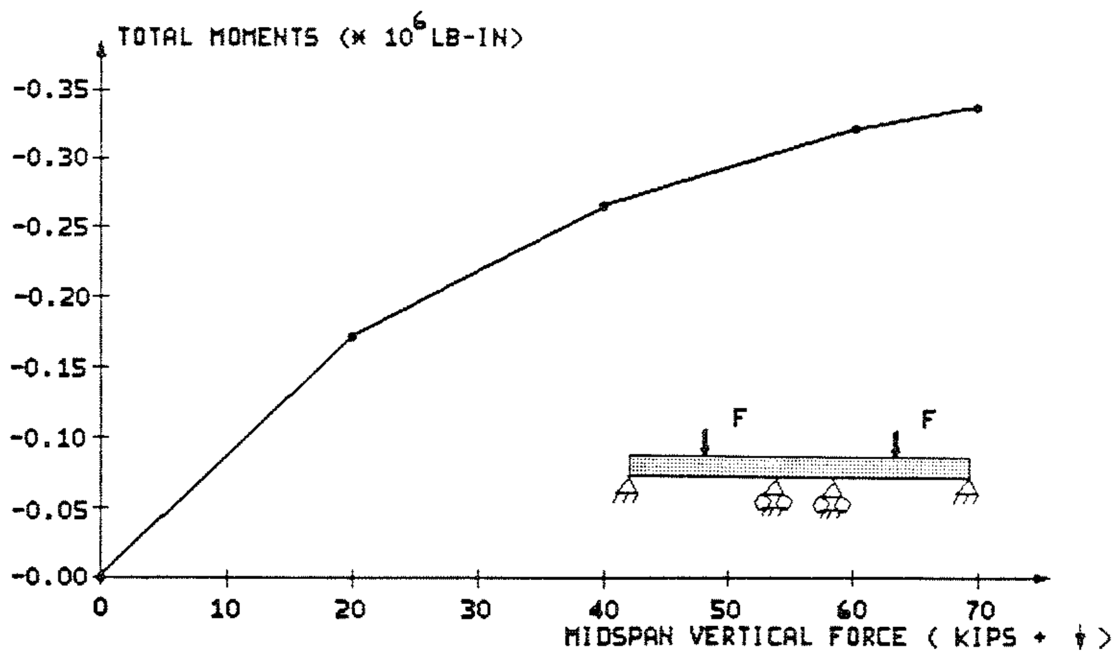


Fig. 5.5.2.4 INNER SUPPORT TOTAL MOMENTS STAGES 4 THRU 7

the structure. The drop in the displacement value due to live load increments is expected since it represents a deformation of the structure in the direction of the applied load. Note that the original displacement before the increments are applied is positive as a result of cambering due to the presence of prestressing. The displacement profile of the connection joint linking the adjacent spans has not been plotted with the displacement profile of these spans shown in **Fig. 5.5.2.2**. The reason is that this short link has been modeled as one beam superelement spanning between the adjacent girders end nodes. The profile of this link would have been similar to the profile of the adjacent spans with its curvature being also negative. The latter behaviour is true since the link has been installed in position at the time when the adjacent span profile under dead loads and prestressing had a negative curvature. Consequently, any subsequent loading in the configuration considered will result in a negative curvature for the link and a reduction in the curvature of the adjacent spans. A negative curvature for the link necessitates negative reinforcement in the top slab. **Fig. 5.5.2.2** highlights the girder displacement profile with the application of the incremental loads and at day 10000 through the service life of the structure. The span maintains an upward displacement throughout the loading and time histories of the structure despite the substantial live load added. This might be attributed to the original time dependent displacements of the simply supported spans through day 28 and the increase in the structure stiffness due to the addition of the in situ slab and the increase in the concrete age with time.

The response for the variations of the total bending moments at midspan and at the inner support plotted in **Fig. 5.5.2.3** and **Fig. 5.5.2.4** is also linear due to live load increments. The observation that positive total bending moment and negative total curvature exist at the midspan location does not present any contradiction since the total curvature is the sum of similar contributions from the simply supported girders and the full I- beam superelements having different stiffnesses and properties subjected to different loading configurations. In other words, the behaviour may be related to the segmental operations across the depth of the cross section. The total bending moment distribution along the spans at day 56 before any incremental loading is initiated, at day 56 corresponding to maximum live load and at day 10000 through the service life of the structure have been plotted in **Fig. 5.5.2.5**. Minimal redistribution in the bending moment values for the portion of the structure

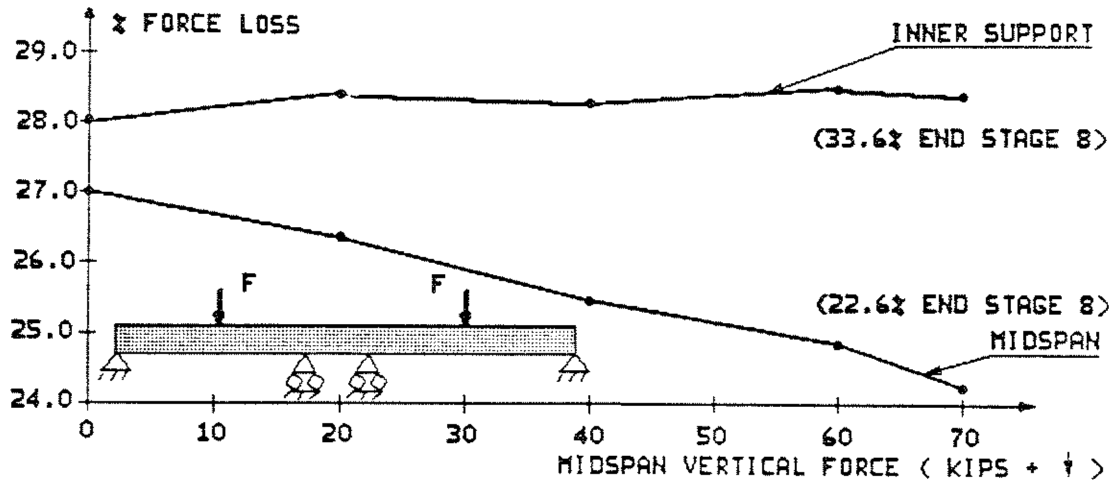


Fig. 5.5.2.7 MIDSPAN AND INNER SUPPORT PRESTRESSING FORCE LOSS
STAGES 4 THRU 8

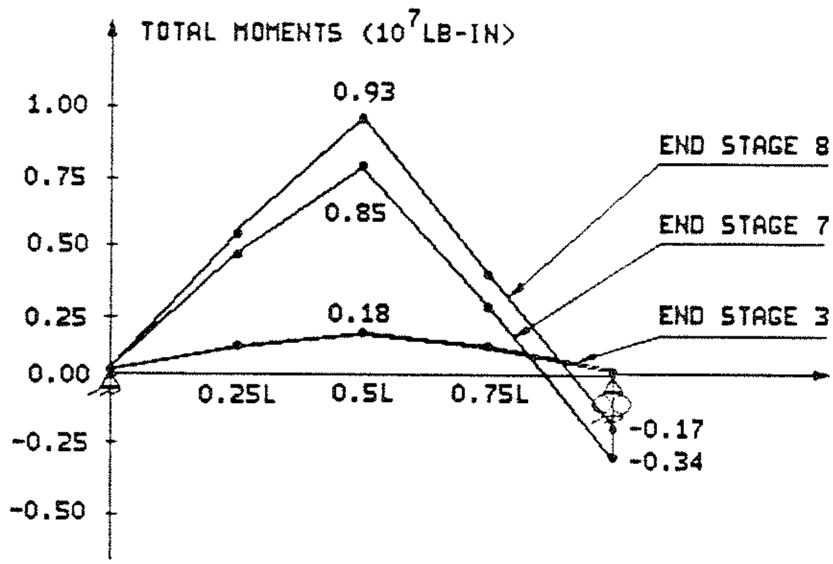


Fig. 5.5.2.5 BENDING MOMENT PROFILES

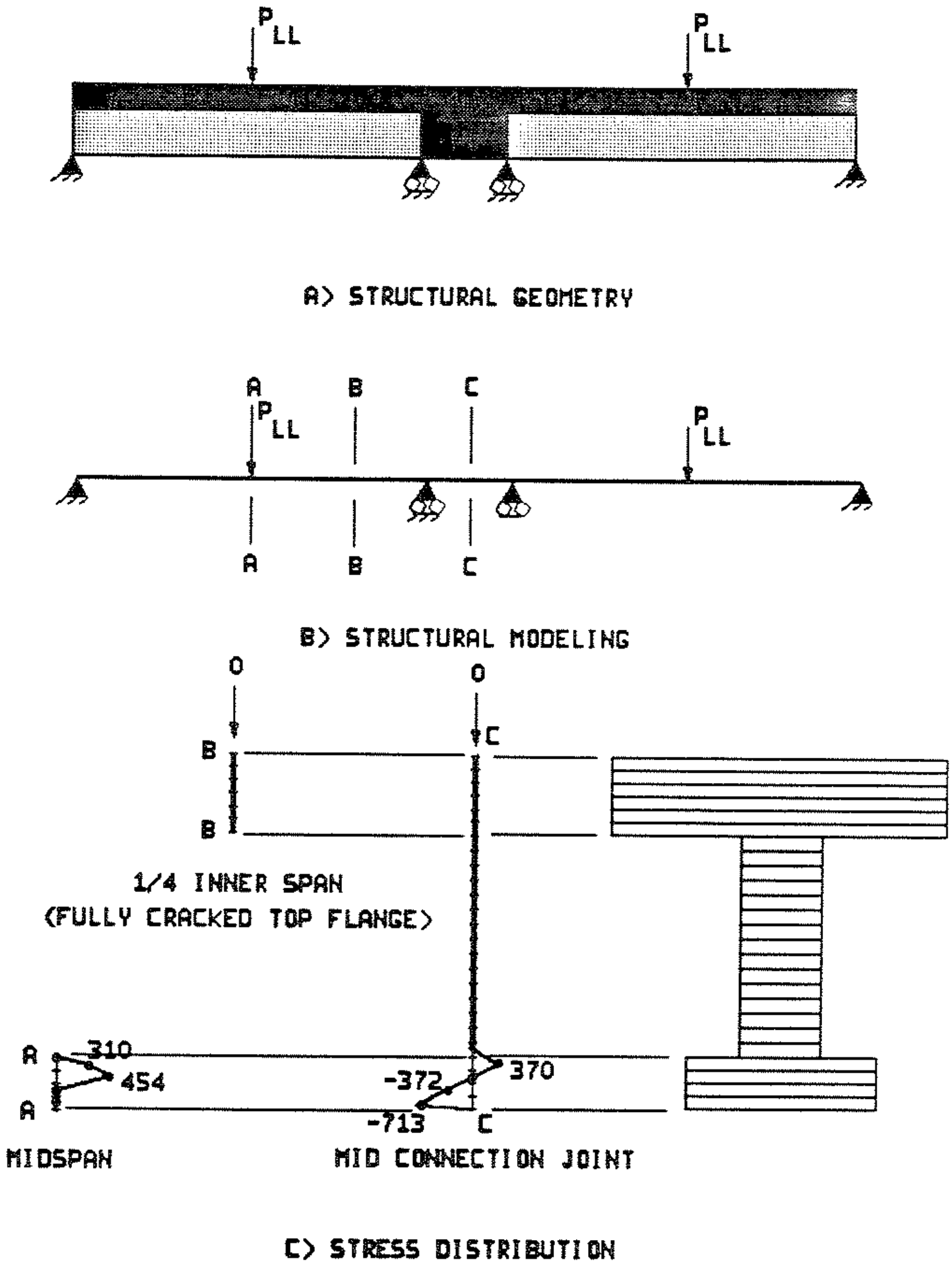


Fig. 5.5.2.6 CONCRETE STRESS DISTRIBUTION END STAGE 8

located between the free end support and the midspan location is noted. However, the redistribution is more significant in the vicinity of the inner support and is considerable (50 %) for the negative support moment. Further investigation **Fig. 5.5.2.6** reveals that the cross section of the short link connection is almost completely cracked in tension at maximum total live load. The top flange and the web have completely cracked and the the flow of stresses between the link and the adjacent spans is done through the reinforcing steel only. Consequently, the degree of continuity that this connection provides is questionable and prestressing steel used in the top deck over the connection would probably be desirable. The degree of continuity (low) may be also noted from the proportion (low) of the incremental total inner support moment to the incremental midspan moment when the structure is subjected to the incremental live load.

Tensile cracking of concrete has been also noted in the lower flange at midspan and the top flange at the quarter inner span **Fig. 5.5.2.6**.

In relation to prestressing, **Fig. 5.5.2.7** presents the prestressing force losses at midspan and at the inner support locations in terms of the applied incremental live load value. For all practical purposes, a 25 % value for losses may be assumed representative.

CHAPTER 6 : Summary and Conclusions

Segmental erection across the depth of one dimensional frame element cross-sections in three dimensional structures together with nonlinear material and time dependent analysis has been presented in this study. An original and rational procedure based on the finite element approach has been described. This research represents an extension of the work by Ketchum [27] for segmental linear analysis of planar frames and by Mari [25] for nonsegmental nonlinear analysis of three dimensional frames.

The computer program **SPCF3D** developed in the present study is a general purpose program that traces the nonlinear (arbitrary) material behaviour of segmentally erected prestressed concrete composite three dimensional frame structures. The finite element used to model the structural element is an incremental one dimensional beam element based on improved Bernoulli-Euler kinematics. The filamented cross-section is assumed to be constant along the length of the structural element and allow composite, segmental erection across its depth. Time dependent effects such as creep, shrinkage, temperature and relaxation are considered automatically. The mathematical modeling of creep and the recursive algorithm adopted by Ketchum [27] have been maintained. Precast or cast in place structural segments, pretensioned or post-tensioned prestressing tendons, traveling formworks in addition to force and displacement boundary conditions may be modeled within any statically feasible construction sequence.

Numerical examples obtained with the computer program **SPCF3D** developed for this study have been presented to determine the validity of the theory on which the incremental beam element is based as well as the accuracy of the implementation and the various features and capabilities of the computer program. The computer program **SPCF3D** has been shown to be a useful tool in the analysis of reinforced and prestressed concrete composite three dimensional frame structures (bridges).

Extensions are needed to improve correlations and accuracy for nonlinear analysis of box girder bridge examples presented. Among these extensions, a displacement control strategy that allows the analysis to continue at low and even negative structural stiffness is needed. A nonlinear torque-twist relationship is another option that is needed especially if it can account for coupling between bending and torsional response. The inclusion of shear deformations as well as transverse bending and shear distortion of the cross section are also desirable additions.

As a next major step, the concepts developed in this study for segmental analysis acrosss the depth and length of one dimensional elements in three dimensional frames should be extended to macro box girder elements such as that used by Choudhury [30] so that three dimensional box girder bridges can be analyzed more accurately to include segmental nonlinear material, geometric and time dependent effects.

REFERENCES

1. Kaar, P. H., Mattock, A. H. "Precast Prestressed Concrete Bridges", Portland Cement Association Journal, Vol. 2, No. 3, Sept. 1960, pgs. 52-78.

2. Hanson, N. W. "Precast Prestressed Concrete Bridges", Portland Cement Association Journal, Vol. 2, No. 2, May 1960, pgs. 38-58.

3. Kaar, P. H., Mattock, A. H. "Precast Prestressed Concrete Bridges", Portland Cement Association Journal, Vol. 3, No. 3, Sept. 1961, pgs. 52-78.

4. Warner, R. F. "Biaxial Moment Thrust Curvature Relations", Journal of the Structural Division, ASCE, No. ST-5, May 1969, pp. 6564-6570.

5. Freyermuth, C. L. "Design of Continuous Highway Bridges with Precast, Prestressed Concrete Girders", Journal of the Prestressed Concrete Institute, April 1969, pp. 14-39. Also reprinted as PCA Engineering Bulletin No. EB 014.01E , Portland Cement Association, Skokie, Illinois, August 1969.

6. ACI Committe 209, "Prediction of Creep, Shrinkage and Temperature Effects in Concrete Structures", ACI Publication SP-27, Paper SP 27-3, 1970.

7. Becker, J. and Bresler, B. "FIRES-RC, A Computer Program for th Fire Response of Structures - Reinforced Concrete Frames ", Report No. UCB FRG 74-3, University of California, Berkeley, August 1974.

8. Kabir, A. F. and Scordelis, A. C. "Computer Program for Curved Bridges on Flexible Bents", Structural Engineering and Structural Mechanics Report No. UC SESM 74-10, University of California, Berkeley, September 1974 (PB 242 470/AS), 170 pgs.

9. Hernandez, H. D. and Gamble, E. L. "Time dependent Prestress Losses in Pretensioned Concrete Construction", Structural Research Series No. 417, Civil Engineering Studies, University of Illinois, Urbana, May 1975.

10. Kang, Y. J. "Nonlinear Geometric, Material and Time Dependent Analysis of Reinforced and Prestressed Concrete Frames", Ph.D. Dissertation, Division of Structural Engineering and Structural Mechanics, Univerity of California, Berkeley, Report No. UC-SESM 77-1, January 1977, 243 pgs.

11. Van Zyl, S. F. "Analysis of Curved Segmentally Erected Prestressed Concrete Box Girder Bridges", Ph.D. Dissertation, Structural Engineering and Structural Mechanics Report No. UC SESM 78-2, University of California, Berkeley, January 1978, 265 pgs.

12. Suttikan, C. "A Generalized Solution for Time-Dependent Response and Strength of Noncomposite and Composite Prestressed Concrete Beams", Ph.D. Thesis, The University of Texas at Austin (1978), 350 pp.

13. Comite Euro-International du Beton "CEB-FIP Model Code for Concrete Structures, Cement and Concrete Association, Wexham Springs, Slough, England, 1978.

14. Scordelis, A. C., Elfgren, L. G. and Larsen, P. K. "Time Dependent Behaviour of Concrete Box Girder Bridges", Journal of the American Concrete Institute, Title No., 76-9, January 1979, pp. 159-177.

15. Van Zyl, S. F. and Scordelis, A. C. "Analysis of Curved Prestressed Segmental Bridges", Journal of the Structural Division, Proceedings of the American Society of Civil Engineers , Volume 105, No. ST-11, November 1979, pp. 2399-2417.

16. Kang, Y. J. and Scordelis, A. C. "Nonlinear Analysis of Prestressed Concrete Frames", Journal of the Structural Division, Proceedings of the American Society of Civil Engineers, Vol. 106, No. ST-2, February 1980, pp. 445-472.

17. Sorensen, S. I. and Scordelis, A. C. "Computer Program for Curved Prestressed Box Girder Bridges", Structures and Materials Research Report No. UC SESM 80-10, Department of Civil Engineering, University of California, Berkeley, December 1980, 67 pgs.

18. Sorensen, S. I. and Scordelis, A. C. "Computer Program for Curved Prestressed Slabs", Structures and Material Research Report No. UC SESM 80-11, Department of Civil Engineering, University of California, Berkeley, December 1980, 66 pgs.

19. Ketchum, M. A. and Scordelis, A. C. "Nonlinear Analysis of a Prestressed Concrete Bridge", Final Report, IABSE Colloquium on Advanced Mechanics of Reinforced Concrete, Delft, June 1981, pp. 561-576.

20. Seible, F. "Nonlinear Analysis and Ultimate Strength of Multi-Cell Reinforced Concrete Box Girder Bridges", Ph.D. Dissertation, Structural Engineering and Structural Mechanics Report No. UC SESM 82-2, University of California, Berkeley, February 1982, 228 pgs.

21. Scordelis, A. C. "Berkeley Computer Programs for the Analysis of Concrete Box Girder Bridges", Proceedings, NATO-Advanced Study Institute on Analysis and Design of Bridges, Cesme, Turkey, July 1982, pp. 1-71.

22. Seible, F. and Scordelis, A. C. "Nonlinear Behaviour and Failure Analysis of Multi-CELL RC Box Girder Bridges", Proceedings, International Conference on Short and Medium Span Bridges, Toronto, Canada, Vol. 1, August 1982, pp. 391-405.

23. Scordelis, A. C. and Seible, F. "Time Dependent Behaviour of a Skew Reinforced Concrete Box Girder Bridge", American Concrete Institute, Concrete International: Design and Construction, Vol. 5, No. 1, January 1983, pp. 82-92.

24. Seible, F. and Scordelis, A. C. "Nonlinear Analysis of Multi-Cell RC Box Girder Bridges", Journnal of Engineering Structures, Vol. 5, January 1983, pp. 45-57.

25. Mari, A. R. "Nonlinear Geometric, Material and Time Dependent Analysis of Three Dimensional Reinforced and Prestressed Concrete Frames", Division of Structural Engineering and Structural Mechanics Report No. UCB SESM 84-12, University of California, Berkeley, June 1984, pgs.

26. Scordelis, A. C., Chan, E. C., Ketchum, M. A. and Van der Walt, P. P. "Computer Programs for Prestressed Concrete Box Girder Bridges", Structural Engineering and Structural Mechanics Report No. UC SESM 85-02, Department of Civil Engineering, University of California, Berkeley, March 1985, 346 pgs.

27. Ketchum, M. A. "Redistribution of Stresses in Segmentally Erected Prestressed Concrete Bridges", Ph.D. Dissertation, Structural Engineering and Structural Mechanics Report No. UCB SESM 86-07, Department of Civil Engineering, University of California, Berkeley, May 1986, pgs.

28. Oesterle, R. "Inelastic Analysis for In-Plane Strength of Reinforced Concrete Shear Walls", Ph.D. Dissertation, Department of Civil Engineering, Northwestern University, Evanston, Ill., June 1986, pgs.

29. Glikin, J. D. and Oesterle, R. G. "Creep and Shrinkage Analyses of Simple Span Precast, Prestressed Bridge Girders Made Continuous", Portland Cement Association, Skokie, Illinois, July 1986, 11pgs.

30. Choudhury, D. "Analysis of Curved Non-Prismatic Reinforced and Prestressed Concrete Box Girder Bridges", Ph.D. Dissertation, Structural Engineering and Structural Mechanics Report No. UCB SESM 86-12, Department of Civil Engineering, University of California, Berkeley, December 1986, pgs.

31. Kasti, F. A. "Time Dependent Linear Analysis of Segementally Erected Composite Prestressed Concrete Bridges", Structural Engineering, Mechanics and Material Research Report UCB 1987, Department of Civil Engineering, University of California, Berkeley.

32. Kristek, V. and Smerda, Z. "Creep and Shrinkage of Concrete Elements and Structures", SNTL-Publishers of Technical Literature, Prague 1988.

33. Ghali, A. and Elbadry, M. "Serviceability Design of Continuous Prestressed Concrete Structures", PCI journal, January-February 1989, pp54-91.

34. Kang, Y. J. "Program for Nonlinear Segmental Analysis of Planar Prestressed Concrete Frames", Structural Engineering and Structural Mechanics Report No. UCB SESM 89-07, Department of Civil Engineering, University of California, Berkeley, July 1989, pgs.

35. Millanes, F. and Calzon, M. "Automatic Evolutive Analysis of Complex Composite Bridges and Systems", Journal of Structural Engineering, Vol. 115, No. 10 , October 1989, pp 2548-2565.

36. Oesterle, J. D., Glikin J. D. and Larse, S. C. "Design of Simple Span Precast Prestressed Bridge Girders Made Continuous", Transportation Research Board, National Research Council, NCHRP 322, Washington D.C., November 1989.

APPENDIX A

In the present appendix, a detailed desciption of the command syntax related to **SPCF3D** user's input manual is given. The expressions that appear in bold face are either meant to stress a specific idea or to indicate that a partial/full modofication has been introduced at that location relative to **SFRAME** [27] input manual. The latter document has been used as a basis for the present **SPCF3D** input manual.

APPENDIX A : SPCF3D USER'S GUIDE

SPCf3D:A COMPUTER PROGRAM FOR THE NONLINEAR MATERIAL AND TIME DEPENDENT ANALYSIS OF SEGMENTALLY ERECTED REINFORCED AND PRESTRESSED CONCRETE COMPOSITE THREE DIMENSIONAL FRAME STRUCTURES

By Fouad A.-K. Kasti, University of California, Berkeley, May 1990.

Table of Contents

1. Overview

SPCF3D is a special purpose structural analysis program, specifically developed for the time dependent **nonlinear material** analysis of segmentally erected prestressed **and reinforced** concrete **composite three dimensional** frame structures and bridges. The frame structure is modeled as a system of frame elements and prestressing tendon elements connected at nodes. Time is divided into a number of time steps and the program computes the response of the structure at the end of each of these time steps.

Capabilities

The program can "build the structure in the computer" using any statically feasible construction sequence for the **three dimensional** frame. **The program allows segmental operations along the length of the structure as well as across the depth of the frame elements**. Frame and tendon elements may be installed and removed at any time step during the solution, **the only restriction being that all frame elements belonging to a given superelement are removed whenever one frame element is removed**. Prestressing tendons may be stressed from one or both ends and may be subsequently restressed. Traveling formwork may be moved to any location. Nodal loads may be applied or removed at any time step. Nodal boundary conditions may be changed from fixed to free or free to fixed state at any time step. All actions are considered to occur linearly (gradually) over the duration of the time step. Thus any instantaneous action must be modelled with a zero length time step.

The solution includes the effects of creep, shrinkage and aging of the concrete, plus friction, anchorage slip and relaxation of the prestressing steel. **Nonlinear material relationship for concrete, mild steel and prestressing steel is included**. The theories used and their numerical implementations are discussed thoroughly in the main body of this report. Material constitutive parameters may be computed by the program to model the time dependent material properties according to laboratory measurements or the ACI or CEB/FIP recommendations.

Output can include incremental and total nodal displacements and reactions, frame element stresses and stress resultants, prestressing tendon forces and resultants. Numerous options exist for the specification of the type of output desired during any solution or output phase.

Preparation of Input Data

Input data are provided to the program from the terminal or from a deck of cards or from an input files or card images depending on the computer system used. All input takes the form of Commands consisting of one or more keywords, followed by optional numerical data required for the execution of the command. The actions and the data requirements of the commands are summarized starting on page A-4 and described in detail starting on page A-8. All keywords have six significant characters and must be in capital letters.

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, 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 by 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 arithmetic expressions 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 or C=200*12+3.5, 400/12,10+20/5-2

These arithmetic expressions are evaluated from left to right without operator hierarchy. The expression 10+20/5-2 is evaluated as (((10+20)/5)-2). Blank spaces within arithmetical expressions 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 user 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 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.

Disclamer

Considerable effort has gone into the development and testing of this program. It can analyse 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 authors assume no responsibility in this respect.

2. COMMAND SUMMARY

The following brief descriptions of each command and its input data requirements are intended as a summary of **SPCF3D**'s command syntax and as a quick reference quide for the experienced user. Detailed descriptions of each command and the meaning of all data are provided in section 3 of this User's Guide.

COMMAND, DATA SUMMARY and DESCRIPTION

Problem Initialization Input :

START R=rstrt **I=iscren**---> Start interpreting the input data

TITLE ---> Print subsequent text lines as title

CONCRETE PARAMETERS N=ncpt T=napt,ntpt,nrpt P=iprn ---> Generate creep coefficients

n M=model N=nact,ntct,nrct R=rett G=igen ... ---> Control Data

A=age1 **F=fult** S=shrn T=time(1),...,time(ntct) ---> Optional parameters

C=crep(1),...,crep(ntct) ---> Creep generation input

Structure Definition Input :

MESH INPUT ---> Start structure definition

NODES N=nnod ---> Input node coordinates

n X=xord Y=yord **Z=zord** C=xord,yord,**zord** S=scale G=n1,n2,inc

SEQUENCE ---> Input node order for DOF numbering

G=n1,n2,inc

CONCRETE PROPERTIES N=ncnc ---> Input concrete material properties

n F=f'_c C=ultcrp S=ultshr W=weight M=model A=alfa **D=mcse N=poison**

MATERIAL MODELS N=ncse,nmse,npse P=maxc,maxm,maxp ---> Input material stress-strain relationships properties

n M=CONC F=scal I=iflg C=prat,urat E=peps,ueps R=mtr1,mtr2 N=cpnt ---> for con-

crete material law, or

n M=mild F=scal I=iflg N=mten S=sigy E=epsy C=acte,knee P=sig0,eps0 ---> for mild steel material law, or

n M=PRES F=scal N=mpse ---> for prestressing steel

N=mpnt C=sigm,epsl ---> Discrete points input

LOCAL SYSTEM N=nloc ---> Local coordinate systems specification

n K=k1,k2,k3 J=j1,j2,j3 N=n1 R=r1,r2,r3,r4,r5 G=g1,g2,ginc X=x1,x2,x3

Y=y1,y2,y3 Z=z1,z2,z3

TSECTION PROPERTIES N=nstr R=nrec ---> Input traveler cross-section properties

mstr R=mrec U=y(1),...,y(mrec) V=z(1),...,z(mrec) A=area

Y=dely(1),...,dely(mrec) Z=delz(1),...,delz(mrec)

I=inyy,inzz,inyz,tors C=ycen,zcen

FSECTION PROPERTIES N=nsct M=maxy,maxz P=iprn ---> Input frame cross-section properties

msct M=ymax,zmax C=ycrd,zcrd Y=ycrd Z=zcrd D=dely,delz S=oshr,yshr,zshr

N=mfil,inty,intz G=gen1,gen2 S=dely,delz C=binc,dinc B=fbas,finc I=cinc

F=iflg ---> input filaments geometry

FRAME ELEMENTS N=nfml ---> Define frame elements in the structure

mfml,nodi,nodj M=modl C=mcnc,mcxs S=mmld,mmxs L=locl D=cday

G=gen1,...,gen6 J=jen1,...,jen5

N=msup,mfrm G=g1,g2,g3,g4 ---> Input frame superelements specification

MILD STEEL PROPERTIES N=nmsl ---> Input mild steel properties

n A=alfa D=mmse W=wegt N=poison

PRESTRESSING STEEL N=npsl ---> Input prestressing steel properties

n C=crvf W=wblf R=rlxp A=alfa M=mpse S=smax Y=syld

TENDON GEOMETRY N=ntdn P=iprn ---> define tendons in the structure

n S=nspn M=matl A=area F=iflg ---> Tendon control data

ns N=nnis G=n1,n2,inc O=ox,oy,oz R=rx,ry,rz S=sx,sy,sz ---> span control line

T=tx,ty,tz L=lod F=iflg J=j1,j2,j3,j4

L=n(1),...,n(nnis) --> Input nodes in span if generation has not been used

R=rlli(s),rllp(s),rlri(s),rlli(t),rllp(t),rlli(t) S=slt,slp,srt T=tlt,tlp,trt --> Tendon points internally generated or

np R=r(np) S=s(np) **T=t(np) C=r(np),s(np),t(np)** --> Direct input of tendon points

TRAVELERS N=ntrv

mtrv X=mstr E=emod W=wegt N=nnit **L=lod**

MESH COMPLETE

Set Ambient Conditions :

SET D=day T=temp G=gx,gy,gz,gxz **C=tult,iutl,astl,istl,tptl,adtl,attl**

A=aclr I=mxit **N=uayz** --> Set solution parameters

Construction Operations :

CHANGE STRUCTURE --> Start construction operation for this step

RESTRAINTS --> Change nodes boundary conditions

n1,n2,inc R=rx,ry,**rz,rxx,ryy**,rzz

BUILD N=n1,n2,inc D=cd --> Install frame elements

REMOVE N=n1,n2,inc --> Remove frame elements

STRESS N=n1,n2,inc R=ra,rb S=sa,sb F=fa,fb D=da,db P=ipm ---> Stress tendon elements

MOVE N=mtrv D=n(1),...n(nnit) **C=xcny,xcnz** ---> Move traveler to destination nodes

CHANGE COMPLETE --> End construction operations for this step

External Loading Commands :

LOADING --> Specify load increment

N=n1,n2,inc F=fx,fy,**fz,fxx,fyy**,fzz D=dx,dy,**dz,dxx,dyy**,dzz --> Nodal loads and displacements increment

L=l1,l2,inc F=fx,fy,**fz T=Opoint,Ypoint,Zpoint** --> Element loads and Temperature

Solution Command :

SOLVE S=nstp D=date P=iprn I=mxit A=aclr U=istf G=igss --> Analyze the structure under the present configuration

Output Commands :

OUTPUT P=iprn --> Output total structure response

CAMBER C=idof --> Output adjusted displacements

Problem Termination Command :

STOP ---> Stop execution and save database for restart

Output Filaments Data Record :

RECORD --> Output data record parameters

FRAME ELEMENTS L=mfml --> Output frame element parameters

FILAMENTS L=mfml I=indx F=iflg --> Output frame element filament parameters

TENDON ELEMENTS L=mtdn S=mseg --> Output tendon element parameters

TRAVELER ELEMENTS L=mtrv S=mseg --> Output traveler element parameters

RECORD COMPLETE --> End data record parameters output

3. 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 (i.e a command such as TRAVELER N=0 must not be used). Default values for input quantities, whenever applicable, are indicated by [?] in the descriptions. [pv] indicates a default value of the previous value entered.

The START Command

Syntax : START R=rstrt **I=iscren [0]**

where : rstrt = Data restore flag (1= restore data base , 0= do not) [0]

iscren = Screen output file dump flag (0 screen output, 1= screen output to file)

Explanation : 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 the database will be restored.

The TITLE Command

Syntax : TITLE

Problem ID text line 1

. . . .

Problem ID text line ...

Explanation : **The TITLE command prints a program identifier in the output file and then prints all subsequent lines of text provided immediately following the command line until a blank line is reached.**

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

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

THE CONCRETE PARAMETERS Command

Syntax : CONCRETE PARAMETERS N=ncpt T=napt,ntpt,nrpt P=iprn

(Concrete Parameters Specifications)

where : ncpt = Total number of concrete parameter types [1]

napt = Maximum "loading ages" in parameter tables [32]

ntpt = Maximum "times after loading" for parameter tables [48]

nrpt = Maximum "retardation times" in parameter table [4]

iprn = Output print flag [1]

Explanation : The CONCRETE PARAMETERS command generates tables of shrinkage strain, elastic modulus 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 analysis of the structure. The values in the tables are normalized for ultimate creep coefficient = 1, ultimate shrinkage strain = 1 and concrete strength $f'_c(28 days) = 1$. **Prior to use in each solution step, these normalized values are scaled on one hand by the creep coefficient, shrinkage strain and ultimate strength values input under the CONCRETE PROPERTIES subcommand of the MESH INPUT command described below and on the other hand by the modulus of elasticity E(28 days) value retrieved from the corresponding concrete material stress-strain diagram.** Thus under most circumstances only one concrete parameter type is required.

The quantity of output is controlled using the P=? identifier. For P=0, only an echo print of the input data is output. For P=1, a summary of the 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 behaviour 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 : (Repeated for each concrete parameter type)

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 N=nact,ntct,nrct R=rett G=igen ...

where : 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

nact = Number of loading ages in this current table [napt]

ntct = Number of times after loading for this current table [ntpt]

nrct = Number of retardation times in this table [nrct]

rett = Smallest retardation time for creep model (days) [5]

igen = Age and time 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 additional data are interpreted from the control line :

A=a B=b C=c D=d E=e F=f S=s T=r W=w

where : a [4] and b [0.85] are used in the function for f'_c (t) :

$$f'_c(t) = f'_c(28) [t / (a+b^*t)]$$

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

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

e [1], f [50] and s [7] are used in the time function for shrinkage :

$$\epsilon^s(t) = \epsilon^s(\infty)(t-s)^e / [f + (t-s)^e]$$

r [0.80] and w [150] are the tensile strength factor and the unit weight of concrete respectively used in the following equations. The modulus of elasticity from the second equation is used only for creep coefficients computations :

$$f'_t(t) = r^* \sqrt{(w^* f'_c(t))}$$

and

$$E(t) = 33.0 * w^{1.5}[f'_c(t)]^{1/2}$$

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

H=humd U=thik D=temp

where : humd is the ambient relative humidity (percent) [70]

thik is the notional thickness (cm) [30]

temp is the temperature (degrees C) [20]

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 ... nact, one additional line of input in the following format is required :

A=age1 T=time(1),...,time(ntct)

where : age1 = Age at loading (days) for table generation

time(?) = Observation times (days) for table generation

When M=LAB is specified, the loading ages, observation times, **ultimate strength**, shrinkage strains and creep strains are not generated by the program and must be input from the following lines. For each loading age 1 ... nact, two additional lines of input in the following format are required :

A=age1 **F=fult** S=shrn T=time(1),...,time(ntct)

C=crep(1),...,crep(ntct)

where : age1 = age at loading (days) for table generation

fult = Ultimate compressive strength F(age1)

shrn = Shrinkage strain ϵ(age1)

time(?) = Observation times (days) for creep strains

crep(?) = Creep strains corresponding to age1 and time

The MESH INPUT Command

Syntax : MESH INPUT

(Mesh Input Subcommands)

Explanation : 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, frame and tendon element geometries for the **three dimensional 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 (node), BUILD (frame element), REMOVE (frame element), STRESS (tendon) and MOVE (traveler) subcommands of the CHANGE STRUCTURE command.

The MESH INPUT subcommands include NODES, SEQUENCE, **LOCAL SYSTEM, FSECTION PROPERTIES, TSECTION PROPERTIES**,CONCRETE PROPERTIES, MILD STEEL PROPERTIES, FRAME ELEMENTS, PRESTRESSING STEEL, TENDON GEOMETRY, TRAVELERS, and MESH COMPLETE described below.

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

The NODES Subcommand

Syntax : Nodes N=nnod

n X=xord Y=yord **Z=zord** C=Xord,yord,**zord** S=scale G=n1,n2,inc

where :

nnod = Total number of nodes in the structure

n = Node number (1 =< n =< nnod)

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

yord = Global Y-coordinate of node n1 [0,pv]

zord = Global Z-coordinate of node n1 [0,pv]

scale = Scale factor for global X-,Y- and Z- coordinates [1,pv]

n1,n2,inc = Generation parameters described blow.

Explanation : 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-, Z-) coordinate system. The input X-, Y- and Z- node coordinates are multiplied by the scale factor immediately after input. Once the scale factor is entered (usually on the first line of the 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 at which generation starts

n2 = A previously specified node number at which generation ends

inc = Node number increment defining the generated nodes [1]

If the 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

Syntax : SEQUENCE

G=n1,n2,inc

where : n1 = First node number in a generation sequence

n1 = Last node number in a generation sequence

inc = Node number increment [1]

Explanation : 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. Skillfull 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 numerical order 1 ... 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 node which is 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

Syntax : CONCRETE PROPERTIES N=ncnc

n F=f'_c C=ultcrp S=ultshr W=weight M=model A=alfa **D=mcse N=poisson**

where : ncnc = Total number of concrete types [1]

n = Concrete type number (1 =< n =< ncnc)

f'_c = Ultimate compressive strength (28days) [0]

ultcrp = Ultimate creep coefficient [0]

ultshr = Ultimate shrinkage coefficient [0]

weight = Unit weight (for gravity dead load only) [0]

model = Time dependent model number (1 =< model =< ncpt) [1]

alfa = Thermal expansion coefficient [0]

mcse = Concrete material stress-strain diagram number [1] (1 =< mcse =< ncse)

poisson = Poisson' ratio [0.33]

Explanation : The CONCRETE PROPERTIES command is used to specify the different basic concrete material properties found in the structure. The concrete type number n must be less or equal to the total number of concrete types input on the CONCRETE PROPERTIES command line. The concrete types may be supplied in any order; however, each concrete type must be specified once and only once.

Only the simple descriptive properties listed above are input under this command. The complete numerical description of the concrete 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.

A table of concrete properties is printed in the output.

The MATERIAL MODELS Subcommand

Syntax : MATERIAL MODELS N=ncse,nmse,npse P=maxc,maxm,maxp

(Material Models Specification)

where : ncse = Total number of concrete stress-strain diagrams [1]

nmse = Total number of mild steel stress-strain diagrams [1]

npse = Total number of prestressed concrete stress-strain diagrams [1]

maxc = Maximum number of discrete points in any concrete stress-strain diagram excluding origin [1]

maxm = Maximum number of discrete points in any mild steel stress-strain diagram tensile or compressive side and excluding origin [1]

maxp = Maximum number of discrete points in any prestressing steel stress-strain diagram excluding origin [1]

Explanation : The MATERIAL MODELS subcommand is used to specify the shape of the concrete, mild steel and prestressing steel material laws. The relationship need not be linear and is totally arbitrary. The number of discrete points needed to define the concrete (maxc) and mild steel (maxm) stress-strain diagrams are simply the number of points in the compression and tension quadrant respectively, excluding the origin. In the case of mild steel, the program can handle load reversal assuming point symmetry about the origin of the stress-strain diagram. The user needs to specify the number of points in the tension or compression side only in this case.

Material Model Specification :

The first data line for each material model is a control data line which takes the following form. If the discrete points are internally generated using a built in material law, this is the only line required for the material type, otherwise additional lines are needed as described below.

For concrete : n M=CONC F=scal I=iflg C=prat,urat E=peps,ueps R=mtr1,mtr2 N=cpnt

For mild steel : n M=MILD F=scal I=iflg N=mten S=sigy E=epsy C=acte,knee P=sig0,eps0

For prestressing : n M=PRES F=scal N=mpse where : n = Concrete, mild steel or prestressing steel model number. The model number tags the material number with respect to the total number of possible material types of the same kind and need not be in ascending order. For concrete (1 =< n =< ncse), for mild steel (1 =< n =< nmse) while for prestressing steel (1 =< n =< npse) [0]

scal = Scaling factor by which the input or generated stress values are multiplied [1]. Once scal has been input, it need not be input again unless it is necessary to reset its value

iflg = input data flag indicator. For concrete, 1= Modified Kent and Park, 2= discretized input [1]. In the event iflg = 1 is chosen, all identifiers (M=?, F=?, I=?, C=?, E=?, R=? and N=?) are needed, while for the discretized input option only M=?, F=?, I=?, R=? and N=? identifiers are needed. For mild steel, 1= Ramberg Osgood, 2= discretized input [1]. In the former case, all identifiers are needed while in the latter only M=?, F=?, I=?, N=?, S=?, E=? are. Once the iflg value has been input, it may not be input again unless to reset its value

prat = Peak stress ratio value [1.00]

peps = Peak strain value corresponding to prat [0.003]

urat = Ultimate stress ratio value [0.85]

ueps = Ultimate strain value corresponding to urat [0.006]

cpnt = Number of points required to discretize the parabolic portion of the Kent and Park relationship excluding the end points in the event Kent and Park representation is chosen. If the discretized input option is used however, the cpnt is the total number of discrete points needed [ncse].

mtr1 = Material stress ratio with respect to peak stress defining the limit at which the linear stress-creep relationship is still valid [0.85]. Beyond mtr1, magnification in stress is needed to compute an equivalent stress to be used in the stress-creep relationship. Hence mtr1 may be considered as a lower bound stress amplification range delimiter.

mtr2 = Upper bound material stress ratio with respect to peak stress. The mtr2 is an upper bound stress amplification range delimiter.

mten = Number of points needed to define the tension (or compressive) side of the mild steel diagram excluding the origin

sigy = Yield stress of mild steel needed in point symmetry operation to copy the other half of the mild steel relationship

epsy = Yield strain of mild steel corresponding to sigy

acte, knee, sig0, eps0 = Parameters needed to define the Ramberg-Osgood curve. These are the A constant, knee factor, sigma(0) and epsilon(0) repectively [60000,0.002,0.0,1.00]

mpse = Number of prestressing steel discrete points defining the tension side of the prestressing steel stress-strain diagram [npse]

Additional data lines are needed in the case the discretized input option is used for concrete, mild steel and prestressing steel material stress-strain laws. One line per point is needed in the following format, no point may be input more than once and the slope of any segment joining any two consecutive points should never be zero, otherwise an error message will be displayed and the execution of the program is terminated :

N=mpnt C=sigm,epsl

where : mpnt = Discrete point number (1 =< mpnt =< mcse, mten or mpse)

sigm = Stress ordinate of point mpnt

epsl = Strain ordinate of point mpnt

The LOCAL SYSTEM Subcommand

Syntax : LOCAL SYSTEM N=nloc

where : nloc = Total number of distinct local coordinate systems [1]

Explanation : The LOCAL SYSTEM subcommand is needed to define the different local coordinate systems with respect to the global reference system. The definition of any of these systems requires the specification of the directions of the local coordinate axes belonging to the system besides its origin. In the former case, the direction may be determined either by two points on each of the local axes or by the three global components. The user need specify the direction of two of the three local axes since the program will automatically find out the third direction from right handed rule requirement on the local coordinate system. In summary, the user has the choice among different options to specify the local coordinate system and the program is equipped with generation capabilities to facilitate such an input. If no identifier is used on the following line (i.e. n only) the default value for X=?, Y=? and Z=? is used which corresponds to the global reference system directions.

n K=k1,k2,k3 J=j1,j2,j3 N=n1 R=r1,r2,r3,r4,r5 G=g1,g2,ginc

X=x1,x2,x3 Y=y1,y2,y3 Z=z1,z2,z3

where : n = Local coordinate system number (1 =< n =< nloc) [0]

k1,k2,k3 = Three nodal point numbers needed to define the local x- axis (k1,k2) and the local (x,y) plane [0,0,0]

j1,j2,j3 = Three nodal point numbers needed to define the local x- axis (k1,k2) and the local (x,z) plane [0,0,0]

n1 = Flag indicator used with the X=?, Y=? and Z=? identifiers. 1= local x-axis direction is missing, 2= local y- is missing and 3= local z- is missing. Once n1 is input do not reenter its value again unless it is necessary to reset it.

r1 = First node number in the curvilinear generation sequence [0]

r2 = Last node number in the curvilinear generation sequence [0]

r3 = Node number increment in the curvilinear generation sequence [0]

r4 = Node number of the center nodal point used in the curvilinear generation [0]

r5 = Flag indicating whether K (r5 = 0) or J (r5 = 1) option is to be used in the generation sequence. [0]

g1 = First local coordinate system number in the generation sequence [1]

g2 = Local coordinate system increment in the generation sequence [1]

ginc = Local coordinate system number increment [1].

x1,x2,x3 = Global X-,Y- and Z- components respectively of a vector lying along the local x- axis. [1,0,0]

y1,y2,y3 = Global X-,Y- and Z- components respectively of a vector lying along the local y- axis. [0,1,0]

z1,z2,z3 = Global X-,Y- and Z- components respectively of a vector lying along the local x- axis. [0,0,1]

Explanation : The local coordinate system axes directions may be specified according to one of the following four options :

(a) The global X-, Y- and Z- components of two of the local coordinate system axes are given. Here n, X=?, Y=?, Z=? and N parameters are needed only. The program finds automatically the components of the third local coordinate system axis unit vector

(b) The generation capabilities of the routines handling the local coordinate system operations may be invoked and all is needed in this case are the n, R=? and G=? parameters. The directions of the local coordinate system are computed based on the global coordinates of the nodal points input under the R=? and G=? identifiers. Curvilinear generation of the local coordinate systems is performed and two directions are used from such generation in defining the local coordinate system. The user is required to choose between a K or J type definition as is the case for K=? and J=? identifiers respectively (see below) . One of the two directions are determined by the end nodes r4 and one of the nodes in the curvilinear sequence while the other direction is defined by two consecutive nodes in the same sequence.

(c) The third option (K option) available consist in defining one direction by assigning two nodes located on that axis (k1 and k2) while the nodes k1,k2 and k3 defines the local x-y plane. The program computes a third direction based on the vectors (k1,k2) and (k1,k3) and then updates the second direction based on the first (k1,k2) and the computed vector in order to satisfy right handed coordinate system requirement. Here only n and K=? are needed

(d) The last option (J option) is similar to the K option except for the fact that the nodal points j1,j2 and j3 defines the local x-z plane.

The TSECTION PROPERTIES Subcommand

Syntax : TSECTION PROPERTIES N=nstr R=nrec

mstr R=mrec U=ycrd(1),...,ycrd(mrec) V=zcrd(1),...,zcrd(mrec)

Y=dely(1),...,dely(mrec) Z=delz(1),...,delz(mrec) A=area

I=inyy,inzz,inyz,tors C=ycen,zcen

where : nstr = Total number of traveler section types [1]

nrec = Maximum number of rectangles any traveler section might have [3]

mstr = A given traveler section number (1 =< mstr =< nstr) [0]

ycrd(i) = Local y of coordinate rectangle number i with respect to traveler reference coordinate system [0]

zcrd(i) = Local z of coordinate rectangle number i with respect to traveler reference coordinate system [0]

area = Cross section area of traveler msct [0]

inyy = Moment of inertia of the cross section about the local y coordinate axis [0]

inzz = Moment of inertia of the cross section about the local z coordinate axis [0]

inyz = Moment of inertia of the cross section about the local y and z coordinate axes [0]

tors = Torsional moment of inertia of the cross section about the local x coordinate axis [0]

ycen = Local y coordinate of the cross section centroid with respect to the local reference coordinate system [0]

zcen = Local z coordinate of the cross section centroid with respect to the local reference coordinate system [0]

dely(i) = Dimension of rectangle number i in the direction of the local y coordinate axis [0]

delz(i) = Dimension of rectangle number i in the direction of the local z coordinate axis [0].

Explanation : The TSECTION PROPERTIES command is used to specify the different section properties found in the traveler elements belonging to the structure under consideration. The section types may be supplied in any order, however, each section type must be specified once and only once.

Input may be either performed by specifying the geometry of each rectangle belonging to the cross-section Fig. A3.2 or by supplying the global properties that would have been computed internally if the first method would have been chosen. In both cases, the cross section geometry is of arbitrary shape but in the former case, and on the second line of the input data, only the mstr, R=?, Y=?, Z=?, U=? and V=? parameters and identifiers are needed while in the latter case only mstr, A=?, I=? and C=? are needed. In both cases, the centroidal distance may be overriden in the MOVE subcommand below by specifying the incremental or change in the ycen or zcen parameters. When all parameters and identifiers are input on the same line, the internally computed values will overrides the global values input under the A=?, I=? and C=? identifiers.

A table of section properties is printed in the output.

The FSECTION PROPERTIES Subcommand

Syntax : FSECTION PROPERTIES N=nsct M=maxy,maxz P=iprn msct M=ymax,zmax C=ycrd,zcrd Y=ycrd Z=zcrd D=dely,delz S=oshr,yshr,zshr P=iprn

where : nsct = Total number of filamented concrete/mild steel cross-sections [1]

maxy = Maximum number of filaments in the local y- direction for any cross-section [1]

maxz = Maximum number of filaments in the local z- direction for any cross-section [1]

iprn = Output flag (0= no filaments geomtrical properties w.r.t reference coordinate system is given, 1= geometrical properties are given) [0]

msct = A given filamented cross-section number (1 =< msct =< nsct) [0]

ymax = Maximum number of filaments in the local y- direction for current cross-section [maxy]

zmax = Maximum number of filaments in the local z- direction for current cross-section [maxz]

ycrd = Local y- coordinate of the local reference axis origin with respect to the rectangular coordinates. ycrd must an integer multiple number of the y-dimension of cross-section fiber

zcrd = Local z- coordinate of the local reference axis origin with respect to the rectangular coordinates. zcrd must be an integer multiple of the y- dimension of cross-section fiber

dely = Local y- dimension of the cross-section fiber [pv]

delz = Local z- dimension of the cross-section fiber [pv]

oshr = Fraction of shrinkage strain at the origin of the local rectangular coordinate system [1]

yshr = Fraction of shrinkage strain at the corner lying on the local y- rectangular axis [1]

zshr = Fraction of shrinkage strain at the corner lying on the local z- rectangular axis [1].

Explanation : The FSECTION PROPERTIES subcommand 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 S=? identifier is used to model non-uniform, though planar, shrinkage distribution over the frame element cross section. The default is uniform shrinkage.

The frame element cross-section may be of arbitrary geometry. To facilitate the input of the frame element cross-sectional shape, the cross-section is assumed to be completely enclosed in a rectangular box the edges of which are parallel to the local reference coordinate system. Each cross-section is assumed to consist of equal dimensions, uniform material fibers the location of which are defined by integer multiple (inty and intz) of fiber dimensions. Similarly, the location of the local reference coordinate system origin is defined by integer multiple of fiber dimensions.

Additional lines are required to define the full geometry of the cross-section. The format of such a typical line is described below :

N=mfil,inty,intz G=gen1,gen2 S=dely,delz C=binc,dinc B=fbas,finc I=cinc F=iflg

where :

mfil = Filament number

inty = Local y- coordinate (integer) of filament mfil (0 =< inty =< ymax)

intz = Local z- coordinate (integer) of filament mfil (0 =< intz =< zmax)

gen1 = First filament number on the diagonal in the generation sequence [0]

gen2 = Total number (including gen1) of filaments on the diagonal in the generation sequence [0]

dely = Increment in the local y- coordinate of consecutive filaments on the diagonal [1]

delz = Increment in the local z- coordinate of consecutive filaments on the diagonal [1]

binc = Increment in filament number parallel to the base of the trapezoidal generation geometry [1]. The direction of the base (local y- or z-) is determined by the F=? identifier.

dinc = Increment in filament number along the diagonal [1].

fbas = Total number of filaments generated along the base including the filament on the diagonal. A negative value indicates generation in the negative local axis direction

finc = Increment in number of total number of filaments generated in a direction parallel to the base

cinc = Increment in local coordinate (local y- or z-) parallel to the base. Must be an integer number [1]

iflg = Flag indicator defining the base direction of generation (1= base generation is parallel to the local y- axis, 2= base generation is parallel to the local z- axis) [1]

Explanation : The frame element cross-sectional geometry may be either specified manually through fiber by fiber input using the N=? identifier only, or may be alternatively defined using the generation capability provided with the N=?, G=?, S=?, C=?, B=?, I=? and F=? identifiers. The generated shape must have parallel bases with arbitrary sloping edges that may be joining adjacent or opposite verteces Fig. A3.1. The above set of lines needed to define the geometry of each frame element cross-section must be terminated with a blank line.

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

The MILD STEEL PROPERTIES Subcommand

Syntax : MILD STEEL PROPERTIES N=nmsl

n A=alfa D=mmse W=wegt N=poison

where : nmsl = Total number of mild steel types [1]

n = Mild steel type number (1 =< n =< nmsl) [0]

alfa = Thermal expansion coefficient [0]

mmse = Mild steel material stress-strain number (1 =< mmse =< nmse) [1]

wegt = Mild steel unit weight (For dead load only) [0]

poison = Poisson's ratio [0.33].

Explanation : The MILD STEEL PROPERTIES command is used to specify the different mild (reinforcing) steel material properties found in the structure. The mild steel type number must be less than or equal to the total number of mild steel types input on the MILD STEEL PROPERTIES command line. Mild steel types may be supplied in any order; however, each mild steel type must be specified once and only once.

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

The FRAME ELEMENTS Subcommand

Syntax : FRAME ELEMENTS N=nfml F=iflg

mfml,nodi,nodj M=modl C=mcnc,mcxs S=mmld,mmxs L=locl

D=cday G=gen1,...,gen6 J=jen1,...,jen5

where : nfml = Total number of frame elements [1]

mfml = A given frame element number (1 =< mfml =< nfml) [0]

nodi = Node number for node I of mfml (1 =< nodi =< nnod) [0]

nodj = Node number for node J of mfml (1 =< nodj =< nnod) [0]

mcnc = Concrete property type number (1 =< mcnc =< ncnc) [0]

mcxs = Concrete cross section number (1 =< mcxs =< ncxs) [0]

mmld = Mild steel property type number (1 =< mmld =< nmld) [0]

mmxs = Mild steel cross section number (1 =< mmxs =< nmxs) [0]

modl = Creep time integration model number [2]

locl = Local coordinate system number (1 =< locl =< nloc) [0]

cday = Casting date of the frame element (days) [0]

iflg = Flag indicator that is needed for frame superelement generation. (1= internal generation is required, 2= number frame superelements as input) [1]

gen1 = First frame element number in the generation sequence (1 =< gen1 =< nfml) [0]

gen2 = Last frame element number in the generation sequence (1 =< gen2 =< nfml) [0]

gen3 = Frame element number increment (1 =< gen1 =< nfml) [0]

gen4 = Node I number increment [0]

gen5 = Node J number increment [0]

gen6 = Frame element casting date increment [0]

jen1 = Frame element concrete type number increment [0]

jen2 = Frame element concrete x-section type number increment [0]

jen3 = Frame element mild steel type number increment [0]

jen4 = Frame element mild steel x-section type number increment [0]

jen5 = Frame element local coordinate system number increment [0].

Explanation : The FRAME ELEMENTS command is used to define all the frame elements and the frame superelements to which they belong that are 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 local coordinate systems for the individual frame elements belonging to the same superelement must be identical.

Each frame element cross-section consists of parallel concrete and mild steel rectangular fibers. To facilitate the input phase on the user, the frame element cross-section has been split into a concrete and a mild steel subcross-sections. The casting date (in days) of the frame element 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 M=? identifier. With M=0, Eq. 2.3.2.1 is used. With M=1, Eq. 2.3.2.2 is used. With M=2, Eq. 2.3.2.3 is used.

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

The frame superelements may be generated automatically by the program or alternatively may be input manually by the user. In the latter case, additional lines (one line for each superelement) are needed as described below, but in either case, only one superelement at most may be located between any given two nodes. Each superelement, however, may contain an unlimited number of frame elements.

N=msup,mfrm G=g1,...,g4

where : msup = A given frame superelement number (1 =< msup =< nsup). The parameter nsup is the total number of actual superelements computed internally by the program based on active frame elements between all pairs of nodal points.

mfrm = a given frame element number of the group of frame elements belonging to superelement msup (1 =< mfrm =< nfml)

Explanation : The frame superelement numbering may be accelerated by using the generation capability represented in the G=? identifier side by side to the individual numbering provided by the N=? identifier. The frame superelement description may be provided in any order, however, each frame superelement description must be specified or generated once and only once. The generation parameters needed for that purpose are defined as :

g1 = First superelement number in a generation sequence (1 =< g1 =< nsup)

g2 = Last superelement number in a generation sequence (1 =< g2 =< nsup)

g3 = Frame superelement number increment

g4 = Frame element number increment. The frame element number belonging to the first frame superelement (g1) in the generation sequence is found internally by the program. In that respect, the (g1) frame superelement description must have been provided previously (previous lines) or on the same data line

using the N=? identifier.

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

The PRESTRESSING STEEL Subcommand

Syntax : PRESTRESSING STEEL N=npsl

n C=crvf W=wblf R=rlxp A=alfa **D=mpse S=smax Y=syld**

Where : npsl = Total number of prestressing steel types [1]

n = Prestressing steel type number (1 =< n =< npsl) [0]

crvf = Curvature friction factor [0]

wblf = Wobble friction factor [0]

rlxp = Relaxation coefficient [0]

alfa = Thermal expansion coefficient [0]

mpse = Prestressing steel material number (1 =< mpse =< npse) [1]

smax = Maximum tensile stress [maximum stress-strain stress] used in the pretensioning/posttensioning load computation operation only

syld = Yield stress used in relaxation computations only [1st point on the stress-strain curve].

Explanation : The PRESTRESSING STEEL command is used to specify the different prestressing steel material properties found in the structure. The prestressing steel type number n must be less than or equal to the total number of prestressing steel types input on the PERSTRESSING STEEL command line. The prestressing steel types may be supplied in any order; however, each prestressing steel type must be specified once and only once.

Prestressing steel is considered as a relaxing **nonlinear material**. The strength **syld** input is used only in the relaxation modelling. The friction coefficients are used in computing the initial tendon forces under the STRESS subcommand of the CHANGE STRUCTURE command.

The TENDON GEOMETRY Subcommand

Syntax : TENDON GEOMETRY N=ntdn P=iprn

(Tendon Geometry Specification)

where : ntdn = Total number of prestressing steel tendons [1]

iprn = Output print flag (1= short form, 2= long form) [1]

Explanation : The tendon geometry command is used to specify and generate the geometry of all the prestressing tendons.

Each tendon is modeled as a series of segments connected at tendon points **Fig. A3.3**. 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 superelement. The global X-, Y- and Z-** coordinates of the tendon points completely define the tendon 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-) **and (R-,T-)** coordinate system specified arbitrarily by the user. The R-, S- **and T-** coordinates are converted internally to the global **(X-, Y-, Z-) 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 bridge. 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 : (Repeated for each tendon)

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

N S=nspn M=matl A=area **F=iflg**

where : n = Tendon identification number (1 =< n =< ntdn) [0]

nspn = Number of spans used to define the tendon point geometry [0]

matl = Prestressing steel material property type number (1 =< matl =< npsl) [1]

area = Cross sectional area for the tendon

iflg = Spans local coordinate systems input flag indicator (0= local coordinate system number is used, 1= local coordinate system is specified with one of the methods described in the following input line) [1].

For each tendon 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-, T-)** coordinate system **Fig. A3.3** 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-, Z-)** coordinate system. Span numbers must be supplied in ascending consecutive order starting with span 1.

ns N=nnis G=n1,n2,inc **O=ox,oy,oz R=rx,ry,rz S=sx,sy,sz**

T=tx,ty,tz L=locl F=iflg J=j1,j2,j3,j4

where : ns = A given span number

nnis = Number of nodes in span ns (including both end nodes)

n1 = Node number at "left" end of span in the direction of R- axis [0]

n2 = Node number at "right" end of span in the direction of R- axis [0]

inc = Node number increment for generating nodes in the span along the the local R- axis [0]

locl = Local coordinate system number for the span [1]

iflg = Spans local coordinate systems input flag indicator (0= local coordinate system number is specified through O=? and L=? identifiers, 1= local coordinate system is specified using the (O=?, R=?, S=? and T=?) or J=? identifiers. The default value of iflg on this line is iflg input on the previous tendon command line. Do not input the value again on this line unless it is necessary to reset the value

ox,oy,oz = Global X-, Y- and Z- coordinates respectively of the local coordinate system origin [0,0,0]

rx,ry,rz = Global X-, Y- and Z- coordinates respectively of an arbitrary point lying on the local R- axis [1,0,0]

sx,sy,sz = Global X-, Y- and Z- coordinates respectively of an arbitrary point lying on the local S- axis [0,1,0]

tx,ty,tz = Global X-, Y- and Z- coordinates respectively of an arbitrary point lying on the local T- axis [0,0,1]

j1 = Node number of the origin of the local coordinate system [0]

j2 = Node number lying on the local R- axis such that the vector formed by the end nodes j1 and j2 defines the local R- axis direction [0]

j3 = Node number lying on the local S- axis such that the vector formed by the end nodes j1 and j3 defines the local S- axis direction [0]

j4 = Node number lying on the local T- axis such that the vector formed by the end nodes j1 and j4 defines the local T- axis direction [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 nnis node numbers included in the span. If the G=? identifier is provided, then this line must not be provided.

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

where : 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 :

R=rlli(s),rllp(s),rlri(s),rlli(t),rllp(t),rlli(t) S=slt,slp,srt T=tlt,tlp,trt

where : rlli(s) = Fraction of the total span length between the "left" end of the span and the "left" inflexion point (a positive number) in the (R-, S-) plane Fig. A3.4

rllp(s) = Fraction of the total span length between the "left" end of the span and the point of zero tendon slope ("low point") relative to the (R-, S-) system (a positive number) Fig. A3.4

rlri(s) = Fraction of the total span length between the "right" inflection point and the right end of the span in the (R-, S-) plane (a positive number) Fig. A3.4

rlli(t) = Fraction of the total span length between the "left" end of the span and the "left" inflexion point (a positive number) in the (R-, T-) plane Fig. A3.4

rllp(t) = Fraction of the total span length between the "left" end of the span and the point of zero tendon slope ("low point") relative to the (R-, t-) system (a positive number) Fig. A3.4

rlri(t) = Fraction of the total span length between the "right" inflection point and the right end of the span in the (R-, T-) plane (a positive number) Fig. A3.4

slt = S- coordinate of "left" end of tendon in (R-, S-) plane Fig. A3.4

slp = S- coordinate of point of zero slope of tendon in (R-, S-) plane Fig. A3.4

srt = S- coordinate of "right" end of tendon in (R-, S-) plane Fig. A3.4

tlt = T- coordinate of "left" end of tendon in (R-, T-) plane Fig. A3.4

tlp = T- coordinate of point of zero slope of tendon in (R-, T-) plane Fig. A3.4

trt = T- coordinate of "right" end of tendon in (R-, T-) plane Fig. A3.4.

If direct tendon point coordinate input is used for the span, the following nnis lines must be provided in place of the one above. **Two options, either the (R=?, S=? and T=?) identifiers or the C=? identifier, are provided for the input of the tendon points local coordinates.** In this case, the default R- coordinates are the R- coordinates of the nodes included in the span. If parametric generation is used, then these lines must not be provided.

np R=r(np) S=s(np) T=t(np) C=r(np),s(np),t(np)

where : np = A given tendon point number (1 =< np =< nnis)

r(np) = R- coordinate of tendon point

s(np) = S- coordinate of tendon point

t(np) = T- coordinate of tendon point

This sequence of lines must not include any blank lines

The TRAVELERS Subcommand

Syntax : TRAVELERS N=ntrv

mtrv X=mstr E=emod W=wegt N=nnit L=locl

where : ntrv = Total number of travelers [1]

mtrv = A given traveler number (1 =< n =< ntrv)

mstr = Cross section type number (1 =< nx =< nsct)

emod = Elastic modulus of traveler material

wegt = Total weight of traveler mtrv

nnit = Number of nodes in traveler mtrv (1 =< nnit =< nnod)

locl = Local coordinate system number [1]

Explanation : The TRAVELERS command is used to describe all the traveling formwork 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 and are input under the MOVE subcommand of the CHANGE STRUCTURE command.

The MESH COMPLETE Subcommand

Syntax : MESH COMPLETE

Explanation : The MESH COMPLETE subcommand 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

Syntax : SET D=day T=temp G=gx,gy,gz,gxz **C=tutl,iutl,astl,istl,tptl,attl,artl**

A=aclr I=mxit **N=uzya**

where : day = Current date (days) [0,pv]

temp = Current temperature (degrees Fahrenheit) [70]

gx, gy, gz = X-, Y- and Z- direction translational gravity load multiplier (fraction) respectively [0,-1,0]

gxz = XX- and ZZ- direction rotational gravity load multiplier (fraction) [-1]

tutl = Unbalanced load ratio tolerance limit with respect to total applied nodal loads [0.01]

iutl = Unbalanced load ratio tolerance limit with respect to incremental applied nodal loads [0.001]

astl = Absolute stress change convergence tolerance limit [1 psi]

istl = Stress change ratio convergence tolerance limit with respect to

incremental fiber stress [0.01]

tptl = Superelement material parameters ratio convergence tolerance limit with respect to total material values [0.01]

attl = Absolute allowable incremental nodal translational displacement for a given iteration within a given time step [0]

artl = Absolute allowable incremental nodal rotational displacement for a given iteration within a given time step [0]

uayz = Fraction of the superelement material parameter at the beginning of the iteration contributing toward the superelement material parameters used to form the superelement characteristic matrices affecting the nodal displacements-strain relationship [0.5]. The remaining contribution is used from the superelement material parameters at the end of the time step.

aclr = Convergence acceleration factor [0.71]

mxit = Maximum iterations per solution step [30]

Explanation : 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 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-, Y-, Z-) directions. The gravity load multipliers gx, gy, gz, gxz influence only the gravity load increment and should be set only once prior to any construction

operations performed under the CHANGE STRUCTURE command.

The C=? identifier is used to set convergence tolerances for the creep **and nonlinear material series** solution. The solution is considered to have converged when the maximum stress change in any element is less than the specified value over the iteration.

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

The I=? identifier is used to set the maximum number of iterations allowed in any solution step. After this number of iterations, the solution terminates. A shorter time step or higher convergence tolerances help reduce the required number of iterations.

The CHANGE STRUCTURE Command

Syntax : CHANGE STRUCTURE

(Construction Subcommands)

Explanation : 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 the frame elements and prestressing tendons, and the movement of travelers in the **three dimensional frame structure**. The structure to be analysed is completely defined by the cumulative effects of the CHANGE STRUCTURE command.

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 tendons), MOVE (to move travelling formwork) 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 changes on the displacements and internal stresses in the structure are found by using the

SOLVE command after the CHANGE STRUCTURE command and any optional LOADING commands.

The RESTRAINTS Subcommand

Syntax : RESTRAINTS

n1,n2,inc **R=rx,ry,rz,rxx,ryy,rzz**

where : 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 the intermediate nodes in the series [1]

rx,ry,rz = **X-, Y- , Z- displacement restraint specification**

rxx,ryy,rzz = **X-, Y- , Z- rotation restraint specification**

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

Each node has six displacement degrees of freedom, each of which may be specified with one of the 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 six degrees of freedom when they are initialized.

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.

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

The BUILD Subcommand

Syntax : BUILD N=n1,n2,inc D=cd

where : 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 used to define the intermediate frame elements in the sequence [1]

cd = Casting date of concrete in the specified elements

Explanation : The BUILD subcommand is used to install new frame elements into the structure. The elements can be later 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 subcommand, backwards 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 frame elements gravity loads are assumed to be acting in the global Y- direction and hence statically equivalent applied moments about the global X- and the Z- axes at the end nodes are also included in the analysis scaled with the corresponding load multipliers from 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 behaviour of the element. This makes it necessary under this subcommand to generate elements in a statically feasible order.

The REMOVE Subcommand

Syntax : REMOVE N=n1,n2,inc

where : 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 that defines the intermediate frame elements in the sequence.

Explanation : 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 all frame elements belonging to the same superelement whenever one element of this group has been removed.**

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 LOADING command described below are **not** automatically removed and must be removed manually (by applying an equal but opposite force with another LOADING command) **before** removing the element.

The STRESS Subcommand

Syntax : STRESS N=n1,n2,inc R=ra,rb S=sa,sb F=fa,fb D=da,db P=iprn

where : n1 = Tendon number of first tendon in a series of tendons with identical stressing specifcations

n2 = Tendon number of last tendon in the series [n1]

inc = Tendon number increment that defines the intermediate tendons in the sequence [1]

ra = Jacking stress ratio at tendon end "A" with respect to the peak stress [0]

rb = Jacking stress ratio at tendon end "B" with respect to the peak stress [0]

sa = Jacking stress at tendon end "A" [0]

sb = Jacking stress at tendon end "B" [0]

fa = Jacking force at tendon end "A" [0]

fb = Jacking force at tendon end "B" [0]

da = Anchorage slip (draw-in) at tendon end "A" [0]

db = Anchorage slip (draw-in) at tendon end "B" [0]

iprn = Output flag [1]

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

A tendon is initially stressed by specifying its jacking force and anchorage slip values under this command. The tendon segment initial forces are then calculated based on this 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 the use of this command with a zero jacking force.

The jacking force is specified using either the R=?, S=? or F=? identifier. When the R=? identifier is used, the jacking force is computed using an expression of the form $f_j = r_j$ * fult * area. When the S=? identifier is used, the jacking force is computed using an expression of the form $f_j = s_j$ * area. When the F=? identifier is used, the jacking force f_j is directly input.

In the first solution step after a STRESS command has been applied to a tendon, only the tendon's equivalent forces are included in the analysis. In subsequent solution steps the stiffness is also included. This represents a tendon which is unbonded during stressing and bonded thereafter.

The P=? identifier is used to control the output from the command. If P=0 is used, there is no output from the command. If P=1 is used, a table of initial segment forces for the tendon is output.

The MOVE Subcommand

Syntax : MOVE N=mtrv D=n(1),...,n(nnit) **C=xcny,xcnz**

where :

mtrv = A given traveler number

n(?) = Destination nodes for the new traveler location

nnit = Number of nodes in traveler mtrv

xcny = Increment in excentricity measured along the local y-axis of the traveler centroidal origin with respect to the superelement reference origin spanning the same end nodes [0]

xcnz = Increment in excentricity measured along the local z-axis of the traveler centroidal origin with respect to the superelement reference origin spanning the same end nodes [0]

Explanation : 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 in the structure as many times as desired during the analysis. The traveler may 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 loads are automatically moved. **The increment in excentricities at the new location with respect to the original excentricity of the traveler centroid with respect to the superelement local reference system input under the TSECTION PROPERTIES subcommand of the MESH INPUT command will override any previously input excentricities under the same command.**

The CHANGE COMPLETE Subcommand

Syntax : CHANGE COMPLETE

Explanation : The CHANGE COMPLETE command has no arguments. It signals the program that the interpretation of the 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

Syntax : LOADING

(Loading Data Lines)

Explanation : 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 effects until they are removed by the application of an equal but opposite loading.

Each loading 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,fz,fxx,fyy,fzz**

where : n1 = Node number of the first node in a series of nodes with identical loading

n2 = Node number of the last node in the series [n1]

inc = Node number increment that defines the intermediate nodes in the sequence [1]

fx, fy, fz = X-, Y-, Z- force increment respectively

fxx, fyy, fzz = X-, Y-, Z- moment increment respectively

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

N=n1,n2,inc **D=dx,dy,dz,dxx,dyy,dzz**

where : n1 = Node number of the first node in a series of nodes with identical applied displacements

n2 = Node number of the last node in the series [n1]

inc = Node number increment that defines the intermediate nodes in the series [1]

dx, dy, dz = X-, Y-, Z- displacement increment respectively

dxx, dyy, dzz = X-, Y-, Z- moment increment respectively

For uniformly distributed frame elements loads the following data must be provided. Frame element loads are in the global directions, and are specified as force per unit projected length **Fig. A3.5**. Frame element loads are converted to concentrated nodal forces for the analysis.

L=l1,l2,inc **F=fx,fy,fz**

where : l1 = Frame element number of the first element in a series of elements with identical applied loads

l2 = Frame element number of the last element in the series [n1]

inc = Frame element number increment that defines the intermediate frame elements in the series [1]

fx, fy, fz = X-, Y-, Z- force per unit X-, Y- and Z- projected length respectively

For temperature change loadings the following data must be provided. The temperature of each element may be different and may vary linearly in either the local y- or z- directions (that is planar distribution) over the depth of the element. Temperature strains are included in the frame elements generated with the L=? identifier, and also in the prestressing tendon segments with the same node numbers as the generated elements. 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 **T=Opoint,Ypoint,Zpoint**

where : l1 = Frame element number of the first element in a series of elements with identical temperatures

l2 = Frame element number of the last element in the series [n1]

inc = Frame element number increment that defines the intermediate frame elements in the series [1]

Opoint,Ypoint,Zpoint = Temperature values at the origin, Y- corner point (point along the local y- axis) and Z- corner point (point along the local z- axis) in the frame element cross section needed to define the planar temperature distribution across the frame element cross section. The mild steel subcross section of the frame element will be subjected to the same temperature distribution applied to the concrete counterpart.

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

The SOLVE Command

Syntax : SOLVE S=nstp D=date P=iprn I=mxit A=aclr **U=istf G=igss**

where : nstp = Number of time steps [1]

date = Day number at the end of the solution [current date]

iprn = Output print flag [0]

mxit = Maximum iterations per time step

aclr = Convergence acceleration factor

istf = Stiffness update indicator (1= do not update, 2= update for each iteration) [1]

igss = Number of gauss points used in the element stiffness integration (either 1 or 3 points) [1]

Explanation : 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 if omitted default to their values from the previous SOLVE command or the values initialized under the SET command. Once these parameters have been entered, they should not be entered again unless it is necessary to reset their values. The initial default values are those described for the SET command.

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.

The solution is performed by dividing the solution interval into nstp time steps. 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 convenient option to simplify input when several time steps are required. The entire external loading increment, however, is applied in the first time step.

The A=? identifier is used to set the maximum number of iterations allowed in each time step. **After this number of iterations, the solution terminates. A shorter time step or higher convergence tolerances help reduce the required number of iterations.**

The quantity of output is controlled using the P=? identifier. For P=0, no response output is printed. For P=1, the response change for each time step is printed. For P=2, the response change for each iteration is also printed. **If P=3 is used, detailed information about node restraint and equation numbering is given too. If P=4 is used, the structure global stiffness is printed out also.** Output of the total response is performed with the OUTPUT command.

The OUTPUT Command

Syntax : OUTPUT P=iprn

where : iprn = Output print flag [1]

Explanation : The OUTPUT command prints a summary of the total displacements, reactions and element end forces of the current structure.

The P=? identifier is used to control the quantity of the element results output. Nodal displacements and reactions are always printed. If P=0 is used, no element output is printed. If P=1 is used, a short form element output is printed. If P=2 is used, a long form element output is printed. For frame elements, the short form consists of the total element end stress resultants in global coordinate system, while the long form additionally consists of **the total end resultants in the local coordinate system. Fiber strains and stresses may be output separately using the RECORD command described below.** For tendon elements, the short form consists of the tendon segment forces in the global coordinate system while the long form additionally consists of the tendon segment forces in the corresponding superelement coordinate system and the percentage total force loss. For traveler elements, the short and long form descriptions is similar to the frame element's.

In addition to the results for the individual elements, a summary of total internal forces for each **frame superelement**, summed from the frame element components plus all tendon segments with the same end nodes, is given. **Similarly to the frame elements, two output forms exist.**

The OUTPUT command performs no numerical operations associated with the time dependent analysis; it only prints the results.

The CAMBER Command

Syntax : CAMBER C=idof

where : idof= Degree of freedom or displacement component to be printed [2]

= 1 For X- displacement

= 2 For Y- displacement

= 3 For Z- displacement

= **4 For X- rotation**

= **5 For Y- rotation**

= **6 For Z- rotation**

Explanation : The CAMBER command is used to print a history of adjusted nodal displacements for all nodes in the structure. Each time the OUTPUT command is used, a record of total nodal displacements is saved. Under the CAMBER command, the current total nodal displacements are subtracted from each set of saved displacaments, and the result is output for the specified displacement component. This process is repeated for each displacement record (i.e. for each time the OUTPUT command was used).

The displacements which are 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 RECORD Command

Syntax : RECORD

Explanation : **The RECORD command has no argument of its own, but is followed by a series of record subcommands which specify the record type to be printed. The record command lists the values of some of the parameters belonging to the record type which can not be listed under the OUTPUT command above. The parameters belonging to the frame elements, traveler elements, tendon elements and finally to the concrete and mild steel fiber cross sections may be printed. Voluminous output may result from the use of this command.**

The RECORD subcommands include FRAME ELEMENTS, FILAMENTS, TENDON ELEMENTS, TRAVELER ELEMENTS and RECORD COMPLETE

described below. The RECORD command may be used as many times as needed. Similarly, each subcommand may be used as may time as needed and may be used in any order.

The FRAME ELEMENTS Subcommand

Syntax : FRAME ELEMENTS L=mfrm

where : mfrm = A given frame element number (1 =< mfml =< nfml) [1]

Explanation : The FRAME ELEMENTS subcommand of the RECORD command is used to print the values of the temperature plane coefficients, nodal forces in the local superelement system, the nodal global forces, the casting date, previous update date, current ultimate compressive and tensile coefficients and the previous ultimate compressive and tensile coefficients. The FRAME ELEMENTS subcommand may be repeated as many times as needed to cover all the frame elements requested.

The FILAMENTS Subcommand

Syntax : FILAMENTS L=mfml I=indx F=iflg

where : mfml = A given frame element number (1 =< mfml =< nfml) [1]

indx = Integer value variable that specifies whether concrete or mild steel filaments data is to printed (1= concrete data, 2= mild steel data) [1]

iflg = Output flag indicator that determines which parameters are to be printed [1]

Explanation : The FILAMENTS subcommand of the RECORD command is used to print the values of the current total filament stress, mechanical strain, tangent modulus, and the parameters related to the independent path loading/unloading stress-strain diagram of the material type chosen using the I=? identifier. IF F=1 is used, the total stress values for all the fiber of the same type as chosen through I=? identifier are printed. If F=2 is used, the mechanical strains for the same fibers are printed.

If F=3 is used, the current tangent modulus of the same fibers are printed. Finally, if F=4 is used, all the values of the parameters related to the loading/unloading stress-strain are printed. The FILAMENTS command may be repeated as many times as desired to cover all the frame elements requested.

The TENDON ELEMENTS Subcommand

Syntax : TENDON ELEMENTS L=mtdn S=mseg

where : mtdn = A given tendon number (1 =< mtdn =< ntdn) [1]

mseg = A given segment number (1 =< mseg =< nseg) [1]

Explanation : The TENDON ELEMENTS subcommand of the RECORD command is used to print the values of the current total stress, strain and modulus of elasticity of the tendon segment in addition to the values of the parameters related to the loading/unloading stress-strain diagram operations, the tendon temperature and the end frame elements to which the tendon segment is connected, the stressing and previous update dates and finally the tendon segment current and previous forces. The TENDON ELEMENTS subcommand may be repeated as many times as needed to cover all the tendon elements requested.

The TRAVELER ELEMENTS Subcommand

Syntax : TRAVELER ELEMENTS L=mtrv S=mseg

where : mtrv = A given traveler number (1 =< mtrv =< ntrv) [1]

mseg = A given segment number (1 =< mseg =< nseg) [1]

Explanation : The TRAVELER ELEMENTS subcommand of the RECORD command is used to print the values of the traveler nodal forces in the traveler local coordinate system (current and previous nodal forces) in addition to the corresponding nodal forces in the corresponding superelement coordinate system.

The RECORD COMPLETE Subcommand

Syntax : **RECORD COMPLETE**

Explanation : The RECORD COMPLETE subcommand has no arguments of its own. Its simply signals the program that the record output phase of the analysis is complete. RECORD command may, however, be used as many times as desired during the analysis.

The STOP Command

Syntax : STOP

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

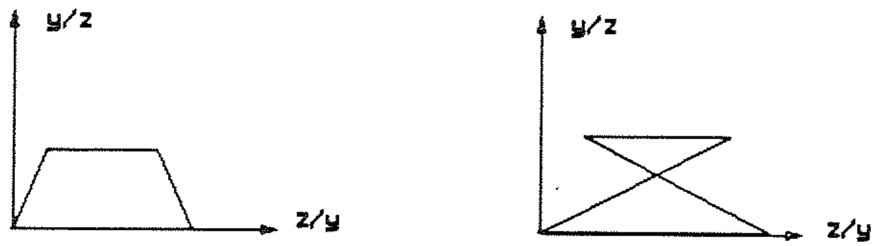


Fig. A3.1 Acceptable Geometric Shapes
For Internal Filament Generation.

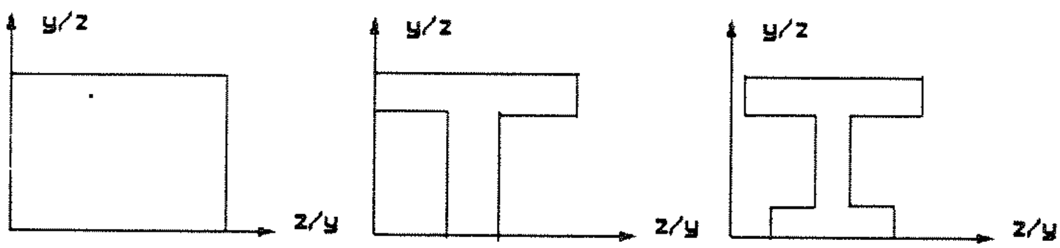


Fig. A3.2 Common Cross-section Geometrical Shapes.

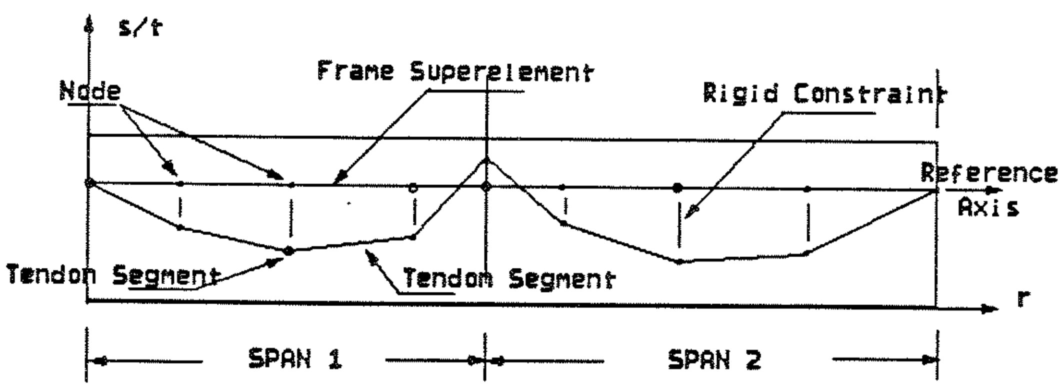


Fig. A3.3 Tendon Discretization
Ketchum [27]

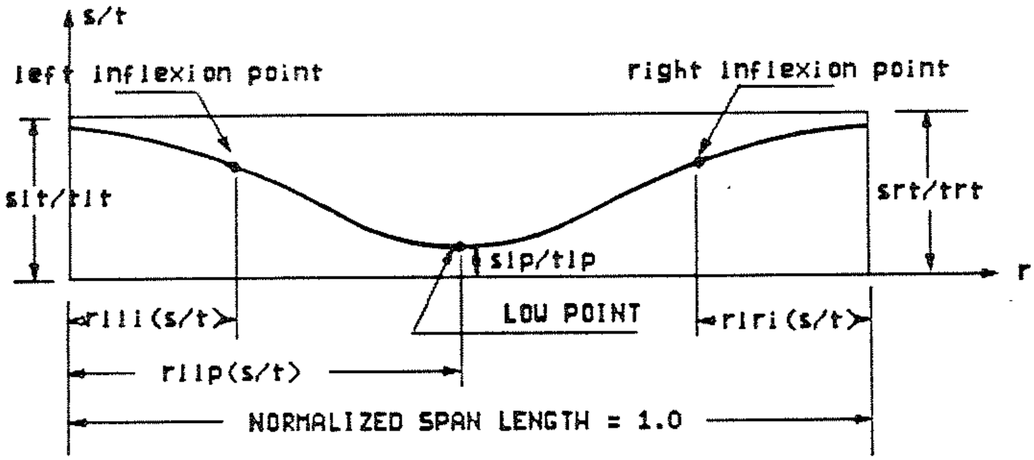


Fig. A3.4 TENDON GEOMETRY GENERATION PARAMETERS

Ketchum [27]

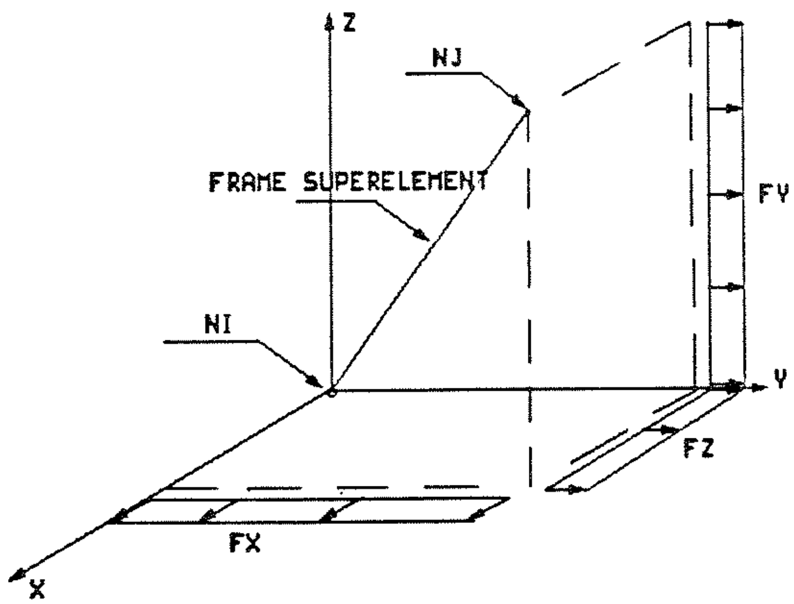


Fig. A3.5 FRAME SUPERELEMENT LOADS

APPENDIX B

INPUT DATA FILE LISTINGS OF

NUMERICAL EXAMPLES DISCUSSED IN CHAPTER 5

AS RUN BY THE COMPUTER PROGRAM SPCF3D

```
START ;
TITLE
EXAMPLE 5.1   - CHOUDHURY STRAIGHT BRIDGE
                THREE SPAN CONTINUOUS POST-TENSIONED STRAIGHT BOX GIRDER BRIDGE
                LOADED TO FAILURE WITH NO CREEP ANALYSIS
                UNITS = LB-IN SYSTEM
CONCRETE PARAMETERS N=1 ! 1 M=ACI W=143.96 T=.6589
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=3975.297 C=0.0 S=0.0 W=155/1728 !
LOCAL SYSTEM N=1 ! 1 !
MATERIAL MODELS N=1,1,1 P=10,2,5
    1 M=PRES ! N=1 C=196600,0.00715 ! N=2 C=220000,0.009 ! N=3 C=240000,0.0115
      N=4 C=245000,0.0135 ! N=5 C=270000,0.058
    1 M=CONC I=1 E=0.002,0.004 N=8
    1 M=MILD I=2 N=2 S=60000 E=2.069E-03
      N=1 C=60000,2.069E-03 ! N=2 C=60000.01,0.03 !
FSECTION PROPERTIES N=6 M=54,136 P=1
    1 C=+17,0 D=4,201
      N=1,0,0 G=1,2 S=1,0 C=1,1 B=1,0 I=1 F=2 !
    2 C=+4,0 D=13.33,30
      N=1,0,0 G=1,7 S=1,0 C=1,1 B=1,0 I=1 F=2 !
    3 C=-13,0 D=2.45,408
      N=1,0,0 G=1,4 S=1,0 C=1,1 B=1,0 I=1 F=2 !
    4 C=75,68 D=0.9,3
      N=1,92,68 !  N=2,58,68 !  N=3,25,68 !
    5 C=38,29 D=1.75,7
      N=1,58,29 !
    6 C=42,68 D=1.6,3
      N=1,2,68 !
MILD STEEL PROPERTIES N=1 ! 1 D=1 !
FRAME ELEMENTS N=60
    01,1,2 D=0 C=1,1 S=1,4 L=1 G=01,20,1,1,1,0
    21,1,2 D=0 C=1,2 S=1,5 L=1 G=21,40,1,1,1,0
    41,1,2 D=0 C=1,3 S=1,6 L=1 G=41,60,1,1,1,0 !
PRESTRESSING STEEL N=1 ! 1 C=0.25 W=0.0002/12 !
TENDON GEOMETRY N=1 ! 1 S=2 M=1 A=26.32 F=0
    1 N=11 G=01,11,1 O=0,0,0 L=1 ! R=0.0,0.5,0.1,0,0,0 S=00,-55,+29 T=0,0,0
    2 N=11 G=11,21,1             ! R=0.2,1.0,1.0,0,0,0 S=29,-55,-55 T=0,0,0
MESH COMPLETE !SET D=28 C=0.05,0.05,1,0.01,0.01,10.0,0.5 N=1.0
CHANGE STRUCTURE
BUILD N=1,60,1 D=0
RESTRAINTS ! 1,21,01 R=0,0,1,1,1,0 !
RESTRAINTS ! 1,11,10 R=0,1,1,1,1,0 ! 21 R=1,0,1,1,1,1 !
STRESS N=1 F=5320000,0 D=0.25
CHANGE COMPLETE
SOLVE !OUTPUT P=1
LOADING ! N=15,19,2 F=0,-48000,0,0,0,0 ! N=21 F=0,-24000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
LOADING ! N=15,19,2 F=0,-48000,0,0,0,0 ! N=21 F=0,-24000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
LOADING ! N=15,19,2 F=0,-48000,0,0,0,0 ! N=21 F=0,-24000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
LOADING ! N=15,19,2 F=0,-24000,0,0,0,0 ! N=21 F=0,-12000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
```

```
LOADING ! N=15,19,2 F=0,-24000,0,0,0,0 ! N=21 F=0,-12000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
LOADING ! N=15,19,2 F=0,-24000,0,0,0,0 ! N=21 F=0,-12000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
LOADING ! N=15,19,2 F=0,-12000,0,0,0,0 ! N=21 F=0,-6000,0,0,0,0 !
SOLVE U=2 I=100 !OUTPUT P=1
LOADING ! N=15,19,2 F=0,-12000,0,0,0,0 ! N=21 F=0,-6000,0,0,0,0 !
SOLVE U=2 I=100 !OUTPUT P=1
STOP
```

```
START ;
TITLE
EXAMPLE 5.2   - CHOUDHURY STRAIGHT BRIDGE
              THREE SPAN CONTINUOUS POST-TENSIONED STRAIGHT BOX GIRDER BRIDGE
              WITH CREEP ANALYSIS UP TO 10000 DAYS (27.4 YEARS) AND THEN LOADED
              TO FAILURE
              UNITS = LB-IN SYSTEM
CONCRETE PARAMETERS N=1 ! 1 M=ACI W=143.96 T=.6589
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=3975.297 W=155/1728 S=0.0008 C=2.35 !
LOCAL SYSTEM N=1 ! 1 !
MATERIAL MODELS N=1,1,1 P=10,2,5
    1 M=PRES ! N=1 C=196600,0.00715 ! N=2 C=220000,0.009 ! N=3 C=240000,0.0115
      N=4 C=245000,0.0135 ! N=5 C=270000,0.058
    1 M=CONC I=1 E=0.002,0.004 N=8
    1 M=MILD I=2 N=2 S=60000 E=2.069E-03
      N=1 C=60000,2.069E-03 ! N=2 C=60001,0.03 !
FSECTION PROPERTIES N=6 M=54,136 P=1
    1 C=+17,0 D=4,201
        N=1,0,0 G=1,2 S=1,0 C=1,1 B=1,0 I=1 F=2 !
    2 C=+4,0 D=13.33,30
        N=1,0,0 G=1,7 S=1,0 C=1,1 B=1,0 I=1 F=2 !
    3 C=-13,0 D=2.45,408
        N=1,0,0 G=1,4 S=1,0 C=1,1 B=1,0 I=1 F=2 !
    4 C=75,68 D=0.9,3
        N=1,92,68 !  N=2,58,68 !  N=3,25,68 !
    5 C=38,29 D=1.75,7
        N=1,58,29 !
    6 C=42,68 D=1.6,3
        N=1,2,68 !
MILD STEEL PROPERTIES N=1 ! 1 D=1 !
FRAME ELEMENTS N=60
    01,1,2 D=28 C=1,1 S=1,4 L=1 G=01,20,1,1,1,0
    21,1,2 D=28 C=1,2 S=1,5 L=1 G=21,40,1,1,1,0
    41,1,2 D=28 C=1,3 S=1,6 L=1 G=41,60,1,1,1,0 !
PRESTRESSING STEEL N=1 ! 1 C=0.25 W=0.0002/12 Y=210000 R=10.0 !
TENDON GEOMETRY N=1 ! 1 S=2 M=1 A=26.32 F=0
    1 N=11 G=01,11,1 O=0,0,0 L=1 ! R=0.0,0.5,0.1,0,0,0 S=00,-55,+29 T=0,0,0
    2 N=11 G=11,21,1             ! R=0.2,1.0,1.0,0,0,0 S=29,-55,-55 T=0,0,0
MESH COMPLETE !SET D=28 G=0,-1,0,0 C=0.05,0.05,1,0.01,0.01,10.0,1.0 N=0.5
CHANGE STRUCTURE
BUILD N=1,60,1 D=0
RESTRAINTS ! 1,21,01 R=0,0,1,1,1,0 !
RESTRAINTS ! 1,11,10 R=0,1,1,1,1,0 ! 21 R=1,0,1,1,1,1 !
STRESS N=1 F=5320000,0 D=0.25
CHANGE COMPLETE
SOLVE D=00000 S=1 ! OUTPUT P=1 !
SOLVE D=00100 S=5 ! OUTPUT P=1 !
SOLVE D=00200 S=5 ! OUTPUT P=1 !
SOLVE D=10000 S=5 ! OUTPUT P=1 !
LOADING ! N=15,19,2 F=0,-48000,0,0,0,0 ! N=21 F=0,-24000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
LOADING ! N=15,19,2 F=0,-48000,0,0,0,0 ! N=21 F=0,-24000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
```

```
LOADING ! N=15,19,2 F=0,-48000,0,0,0,0 ! N=21 F=0,-24000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
LOADING ! N=15,19,2 F=0,-24000,0,0,0,0 ! N=21 F=0,-12000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
LOADING ! N=15,19,2 F=0,-24000,0,0,0,0 ! N=21 F=0,-12000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
LOADING ! N=15,19,2 F=0,-12000,0,0,0,0 ! N=21 F=0,-6000.,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
LOADING ! N=15,19,2 F=0,-6000,0,0,0,0 ! N=21 F=0,-3000.,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
LOADING ! N=15,19,2 F=0,-6000,0,0,0,0 ! N=21 F=0,-3000.,0,0,0,0 !
SOLVE !OUTPUT P=1
STOP
```

```
START ;
TITLE
EXAMPLE 5.3A - CHOUDHURY CURVED BRIDGE (CONCENTRIC LOADING TO FAILURE)
               THREE SPAN CONTINUOUS POST-TENSIONED CURVED BOX GIRDER BRIDGE
               UNITS = LB-IN SYSTEM
CONCRETE PARAMETERS N=1 ! 1 M=ACI W=143.96 T=.6589
MESH INPUT
NODES N=82
     1 X=-2979.345 Z=-5159.580 !2 X=-2927.235 Z=-5189.321
     3 X=-2874.828 Z=-5218.536 !4 X=-2822.130 Z=-5247.222
     5 X=-2769.146 Z=-5275.376 !6 X=-2715.881 Z=-5302.994
     7 X=-2662.340 Z=-5330.075 !8 X=-2608.530 Z=-5356.615
     9 X=-2554.454 Z=-5382.613 !10 X=-2500.120 Z=-5408.064
     11 X=-2445.533 Z=-5432.967 !12 X=-2390.697 Z=-5457.318
     13 X=-2335.619 Z=-5481.117 !14 X=-2280.304 Z=-5504.359
     15 X=-2224.757 Z=-5527.044 !16 X=-2168.985 Z=-5549.167
     17 X=-2112.993 Z=-5570.728 !18 X=-2068.045 Z=-5587.571
     19 X=-2022.963 Z=-5604.050 !20 X=-1977.750 Z=-5620.166
     21 X=-1932.408 Z=-5635.917 !22 X=-1886.940 Z=-5651.302
     23 X=-1841.350 Z=-5666.320 !24 X=-1795.641 Z=-5680.971
     25 X=-1749.815 Z=-5695.253 !26 X=-1703.876 Z=-5709.165
     27 X=-1657.825 Z=-5722.707 !28 X=-1611.668 Z=-5735.877
     29 X=-1565.406 Z=-5748.675 !30 X=-1519.042 Z=-5761.100
     31 X=-1472.579 Z=-5773.151 !32 X=-1426.021 Z=-5784.827
     33 X=-1379.370 Z=-5796.127 !34 X=-1356.011 Z=-5801.637
     35 X=-1332.630 Z=-5807.052 !36 X=-1309.227 Z=-5812.373
     37 X=-1285.803 Z=-5817.600 !38 X=-1262.359 Z=-5822.732
     39 X=-1238.893 Z=-5827.770 !40 X=-1215.408 Z=-5832.713
     41 X=-1191.903 Z=-5837.562 !42 X=-1162.493 Z=-5843.489
     43 X=-1133.054 Z=-5849.269 !44 X=-1103.587 Z=-5854.900
     45 X=-1074.092 Z=-5860.383 !46 X=-1044.569 Z=-5865.717
     47 X=-1015.020 Z=-5870.902 !48 X=-985.4451 Z=-5875.939
     49 X=-955.8452 Z=-5880.826 !50 X=-917.3359 Z=-5886.956
     51 X=-878.7816 Z=-5892.835 !52 X=-840.1897 Z=-5898.461
     53 X=-801.5617 Z=-5903.834 !54 X=-762.8994 Z=-5908.954
     55 X=-724.2044 Z=-5913.822 !56 X=-685.4784 Z=-5918.435
     57 X=-646.7230 Z=-5922.796 !58 X=-619.8760 Z=-5925.666
     59 X=-593.0162 Z=-5928.414 !60 X=-566.1443 Z=-5931.040
     61 X=-539.2608 Z=-5933.545 !62 X=-512.3661 Z=-5935.928
     63 X=-485.4610 Z=-5938.189 !64 X=-458.5459 Z=-5940.328
     65 X=-431.6214 Z=-5942.345 !66 X=-404.6880 Z=-5944.240
     67 X=-377.7463 Z=-5946.013 !68 X=-350.7968 Z=-5947.663
     69 X=-323.8402 Z=-5949.192 !70 X=-296.8769 Z=-5950.598
     71 X=-269.9075 Z=-5951.883 !72 X=-242.9325 Z=-5953.045
     73 X=-215.9526 Z=-5954.085 !74 X=-188.9682 Z=-5955.002
     75 X=-161.9799 Z=-5955.797 !76 X=-134.9884 Z=-5956.470
     77 X=-107.9940 Z=-5957.021 !78 X=-80.99747 Z=-5957.449
     79 X=-53.99924 Z=-5957.755 !80 X=-26.99989 Z=-5957.938
     81 X=-00.00000 Z=-5958.000 !82 X=0 Y=0 Z=0 !
CONCRETE PROPERTIES N=1 ! 1 F=3975.297 C=0.0 S=0.0 W=155/1728 !
LOCAL SYSTEM N=80
     1 R=1,81,1,82,1 G=1,1 !
MATERIAL MODELS N=1,1,1 P=10,2,5
     1 M=PRES ! N=1 C=196600,0.00715 ! N=2 C=220000,0.009 ! N=3 C=240000,0.0115
       N=4 C=245000,0.0135 ! N=5 C=270000,0.058
     1 M=CONC I=1 E=0.002,0.004 N=8
     1 M=MILD I=2 N=2 S=60000 E=2.069E-03
       N=1 C=60000,2.069E-03 ! N=2 C=60000.01,0.03 !
FSECTION PROPERTIES N=6 M=54,136 P=1
     1 C=+17,2 D=4,40.2 F=1.0 ; Bottom concrete flange
```

```
      N=1,0,0 G=1,2 S=1,0 C=1,5 B=5,0 I=1 F=2 !
   2 C=+4,0 D=13.33,15 ; Concrete web
      N=1,0,-7 G=1,7 S=1,0 C=1,2 B=2,0 I=14 F=2 !
   3 C=-7,0 D=5,37 ; Top concrete flange
      N=1,0,-5 G=1,2 S=1,0 C=1,11 B=11,0 I=1 F=2 !
   4 C=64,82 D=0.9822,0.9822 ; Bottom flange steel reinforcement
      N=1,0,0 ! N=2,0,+41 ! N=3,0,+82 ! N=4,0,+123 ! N=5,0,+164 !
   5 C=52,0 D=0.9439,0.9439 ; Web steel reinforcement
      N=01,000,-103 ! N=02,000,+103 ! N=03,021,-103 ! N=04,021,+103
      N=05,042,-103 ! N=06,042,+103 ! N=07,063,-103 ! N=08,063,+103
      N=09,084,-103 ! N=10,084,+103 ! N=11,105,-103 ! N=12,105,+103 !
   6 C=91,0 D=0.74,0.74 ; Top flange steel reinforcement
      N=01,139,-250 ! N=2,139,-200 ! N=3,139,-150 ! N=4,139,-100 ! N=05,139,-050
      N=06,139,-000 ! N=7,139,+050 ! N=8,139,+100 ! N=9,139,+150 ! N=10,139,+200
      N=11,139,+250 !
MILD STEEL PROPERTIES N=1 ! 1 D=1 !
FRAME ELEMENTS N=240
   001,1,2 D=0 C=1,1 S=1,4 L=1 J=0,0,0,0,1 G=001,080,1,1,1,0
   081,1,2 D=0 C=1,2 S=1,5 L=1 J=0,0,0,0,1 G=081,160,1,1,1,0
   161,1,2 D=0 C=1,3 S=1,6 L=1 J=0,0,0,0,1 G=161,240,1,1,1,0 !
PRESTRESSING STEEL N=1 ! 1 C=0.25 W=0.0002/12 !
TENDON GEOMETRY N=2
   1 S=1 M=1 A=13.16 F=-1
      1 N=81 G=1,81
     X=-3029.6008 Y=+0.000e+00 Z=-5246.6123 ! X=-2976.6121 Y=-0.677e+01 Z=-5276.8553
     X=-2923.3216 Y=-0.131e+02 Z=-5306.5631 ! X=-2869.7345 Y=-0.190e+02 Z=-5335.7329
     X=-2815.8565 Y=-0.244e+02 Z=-5364.3614 ! X=-2761.6929 Y=-0.294e+02 Z=-5392.4460
     X=-2707.2491 Y=-0.339e+02 Z=-5419.9837 ! X=-2652.5309 Y=-0.380e+02 Z=-5446.9718
     X=-2597.5436 Y=-0.417e+02 Z=-5473.4074 ! X=-2542.2929 Y=-0.449e+02 Z=-5499.2880
     X=-2486.7844 Y=-0.477e+02 Z=-5524.6108 ! X=-2431.0237 Y=-0.500e+02 Z=-5549.3734
     X=-2375.0165 Y=-0.519e+02 Z=-5573.5732 ! X=-2318.7684 Y=-0.533e+02 Z=-5597.2078
     X=-2262.2851 Y=-0.543e+02 Z=-5620.2747 ! X=-2205.5724 Y=-0.549e+02 Z=-5642.7716
     X=-2148.6361 Y=-0.550e+02 Z=-5664.6963 ! X=-2102.9298 Y=-0.546e+02 Z=-5681.8226
     X=-2057.0871 Y=-0.536e+02 Z=-5698.5800 ! X=-2011.1108 Y=-0.522e+02 Z=-5714.9676
     X=-1965.0040 Y=-0.502e+02 Z=-5730.9843 ! X=-1918.7697 Y=-0.477e+02 Z=-5746.6290
     X=-1872.4108 Y=-0.447e+02 Z=-5761.9007 ! X=-1825.9304 Y=-0.412e+02 Z=-5776.7984
     X=-1779.3315 Y=-0.371e+02 Z=-5791.3212 ! X=-1732.6171 Y=-0.326e+02 Z=-5805.4681
     X=-1685.7903 Y=-0.275e+02 Z=-5819.2382 ! X=-1638.8540 Y=-0.220e+02 Z=-5832.6305
     X=-1591.8114 Y=-0.159e+02 Z=-5845.6444 ! X=-1544.6654 Y=-0.937e+01 Z=-5858.2788
     X=-1497.4192 Y=-0.233e+01 Z=-5870.5330 ! X=-1450.0758 Y=+0.520e+01 Z=-5882.4061
     X=-1402.6383 Y=+0.132e+02 Z=-5893.8975 ! X=-1378.8851 Y=+0.169e+02 Z=-5899.4998
     X=-1355.1096 Y=+0.201e+02 Z=-5905.0063 ! X=-1331.3120 Y=+0.228e+02 Z=-5910.4171
     X=-1307.4929 Y=+0.251e+02 Z=-5915.7319 ! X=-1283.6526 Y=+0.268e+02 Z=-5920.9507
     X=-1259.7914 Y=+0.280e+02 Z=-5926.0735 ! X=-1235.9098 Y=+0.288e+02 Z=-5931.1001
     X=-1212.0081 Y=+0.290e+02 Z=-5936.0305 ! X=-1182.1034 Y=+0.287e+02 Z=-5942.0580
     X=-1152.1688 Y=+0.279e+02 Z=-5947.9348 ! X=-1122.2050 Y=+0.264e+02 Z=-5953.6608
     X=-1092.2127 Y=+0.245e+02 Z=-5959.2359 ! X=-1062.1927 Y=+0.219e+02 Z=-5964.6599
     X=-1032.1458 Y=+0.188e+02 Z=-5969.9327 ! X=-1002.0728 Y=+0.152e+02 Z=-5975.0541
     X=-971.97434 Y=+0.110e+02 Z=-5980.0240 ! X=-932.80966 Y=+0.560e+01 Z=-5986.2582
     X=-893.60502 Y=+0.415e+00 Z=-5992.2360 ! X=-854.36208 Y=-0.453e+01 Z=-5997.9569
     X=-815.08254 Y=-0.923e+01 Z=-6003.4209 ! X=-775.76807 Y=-0.137e+02 Z=-6008.6276
     X=-736.42036 Y=-0.179e+02 Z=-6013.5769 ! X=-697.04110 Y=-0.219e+02 Z=-6018.2685
     X=-657.63198 Y=-0.256e+02 Z=-6022.7022 ! X=-630.33211 Y=-0.281e+02 Z=-6025.6206
     X=-603.01931 Y=-0.304e+02 Z=-6028.4152 ! X=-575.69411 Y=-0.327e+02 Z=-6031.0860
     X=-548.35710 Y=-0.348e+02 Z=-6033.6329 ! X=-521.00882 Y=-0.368e+02 Z=-6036.0560
     X=-493.64985 Y=-0.387e+02 Z=-6038.3550 ! X=-466.28074 Y=-0.405e+02 Z=-6040.5300
     X=-438.90205 Y=-0.422e+02 Z=-6042.5811 ! X=-411.51434 Y=-0.438e+02 Z=-6044.5081
     X=-384.11819 Y=-0.452e+02 Z=-6046.3108 ! X=-356.71415 Y=-0.466e+02 Z=-6047.9895
     X=-329.30278 Y=-0.479e+02 Z=-6049.5439 ! X=-301.88465 Y=-0.490e+02 Z=-6050.9741
     X=-274.46032 Y=-0.501e+02 Z=-6052.2800 ! X=-247.03036 Y=-0.510e+02 Z=-6053.4616
```

```
X=-219.59532 Y=-0.519e+02 Z=-6054.5189 ! X=-192.15577 Y=-0.526e+02 Z=-6055.4519
X=-164.71227 Y=-0.532e+02 Z=-6056.2605 ! X=-137.26539 Y=-0.538e+02 Z=-6056.9448
X=-109.81570 Y=-0.542e+02 Z=-6057.5046 ! X=-82.363749 Y=-0.546e+02 Z=-6057.9401
X=-54.910106 Y=-0.548e+02 Z=-6058.2511 ! X=-27.455335 Y=-0.550e+02 Z=-6058.4377
X=-0.0000000 Y=-0.550e+02 Z=-6058.5000
2 S=1 M=1 A=13.16 F=-1
1 N=81 G=1,81
X=-2891.9922 Y=+0.000e+00 Z=-5093.7890 ! X=-2842.8858 Y=-0.677e+01 Z=-5121.3578
X=-2793.5165 Y=-0.131e+02 Z=-5148.4533 ! X=-2743.8890 Y=-0.190e+02 Z=-5175.0728
X=-2694.0079 Y=-0.244e+02 Z=-5201.2140 ! X=-2643.8778 Y=-0.294e+02 Z=-5226.8744
X=-2593.5033 Y=-0.339e+02 Z=-5252.0516 ! X=-2542.8890 Y=-0.380e+02 Z=-5276.7434
X=-2492.0397 Y=-0.417e+02 Z=-5300.9474 ! X=-2440.9600 Y=-0.449e+02 Z=-5324.6614
X=-2389.6548 Y=-0.477e+02 Z=-5347.8833 ! X=-2338.1286 Y=-0.500e+02 Z=-5370.6108
X=-2286.3863 Y=-0.519e+02 Z=-5392.8419 ! X=-2234.4326 Y=-0.533e+02 Z=-5414.5744
X=-2182.2725 Y=-0.543e+02 Z=-5435.8065 ! X=-2129.9106 Y=-0.549e+02 Z=-5456.5361
X=-2077.3518 Y=-0.550e+02 Z=-5476.7614 ! X=-2033.1619 Y=-0.546e+02 Z=-5493.3194
X=-1988.8401 Y=-0.536e+02 Z=-5509.5209 ! X=-1944.3891 Y=-0.522e+02 Z=-5525.3648
X=-1899.8120 Y=-0.502e+02 Z=-5540.8501 ! X=-1855.1116 Y=-0.477e+02 Z=-5555.9758
X=-1810.2907 Y=-0.447e+02 Z=-5570.7408 ! X=-1765.3524 Y=-0.412e+02 Z=-5585.1443
X=-1720.2995 Y=-0.371e+02 Z=-5599.1852 ! X=-1675.1349 Y=-0.326e+02 Z=-5612.8628
X=-1629.8616 Y=-0.275e+02 Z=-5626.1760 ! X=-1584.4825 Y=-0.220e+02 Z=-5639.1241
X=-1539.0006 Y=-0.159e+02 Z=-5651.7062 ! X=-1493.4188 Y=-0.937e+01 Z=-5663.9214
X=-1447.7400 Y=-0.233e+01 Z=-5675.7691 ! X=-1401.9673 Y=+0.520e+01 Z=-5687.2483
X=-1356.1036 Y=+0.132e+02 Z=-5698.3584 ! X=-1333.1385 Y=+0.169e+02 Z=-5703.7748
X=-1310.1517 Y=+0.201e+02 Z=-5709.0987 ! X=-1287.1437 Y=+0.228e+02 Z=-5714.3299
X=-1264.1148 Y=+0.251e+02 Z=-5719.4684 ! X=-1241.0654 Y=+0.268e+02 Z=-5724.5141
X=-1217.9959 Y=+0.280e+02 Z=-5729.4670 ! X=-1194.9066 Y=+0.288e+02 Z=-5734.3268
X=-1171.7979 Y=+0.290e+02 Z=-5739.0936 ! X=-1142.8853 Y=+0.287e+02 Z=-5744.9211
X=-1113.9439 Y=+0.279e+02 Z=-5750.6030 ! X=-1084.9741 Y=+0.264e+02 Z=-5756.1390
X=-1055.9769 Y=+0.245e+02 Z=-5761.5292 ! X=-1026.9529 Y=+0.219e+02 Z=-5766.7732
X=-997.90287 Y=+0.188e+02 Z=-5771.8711 ! X=-968.82752 Y=+0.152e+02 Z=-5776.8226
X=-939.72761 Y=+0.110e+02 Z=-5781.6276 ! X=-901.86228 Y=+0.560e+01 Z=-5787.6550
X=-863.95830 Y=+0.415e+00 Z=-5793.4344 ! X=-826.01731 Y=-0.453e+01 Z=-5798.9655
X=-788.04093 Y=-0.923e+01 Z=-5804.2482 ! X=-750.03078 Y=-0.137e+02 Z=-5809.2822
X=-711.98849 Y=-0.179e+02 Z=-5814.0673 ! X=-673.91570 Y=-0.219e+02 Z=-5818.6032
X=-635.81403 Y=-0.256e+02 Z=-5822.8898 ! X=-609.41988 Y=-0.281e+02 Z=-5825.7114
X=-583.01322 Y=-0.304e+02 Z=-5828.4133 ! X=-556.59458 Y=-0.327e+02 Z=-5830.9955
X=-530.16451 Y=-0.348e+02 Z=-5833.4579 ! X=-503.72356 Y=-0.368e+02 Z=-5835.8006
X=-477.27226 Y=-0.387e+02 Z=-5838.0234 ! X=-450.81116 Y=-0.405e+02 Z=-5840.1263
X=-424.34080 Y=-0.422e+02 Z=-5842.1093 ! X=-397.86173 Y=-0.438e+02 Z=-5843.9723
X=-371.37448 Y=-0.452e+02 Z=-5845.7152 ! X=-344.87961 Y=-0.466e+02 Z=-5847.3382
X=-318.37766 Y=-0.479e+02 Z=-5848.8410 ! X=-291.86917 Y=-0.490e+02 Z=-5850.2238
X=-265.35468 Y=-0.501e+02 Z=-5851.4864 ! X=-238.83475 Y=-0.510e+02 Z=-5852.6288
X=-212.30991 Y=-0.519e+02 Z=-5853.6510 ! X=-185.78071 Y=-0.526e+02 Z=-5854.5530
X=-159.24769 Y=-0.532e+02 Z=-5855.3348 ! X=-132.71140 Y=-0.538e+02 Z=-5855.9964
X=-106.17239 Y=-0.542e+02 Z=-5856.5376 ! X=-79.631206 Y=-0.546e+02 Z=-5856.9586
X=-53.088379 Y=-0.548e+02 Z=-5857.2594 ! X=-26.544462 Y=-0.550e+02 Z=-5857.4398
X=-0.0000000 Y=-0.550e+02 Z=-5857.5000 !
MESH COMPLETE !SET G=0,-1,0,0 D=28 C=0.05,0.05,1,0.01,0.01,10.0,0.5 N=1.0
CHANGE STRUCTURE
BUILD N=1,240,1 D=0
RESTRAINTS ! 1,81,01 R=0,0,0,0,0,0 ! 82 R=1,1,1,1,1,1 !
RESTRAINTS ! 1 R=0,1,0,0,0,0 ! 41 R=0,1,1,1,0,0 ! 81 R=1,0,0,0,1,1 !
STRESS N=1,2 F=2660000,0 D=0.25
CHANGE COMPLETE
SOLVE U=2 !OUTPUT P=0
LOADING ! N=57,73,8 F=0,-48000,0,0,0,0 ! N=81 F=0,-24000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=0
```

```
LOADING ! N=57,73,8 F=0,-48000,0,0,0,0 ! N=81 F=0,-24000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=0
LOADING ! N=57,73,8 F=0,-48000,0,0,0,0 ! N=81 F=0,-24000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=0
LOADING ! N=57,73,8 F=0,-24000,0,0,0,0 ! N=81 F=0,-12000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=0
LOADING ! N=57,73,8 F=0,-24000,0,0,0,0 ! N=81 F=0,-12000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=0
LOADING ! N=57,73,8 F=0,-24000,0,0,0,0 ! N=81 F=0,-12000,0,0,0,0 !
SOLVE U=2 !OUTPUT P=0
LOADING ! N=57,73,8 F=0,-12000,0,0,0,0 ! N=81 F=0,-6000,0,0,0,0 !
SOLVE U=2 I=100 !OUTPUT P=0
LOADING ! N=57,73,8 F=0,-12000,0,0,0,0 ! N=81 F=0,-6000,0,0,0,0 !
SOLVE U=2 I=100 !OUTPUT P=0
STOP
```

```
START ;
TITLE
EXAMPLE 5.3B - CHOUDHURY CURVED BRIDGE (ECCENTRIC LOADING I TO FAILURE)
              THREE SPAN CONTINUOUS POST-TENSIONED CURVED BOX GIRDER BRIDGE
              UNITS = LB-IN SYSTEM
CONCRETE PARAMETERS N=1 ! 1 M=ACI W=143.96 T=.6589
MESH INPUT
NODES N=82
    1 X=-2979.345 Z=-5159.580 !2  X=-2927.235 Z=-5189.321
    3 X=-2874.828 Z=-5218.536 !4  X=-2822.130 Z=-5247.222
    5 X=-2769.146 Z=-5275.376 !6  X=-2715.881 Z=-5302.994
    7 X=-2662.340 Z=-5330.075 !8  X=-2608.530 Z=-5356.615
    9 X=-2554.454 Z=-5382.613 !10 X=-2500.120 Z=-5408.064
   11 X=-2445.533 Z=-5432.967 !12 X=-2390.697 Z=-5457.318
   13 X=-2335.619 Z=-5481.117 !14 X=-2280.304 Z=-5504.359
   15 X=-2224.757 Z=-5527.044 !16 X=-2168.985 Z=-5549.167
   17 X=-2112.993 Z=-5570.728 !18 X=-2068.045 Z=-5587.571
   19 X=-2022.963 Z=-5604.050 !20 X=-1977.750 Z=-5620.166
   21 X=-1932.408 Z=-5635.917 !22 X=-1886.940 Z=-5651.302
   23 X=-1841.350 Z=-5666.320 !24 X=-1795.641 Z=-5680.971
   25 X=-1749.815 Z=-5695.253 !26 X=-1703.876 Z=-5709.165
   27 X=-1657.825 Z=-5722.707 !28 X=-1611.668 Z=-5735.877
   29 X=-1565.406 Z=-5748.675 !30 X=-1519.042 Z=-5761.100
   31 X=-1472.579 Z=-5773.151 !32 X=-1426.021 Z=-5784.827
   33 X=-1379.370 Z=-5796.127 !34 X=-1356.011 Z=-5801.637
   35 X=-1332.630 Z=-5807.052 !36 X=-1309.227 Z=-5812.373
   37 X=-1285.803 Z=-5817.600 !38 X=-1262.359 Z=-5822.732
   39 X=-1238.893 Z=-5827.770 !40 X=-1215.408 Z=-5832.713
   41 X=-1191.903 Z=-5837.562 !42 X=-1162.493 Z=-5843.489
   43 X=-1133.054 Z=-5849.269 !44 X=-1103.587 Z=-5854.900
   45 X=-1074.092 Z=-5860.383 !46 X=-1044.569 Z=-5865.717
   47 X=-1015.020 Z=-5870.902 !48 X=-985.4451 Z=-5875.939
   49 X=-955.8452 Z=-5880.826 !50 X=-917.3359 Z=-5886.956
   51 X=-878.7816 Z=-5892.835 !52 X=-840.1897 Z=-5898.461
   53 X=-801.5617 Z=-5903.834 !54 X=-762.8994 Z=-5908.954
   55 X=-724.2044 Z=-5913.822 !56 X=-685.4784 Z=-5918.435
   57 X=-646.7230 Z=-5922.796 !58 X=-619.8760 Z=-5925.666
   59 X=-593.0162 Z=-5928.414 !60 X=-566.1443 Z=-5931.040
   61 X=-539.2608 Z=-5933.545 !62 X=-512.3661 Z=-5935.928
   63 X=-485.4610 Z=-5938.189 !64 X=-458.5459 Z=-5940.328
   65 X=-431.6214 Z=-5942.345 !66 X=-404.6880 Z=-5944.240
   67 X=-377.7463 Z=-5946.013 !68 X=-350.7968 Z=-5947.663
   69 X=-323.8402 Z=-5949.192 !70 X=-296.8769 Z=-5950.598
   71 X=-269.9075 Z=-5951.883 !72 X=-242.9325 Z=-5953.045
   73 X=-215.9526 Z=-5954.085 !74 X=-188.9682 Z=-5955.002
   75 X=-161.9799 Z=-5955.797 !76 X=-134.9884 Z=-5956.470
   77 X=-107.9940 Z=-5957.021 !78 X=-80.99747 Z=-5957.449
   79 X=-53.99924 Z=-5957.755 !80 X=-26.99989 Z=-5957.938
   81 X=-00.00000 Z=-5958.000 !82 X=0 Y=0 Z=0 !
CONCRETE PROPERTIES N=1 ! 1 F=3975.297 C=0.0 S=0.0 W=155/1728 !
LOCAL SYSTEM N=80
    1 R=1,81,1,82,1 G=1,1 !
MATERIAL MODELS N=1,1,1 P=10,2,5
    1 M=PRES ! N=1 C=196600,0.00715 ! N=2 C=220000,0.009 ! N=3 C=240000,0.0115
      N=4 C=245000,0.0135 ! N=5 C=270000,0.058
    1 M=CONC I=1 E=0.002,0.004 N=8
    1 M=MILD I=2 N=2 S=60000 E=2.069E-03
      N=1 C=60000,2.069E-03 ! N=2 C=60000.01,0.03 !
FSECTION PROPERTIES N=6 M=54,136 P=1
    1 C=+17,2 D=4,40.2 F=1.0 ; Bottom concrete flange
```

```
      N=1,0,0 G=1,2 S=1,0 C=1,5 B=5,0 I=1 F=2 !
   2 C=+4,0 D=13.33,15 ; Concrete web
      N=1,0,-7 G=1,7 S=1,0 C=1,2 B=2,0 I=14 F=2 !
   3 C=-7,0 D=5,37 ; Top concrete flange
      N=1,0,-5 G=1,2 S=1,0 C=1,11 B=11,0 I=1 F=2 !
   4 C=64,82 D=0.9822,0.9822 ; Bottom flange steel reinforcement
      N=1,0,0 ! N=2,0,+41 ! N=3,0,+82 ! N=4,0,+123 ! N=5,0,+164 !
   5 C=52,0 D=0.9439,0.9439 ; Web steel reinforcement
      N=01,000,-103 ! N=02,000,+103 ! N=03,021,-103 ! N=04,021,+103
      N=05,042,-103 ! N=06,042,+103 ! N=07,063,-103 ! N=08,063,+103
      N=09,084,-103 ! N=10,084,+103 ! N=11,105,-103 ! N=12,105,+103 !
   6 C=91,0 D=0.74,0.74 ; Top flange steel reinforcement
      N=01,139,-250 ! N=2,139,-200 ! N=3,139,-150 ! N=4,139,-100 ! N=05,139,-050
      N=06,139,-000 ! N=7,139,+050 ! N=8,139,+100 ! N=9,139,+150 ! N=10,139,+200
      N=11,139,+250 !
MILD STEEL PROPERTIES N=1 ! 1 D=1 !
FRAME ELEMENTS N=240
   001,1,2 D=0 C=1,1 S=1,4 L=1 J=0,0,0,0,1 G=001,080,1,1,1,0
   081,1,2 D=0 C=1,2 S=1,5 L=1 J=0,0,0,0,1 G=081,160,1,1,1,0
   161,1,2 D=0 C=1,3 S=1,6 L=1 J=0,0,0,0,1 G=161,240,1,1,1,0 !
PRESTRESSING STEEL N=1 ! 1 C=0.25 W=0.0002/12 !
TENDON GEOMETRY N=2
   1 S=1 M=1 A=13.16 F=-1
      1 N=81 G=1,81
    X=-3029.6008 Y=+0.000e+00 Z=-5246.6123 ! X=-2976.6121 Y=-0.677e+01 Z=-5276.8553
    X=-2923.3216 Y=-0.131e+02 Z=-5306.5631 ! X=-2869.7345 Y=-0.190e+02 Z=-5335.7329
    X=-2815.8565 Y=-0.244e+02 Z=-5364.3614 ! X=-2761.6929 Y=-0.294e+02 Z=-5392.4460
    X=-2707.2491 Y=-0.339e+02 Z=-5419.9837 ! X=-2652.5309 Y=-0.380e+02 Z=-5446.9718
    X=-2597.5436 Y=-0.417e+02 Z=-5473.4074 ! X=-2542.2929 Y=-0.449e+02 Z=-5499.2880
    X=-2486.7844 Y=-0.477e+02 Z=-5524.6108 ! X=-2431.0237 Y=-0.500e+02 Z=-5549.3734
    X=-2375.0165 Y=-0.519e+02 Z=-5573.5732 ! X=-2318.7684 Y=-0.533e+02 Z=-5597.2078
    X=-2262.2851 Y=-0.543e+02 Z=-5620.2747 ! X=-2205.5724 Y=-0.549e+02 Z=-5642.7716
    X=-2148.6361 Y=-0.550e+02 Z=-5664.6963 ! X=-2102.9298 Y=-0.546e+02 Z=-5681.8226
    X=-2057.0871 Y=-0.536e+02 Z=-5698.5800 ! X=-2011.1108 Y=-0.522e+02 Z=-5714.9676
    X=-1965.0040 Y=-0.502e+02 Z=-5730.9843 ! X=-1918.7697 Y=-0.477e+02 Z=-5746.6290
    X=-1872.4108 Y=-0.447e+02 Z=-5761.9007 ! X=-1825.9304 Y=-0.412e+02 Z=-5776.7984
    X=-1779.3315 Y=-0.371e+02 Z=-5791.3212 ! X=-1732.6171 Y=-0.326e+02 Z=-5805.4681
    X=-1685.7903 Y=-0.275e+02 Z=-5819.2382 ! X=-1638.8540 Y=-0.220e+02 Z=-5832.6305
    X=-1591.8114 Y=-0.159e+02 Z=-5845.6444 ! X=-1544.6654 Y=-0.937e+01 Z=-5858.2788
    X=-1497.4192 Y=-0.233e+01 Z=-5870.5330 ! X=-1450.0758 Y=+0.520e+01 Z=-5882.4061
    X=-1402.6383 Y=+0.132e+02 Z=-5893.8975 ! X=-1378.8851 Y=+0.169e+02 Z=-5899.4998
    X=-1355.1096 Y=+0.201e+02 Z=-5905.0063 ! X=-1331.3120 Y=+0.228e+02 Z=-5910.4171
    X=-1307.4929 Y=+0.251e+02 Z=-5915.7319 ! X=-1283.6526 Y=+0.268e+02 Z=-5920.9507
    X=-1259.7914 Y=+0.280e+02 Z=-5926.0735 ! X=-1235.9098 Y=+0.288e+02 Z=-5931.1001
    X=-1212.0081 Y=+0.290e+02 Z=-5936.0305 ! X=-1182.1034 Y=+0.287e+02 Z=-5942.0580
    X=-1152.1688 Y=+0.279e+02 Z=-5947.9348 ! X=-1122.2050 Y=+0.264e+02 Z=-5953.6608
    X=-1092.2127 Y=+0.245e+02 Z=-5959.2359 ! X=-1062.1927 Y=+0.219e+02 Z=-5964.6599
    X=-1032.1458 Y=+0.188e+02 Z=-5969.9327 ! X=-1002.0728 Y=+0.152e+02 Z=-5975.0541
    X=-971.97434 Y=+0.110e+02 Z=-5980.0240 ! X=-932.80966 Y=+0.560e+01 Z=-5986.2582
    X=-893.60502 Y=+0.415e+00 Z=-5992.2360 ! X=-854.36208 Y=-0.453e+01 Z=-5997.9569
    X=-815.08254 Y=-0.923e+01 Z=-6003.4209 ! X=-775.76807 Y=-0.137e+02 Z=-6008.6276
    X=-736.42036 Y=-0.179e+02 Z=-6013.5769 ! X=-697.04110 Y=-0.219e+02 Z=-6018.2685
    X=-657.63198 Y=-0.256e+02 Z=-6022.7022 ! X=-630.33211 Y=-0.281e+02 Z=-6025.6206
    X=-603.01931 Y=-0.304e+02 Z=-6028.4152 ! X=-575.69411 Y=-0.327e+02 Z=-6031.0860
    X=-548.35710 Y=-0.348e+02 Z=-6033.6329 ! X=-521.00882 Y=-0.368e+02 Z=-6036.0560
    X=-493.64985 Y=-0.387e+02 Z=-6038.3550 ! X=-466.28074 Y=-0.405e+02 Z=-6040.5300
    X=-438.90205 Y=-0.422e+02 Z=-6042.5811 ! X=-411.51434 Y=-0.438e+02 Z=-6044.5081
    X=-384.11819 Y=-0.452e+02 Z=-6046.3108 ! X=-356.71415 Y=-0.466e+02 Z=-6047.9895
    X=-329.30278 Y=-0.479e+02 Z=-6049.5439 ! X=-301.88465 Y=-0.490e+02 Z=-6050.9741
    X=-274.46032 Y=-0.501e+02 Z=-6052.2800 ! X=-247.03036 Y=-0.510e+02 Z=-6053.4616
```

```
        X=-219.59532 Y=-0.519e+02 Z=-6054.5189 ! X=-192.15577 Y=-0.526e+02 Z=-6055.4519
        X=-164.71227 Y=-0.532e+02 Z=-6056.2605 ! X=-137.26539 Y=-0.538e+02 Z=-6056.9448
        X=-109.81570 Y=-0.542e+02 Z=-6057.5046 ! X=-82.363749 Y=-0.546e+02 Z=-6057.9401
        X=-54.910106 Y=-0.548e+02 Z=-6058.2511 ! X=-27.455335 Y=-0.550e+02 Z=-6058.4377
        X=-0.0000000 Y=-0.550e+02 Z=-6058.5000
      2 S=1 M=1 A=13.16 F=-1
         1 N=81 G=1,81
        X=-2891.9922 Y=+0.000e+00 Z=-5093.7890 ! X=-2842.8858 Y=-0.677e+01 Z=-5121.3578
        X=-2793.5165 Y=-0.131e+02 Z=-5148.4533 ! X=-2743.8890 Y=-0.190e+02 Z=-5175.0728
        X=-2694.0079 Y=-0.244e+02 Z=-5201.2140 ! X=-2643.8778 Y=-0.294e+02 Z=-5226.8744
        X=-2593.5033 Y=-0.339e+02 Z=-5252.0516 ! X=-2542.8890 Y=-0.380e+02 Z=-5276.7434
        X=-2492.0397 Y=-0.417e+02 Z=-5300.9474 ! X=-2440.9600 Y=-0.449e+02 Z=-5324.6614
        X=-2389.6548 Y=-0.477e+02 Z=-5347.8833 ! X=-2338.1286 Y=-0.500e+02 Z=-5370.6108
        X=-2286.3863 Y=-0.519e+02 Z=-5392.8419 ! X=-2234.4326 Y=-0.533e+02 Z=-5414.5744
        X=-2182.2725 Y=-0.543e+02 Z=-5435.8065 ! X=-2129.9106 Y=-0.549e+02 Z=-5456.5361
        X=-2077.3518 Y=-0.550e+02 Z=-5476.7614 ! X=-2033.1619 Y=-0.546e+02 Z=-5493.3194
        X=-1988.8401 Y=-0.536e+02 Z=-5509.5209 ! X=-1944.3891 Y=-0.522e+02 Z=-5525.3648
        X=-1899.8120 Y=-0.502e+02 Z=-5540.8501 ! X=-1855.1116 Y=-0.477e+02 Z=-5555.9758
        X=-1810.2907 Y=-0.447e+02 Z=-5570.7408 ! X=-1765.3524 Y=-0.412e+02 Z=-5585.1443
        X=-1720.2995 Y=-0.371e+02 Z=-5599.1852 ! X=-1675.1349 Y=-0.326e+02 Z=-5612.8628
        X=-1629.8616 Y=-0.275e+02 Z=-5626.1760 ! X=-1584.4825 Y=-0.220e+02 Z=-5639.1241
        X=-1539.0006 Y=-0.159e+02 Z=-5651.7062 ! X=-1493.4188 Y=-0.937e+01 Z=-5663.9214
        X=-1447.7400 Y=-0.233e+01 Z=-5675.7691 ! X=-1401.9673 Y=+0.520e+01 Z=-5687.2483
        X=-1356.1036 Y=+0.132e+02 Z=-5698.3584 ! X=-1333.1385 Y=+0.169e+02 Z=-5703.7748
        X=-1310.1517 Y=+0.201e+02 Z=-5709.0987 ! X=-1287.1437 Y=+0.228e+02 Z=-5714.3299
        X=-1264.1148 Y=+0.251e+02 Z=-5719.4684 ! X=-1241.0654 Y=+0.268e+02 Z=-5724.5141
        X=-1217.9959 Y=+0.280e+02 Z=-5729.4670 ! X=-1194.9066 Y=+0.288e+02 Z=-5734.3268
        X=-1171.7979 Y=+0.290e+02 Z=-5739.0936 ! X=-1142.8853 Y=+0.287e+02 Z=-5744.9211
        X=-1113.9439 Y=+0.279e+02 Z=-5750.6030 ! X=-1084.9741 Y=+0.264e+02 Z=-5756.1390
        X=-1055.9769 Y=+0.245e+02 Z=-5761.5292 ! X=-1026.9529 Y=+0.219e+02 Z=-5766.7732
        X=-997.90287 Y=+0.188e+02 Z=-5771.8711 ! X=-968.82752 Y=+0.152e+02 Z=-5776.8226
        X=-939.72761 Y=+0.110e+02 Z=-5781.6276 ! X=-901.86228 Y=+0.560e+01 Z=-5787.6550
        X=-863.95830 Y=+0.415e+00 Z=-5793.4344 ! X=-826.01731 Y=-0.453e+01 Z=-5798.9655
        X=-788.04093 Y=-0.923e+01 Z=-5804.2482 ! X=-750.03078 Y=-0.137e+02 Z=-5809.2822
        X=-711.98849 Y=-0.179e+02 Z=-5814.0673 ! X=-673.91570 Y=-0.219e+02 Z=-5818.6032
        X=-635.81403 Y=-0.256e+02 Z=-5822.8898 ! X=-609.41988 Y=-0.281e+02 Z=-5825.7114
        X=-583.01322 Y=-0.304e+02 Z=-5828.4133 ! X=-556.59458 Y=-0.327e+02 Z=-5830.9955
        X=-530.16451 Y=-0.348e+02 Z=-5833.4579 ! X=-503.72356 Y=-0.368e+02 Z=-5835.8006
        X=-477.27226 Y=-0.387e+02 Z=-5838.0234 ! X=-450.81116 Y=-0.405e+02 Z=-5840.1263
        X=-424.34080 Y=-0.422e+02 Z=-5842.1093 ! X=-397.86173 Y=-0.438e+02 Z=-5843.9723
        X=-371.37448 Y=-0.452e+02 Z=-5845.7152 ! X=-344.87961 Y=-0.466e+02 Z=-5847.3382
        X=-318.37766 Y=-0.479e+02 Z=-5848.8410 ! X=-291.86917 Y=-0.490e+02 Z=-5850.2238
        X=-265.35468 Y=-0.501e+02 Z=-5851.4864 ! X=-238.83475 Y=-0.510e+02 Z=-5852.6288
        X=-212.30991 Y=-0.519e+02 Z=-5853.6510 ! X=-185.78071 Y=-0.526e+02 Z=-5854.5530
        X=-159.24769 Y=-0.532e+02 Z=-5855.3348 ! X=-132.71140 Y=-0.538e+02 Z=-5855.9964
        X=-106.17239 Y=-0.542e+02 Z=-5856.5376 ! X=-79.631206 Y=-0.546e+02 Z=-5856.9586
        X=-53.088379 Y=-0.548e+02 Z=-5857.2594 ! X=-26.544462 Y=-0.550e+02 Z=-5857.4398
        X=-0.0000000 Y=-0.550e+02 Z=-5857.5000 !
MESH COMPLETE !SET G=0,-1,0,0 D=28 C=0.05,0.05,1,0.01,0.01,10.0,0.5 N=1.0
CHANGE STRUCTURE
BUILD N=1,240,1 D=0
RESTRAINTS ! 1,81,01 R=0,0,0,0,0,0 ! 82 R=1,1,1,1,1,1 !
RESTRAINTS ! 1 R=0,1,0,0,0,0 ! 41 R=0,1,1,1,0,0 ! 81 R=1,0,0,0,1,1 !
STRESS N=1,2 F=2660000,0 D=0.25
CHANGE COMPLETE
SOLVE U=2 !OUTPUT P=0
LOADING
      N=57 F=0,-48000,0,47.72E+5,0,-5.2E+5
      N=65 F=0,-48000,0,47.87E+5,0,-3.5E+5
      N=73 F=0,-48000,0,47.97E+5,0,-1.7E+5
```

```
        N=81 F=0,-24000,0,24.00E+5,0,-000000 !
        SOLVE U=2 !OUTPUT P=0
LOADING
        N=57 F=0,-48000,0,47.72E+5,0,-5.2E+5
        N=65 F=0,-48000,0,47.87E+5,0,-3.5E+5
        N=73 F=0,-48000,0,47.97E+5,0,-1.7E+5
        N=81 F=0,-24000,0,24.00E+5,0,-000000 !
SOLVE U=2 !OUTPUT P=0
LOADING
        N=57 F=0,-48000,0,47.72E+5,0,-5.2E+5
        N=65 F=0,-48000,0,47.87E+5,0,-3.5E+5
        N=73 F=0,-48000,0,47.97E+5,0,-1.7E+5
        N=81 F=0,-24000,0,24.00E+5,0,-000000 !
SOLVE U=2 !OUTPUT P=0
LOADING
        N=57 F=0,-24000,0,23.86E+5,0,-2.6E+5
        N=65 F=0,-24000,0,23.94E+5,0,-1.8E+5
        N=73 F=0,-24000,0,23.99E+5,0,-0.9E+5
        N=81 F=0,-12000,0,12.00E+5,0,-000000 !
SOLVE U=2 !OUTPUT P=0
LOADING
        N=57 F=0,-24000,0,23.86E+5,0,-2.6E+5
        N=65 F=0,-24000,0,23.94E+5,0,-1.8E+5
        N=73 F=0,-24000,0,23.99E+5,0,-0.9E+5
        N=81 F=0,-12000,0,12.00E+5,0,-000000 !
SOLVE U=2 !OUTPUT P=0
LOADING
        N=57 F=0,-24000,0,23.86E+5,0,-2.6E+5
        N=65 F=0,-24000,0,23.94E+5,0,-1.8E+5
        N=73 F=0,-24000,0,23.99E+5,0,-0.9E+5
        N=81 F=0,-12000,0,12.00E+5,0,-000000 !
SOLVE U=2 !OUTPUT P=0
LOADING
        N=57 F=0,-12000,0,11.93E+5,0,-1.3E+5
        N=65 F=0,-12000,0,11.97E+5,0,-0.9E+5
        N=73 F=0,-12000,0,11.99E+5,0,-0.4E+5
        N=81 F=0,-06000,0,12.00E+5,0,-000000 !
SOLVE U=2 I=100 !OUTPUT P=0
LOADING
        N=57 F=0,-12000,0,11.93E+5,0,-1.3E+5
        N=65 F=0,-12000,0,11.97E+5,0,-0.9E+5
        N=73 F=0,-12000,0,11.99E+5,0,-0.4E+5
        N=81 F=0,-06000,0,12.00E+5,0,-000000 !
SOLVE U=2 I=100 !OUTPUT P=0
STOP
```

```
START ;
TITLE
EXAMPLE 5.3C  - CHOUDHURY CURVED BRIDGE (ECCENTRIC LOADING II TO FAILURE)
                THREE SPAN CONTINUOUS POST-TENSIONED CURVED BOX GIRDER BRIDGE
                UNITS = LB-IN SYSTEM
CONCRETE PARAMETERS N=1 ! 1 M=ACI W=143.96 T=.6589
MESH INPUT
NODES N=82
      1 X=-2979.345 Z=-5159.580 !2  X=-2927.235 Z=-5189.321
      3 X=-2874.828 Z=-5218.536 !4  X=-2822.130 Z=-5247.222
      5 X=-2769.146 Z=-5275.376 !6  X=-2715.881 Z=-5302.994
      7 X=-2662.340 Z=-5330.075 !8  X=-2608.530 Z=-5356.615
      9 X=-2554.454 Z=-5382.613 !10 X=-2500.120 Z=-5408.064
     11 X=-2445.533 Z=-5432.967 !12 X=-2390.697 Z=-5457.318
     13 X=-2335.619 Z=-5481.117 !14 X=-2280.304 Z=-5504.359
     15 X=-2224.757 Z=-5527.044 !16 X=-2168.985 Z=-5549.167
     17 X=-2112.993 Z=-5570.728 !18 X=-2068.045 Z=-5587.571
     19 X=-2022.963 Z=-5604.050 !20 X=-1977.750 Z=-5620.166
     21 X=-1932.408 Z=-5635.917 !22 X=-1886.940 Z=-5651.302
     23 X=-1841.350 Z=-5666.320 !24 X=-1795.641 Z=-5680.971
     25 X=-1749.815 Z=-5695.253 !26 X=-1703.876 Z=-5709.165
     27 X=-1657.825 Z=-5722.707 !28 X=-1611.668 Z=-5735.877
     29 X=-1565.406 Z=-5748.675 !30 X=-1519.042 Z=-5761.100
     31 X=-1472.579 Z=-5773.151 !32 X=-1426.021 Z=-5784.827
     33 X=-1379.370 Z=-5796.127 !34 X=-1356.011 Z=-5801.637
     35 X=-1332.630 Z=-5807.052 !36 X=-1309.227 Z=-5812.373
     37 X=-1285.803 Z=-5817.600 !38 X=-1262.359 Z=-5822.732
     39 X=-1238.893 Z=-5827.770 !40 X=-1215.408 Z=-5832.713
     41 X=-1191.903 Z=-5837.562 !42 X=-1162.493 Z=-5843.489
     43 X=-1133.054 Z=-5849.269 !44 X=-1103.587 Z=-5854.900
     45 X=-1074.092 Z=-5860.383 !46 X=-1044.569 Z=-5865.717
     47 X=-1015.020 Z=-5870.902 !48 X=-985.4451 Z=-5875.939
     49 X=-955.8452 Z=-5880.826 !50 X=-917.3359 Z=-5886.956
     51 X=-878.7816 Z=-5892.835 !52 X=-840.1897 Z=-5898.461
     53 X=-801.5617 Z=-5903.834 !54 X=-762.8994 Z=-5908.954
     55 X=-724.2044 Z=-5913.822 !56 X=-685.4784 Z=-5918.435
     57 X=-646.7230 Z=-5922.796 !58 X=-619.8760 Z=-5925.666
     59 X=-593.0162 Z=-5928.414 !60 X=-566.1443 Z=-5931.040
     61 X=-539.2608 Z=-5933.545 !62 X=-512.3661 Z=-5935.928
     63 X=-485.4610 Z=-5938.189 !64 X=-458.5459 Z=-5940.328
     65 X=-431.6214 Z=-5942.345 !66 X=-404.6880 Z=-5944.240
     67 X=-377.7463 Z=-5946.013 !68 X=-350.7968 Z=-5947.663
     69 X=-323.8402 Z=-5949.192 !70 X=-296.8769 Z=-5950.598
     71 X=-269.9075 Z=-5951.883 !72 X=-242.9325 Z=-5953.045
     73 X=-215.9526 Z=-5954.085 !74 X=-188.9682 Z=-5955.002
     75 X=-161.9799 Z=-5955.797 !76 X=-134.9884 Z=-5956.470
     77 X=-107.9940 Z=-5957.021 !78 X=-80.99747 Z=-5957.449
     79 X=-53.99924 Z=-5957.755 !80 X=-26.99989 Z=-5957.938
     81 X=-00.00000 Z=-5958.000 !82 X=0 Y=0 Z=0 !
CONCRETE PROPERTIES N=1 ! 1 F=3975.297 C=0.0 S=0.0 W=155/1728 !
LOCAL SYSTEM N=80
      1 R=1,81,1,82,1 G=1,1 !
MATERIAL MODELS N=1,1,1 P=10,2,5
      1 M=PRES ! N=1 C=196600,0.00715 ! N=2 C=220000,0.009 ! N=3 C=240000,0.0115
        N=4 C=245000,0.0135 ! N=5 C=270000,0.058
      1 M=CONC I=1 E=0.002,0.004 N=8
      1 M=MILD I=2 N=2 S=60000 E=2.069E-03
        N=1 C=60000,2.069E-03 ! N=2 C=60000.01,0.03 !
FSECTION PROPERTIES N=6 M=54,136 P=1
      1 C=+17,2 D=4,40.2 F=1.0 ; Bottom concrete flange
```

```
        N=1,0,0 G=1,2 S=1,0 C=1,5 B=5,0 I=1 F=2 !
    2 C=+4,0 D=13.33,15 ; Concrete web
        N=1,0,-7 G=1,7 S=1,0 C=1,2 B=2,0 I=14 F=2 !
    3 C=-7,0 D=5,37 ; Top concrete flange
        N=1,0,-5 G=1,2 S=1,0 C=1,11 B=11,0 I=1 F=2 !
    4 C=64,82 D=0.9822,0.9822 ; Bottom flange steel reinforcement
        N=1,0,0 ! N=2,0,+41 ! N=3,0,+82 ! N=4,0,+123 ! N=5,0,+164 !
    5 C=52,0 D=0.9439,0.9439 ; Web steel reinforcement
        N=01,000,-103 ! N=02,000,+103 ! N=03,021,-103 ! N=04,021,+103
        N=05,042,-103 ! N=06,042,+103 ! N=07,063,-103 ! N=08,063,+103
        N=09,084,-103 ! N=10,084,+103 ! N=11,105,-103 ! N=12,105,+103 !
    6 C=91,0 D=0.74,0.74 ; Top flange steel reinforcement
        N=01,139,-250 ! N=2,139,-200 ! N=3,139,-150 ! N=4,139,-100 ! N=05,139,-050
        N=06,139,-000 ! N=7,139,+050 ! N=8,139,+100 ! N=9,139,+150 ! N=10,139,+200
        N=11,139,+250 !
MILD STEEL PROPERTIES N=1 ! 1 D=1 !
FRAME ELEMENTS N=240
    001,1,2 D=0 C=1,1 S=1,4 L=1 J=0,0,0,0,1 G=001,080,1,1,1,0
    081,1,2 D=0 C=1,2 S=1,5 L=1 J=0,0,0,0,1 G=081,160,1,1,1,0
    161,1,2 D=0 C=1,3 S=1,6 L=1 J=0,0,0,0,1 G=161,240,1,1,1,0 !
PRESTRESSING STEEL N=1 ! 1 C=0.25 W=0.0002/12 !
TENDON GEOMETRY N=2
    1 S=1 M=1 A=13.16 F=-1
        1 N=81 G=1,81
      X=-3029.6008 Y=+0.000e+00 Z=-5246.6123 ! X=-2976.6121 Y=-0.677e+01 Z=-5276.8553
      X=-2923.3216 Y=-0.131e+02 Z=-5306.5631 ! X=-2869.7345 Y=-0.190e+02 Z=-5335.7329
      X=-2815.8565 Y=-0.244e+02 Z=-5364.3614 ! X=-2761.6929 Y=-0.294e+02 Z=-5392.4460
      X=-2707.2491 Y=-0.339e+02 Z=-5419.9837 ! X=-2652.5309 Y=-0.380e+02 Z=-5446.9718
      X=-2597.5436 Y=-0.417e+02 Z=-5473.4074 ! X=-2542.2929 Y=-0.449e+02 Z=-5499.2880
      X=-2486.7844 Y=-0.477e+02 Z=-5524.6108 ! X=-2431.0237 Y=-0.500e+02 Z=-5549.3734
      X=-2375.0165 Y=-0.519e+02 Z=-5573.5732 ! X=-2318.7684 Y=-0.533e+02 Z=-5597.2078
      X=-2262.2851 Y=-0.543e+02 Z=-5620.2747 ! X=-2205.5724 Y=-0.549e+02 Z=-5642.7716
      X=-2148.6361 Y=-0.550e+02 Z=-5664.6963 ! X=-2102.9298 Y=-0.546e+02 Z=-5681.8226
      X=-2057.0871 Y=-0.536e+02 Z=-5698.5800 ! X=-2011.1108 Y=-0.522e+02 Z=-5714.9676
      X=-1965.0040 Y=-0.502e+02 Z=-5730.9843 ! X=-1918.7697 Y=-0.477e+02 Z=-5746.6290
      X=-1872.4108 Y=-0.447e+02 Z=-5761.9007 ! X=-1825.9304 Y=-0.412e+02 Z=-5776.7984
      X=-1779.3315 Y=-0.371e+02 Z=-5791.3212 ! X=-1732.6171 Y=-0.326e+02 Z=-5805.4681
      X=-1685.7903 Y=-0.275e+02 Z=-5819.2382 ! X=-1638.8540 Y=-0.220e+02 Z=-5832.6305
      X=-1591.8114 Y=-0.159e+02 Z=-5845.6444 ! X=-1544.6654 Y=-0.937e+01 Z=-5858.2788
      X=-1497.4192 Y=-0.233e+01 Z=-5870.5330 ! X=-1450.0758 Y=+0.520e+01 Z=-5882.4061
      X=-1402.6383 Y=+0.132e+02 Z=-5893.8975 ! X=-1378.8851 Y=+0.169e+02 Z=-5899.4998
      X=-1355.1096 Y=+0.201e+02 Z=-5905.0063 ! X=-1331.3120 Y=+0.228e+02 Z=-5910.4171
      X=-1307.4929 Y=+0.251e+02 Z=-5915.7319 ! X=-1283.6526 Y=+0.268e+02 Z=-5920.9507
      X=-1259.7914 Y=+0.280e+02 Z=-5926.0735 ! X=-1235.9098 Y=+0.288e+02 Z=-5931.1001
      X=-1212.0081 Y=+0.290e+02 Z=-5936.0305 ! X=-1182.1034 Y=+0.287e+02 Z=-5942.0580
      X=-1152.1688 Y=+0.279e+02 Z=-5947.9348 ! X=-1122.2050 Y=+0.264e+02 Z=-5953.6608
      X=-1092.2127 Y=+0.245e+02 Z=-5959.2359 ! X=-1062.1927 Y=+0.219e+02 Z=-5964.6599
      X=-1032.1458 Y=+0.188e+02 Z=-5969.9327 ! X=-1002.0728 Y=+0.152e+02 Z=-5975.0541
      X=-971.97434 Y=+0.110e+02 Z=-5980.0240 ! X=-932.80966 Y=+0.560e+01 Z=-5986.2582
      X=-893.60502 Y=+0.415e+00 Z=-5992.2360 ! X=-854.36208 Y=-0.453e+01 Z=-5997.9569
      X=-815.08254 Y=-0.923e+01 Z=-6003.4209 ! X=-775.76807 Y=-0.137e+02 Z=-6008.6276
      X=-736.42036 Y=-0.179e+02 Z=-6013.5769 ! X=-697.04110 Y=-0.219e+02 Z=-6018.2685
      X=-657.63198 Y=-0.256e+02 Z=-6022.7022 ! X=-630.33211 Y=-0.281e+02 Z=-6025.6206
      X=-603.01931 Y=-0.304e+02 Z=-6028.4152 ! X=-575.69411 Y=-0.327e+02 Z=-6031.0860
      X=-548.35710 Y=-0.348e+02 Z=-6033.6329 ! X=-521.00882 Y=-0.368e+02 Z=-6036.0560
      X=-493.64985 Y=-0.387e+02 Z=-6038.3550 ! X=-466.28074 Y=-0.405e+02 Z=-6040.5300
      X=-438.90205 Y=-0.422e+02 Z=-6042.5811 ! X=-411.51434 Y=-0.438e+02 Z=-6044.5081
      X=-384.11819 Y=-0.452e+02 Z=-6046.3108 ! X=-356.71415 Y=-0.466e+02 Z=-6047.9895
      X=-329.30278 Y=-0.479e+02 Z=-6049.5439 ! X=-301.88465 Y=-0.490e+02 Z=-6050.9741
      X=-274.46032 Y=-0.501e+02 Z=-6052.2800 ! X=-247.03036 Y=-0.510e+02 Z=-6053.4616
```

```
      X=-219.59532 Y=-0.519e+02 Z=-6054.5189 ! X=-192.15577 Y=-0.526e+02 Z=-6055.4519
      X=-164.71227 Y=-0.532e+02 Z=-6056.2605 ! X=-137.26539 Y=-0.538e+02 Z=-6056.9448
      X=-109.81570 Y=-0.542e+02 Z=-6057.5046 ! X=-82.363749 Y=-0.546e+02 Z=-6057.9401
      X=-54.910106 Y=-0.548e+02 Z=-6058.2511 ! X=-27.455335 Y=-0.550e+02 Z=-6058.4377
      X=-0.0000000 Y=-0.550e+02 Z=-6058.5000
    2 S=1 M=1 A=13.16 F=-1
      1 N=81 G=1,81
      X=-2891.9922 Y=+0.000e+00 Z=-5093.7890 ! X=-2842.8858 Y=-0.677e+01 Z=-5121.3578
      X=-2793.5165 Y=-0.131e+02 Z=-5148.4533 ! X=-2743.8890 Y=-0.190e+02 Z=-5175.0728
      X=-2694.0079 Y=-0.244e+02 Z=-5201.2140 ! X=-2643.8778 Y=-0.294e+02 Z=-5226.8744
      X=-2593.5033 Y=-0.339e+02 Z=-5252.0516 ! X=-2542.8890 Y=-0.380e+02 Z=-5276.7434
      X=-2492.0397 Y=-0.417e+02 Z=-5300.9474 ! X=-2440.9600 Y=-0.449e+02 Z=-5324.6614
      X=-2389.6548 Y=-0.477e+02 Z=-5347.8833 ! X=-2338.1286 Y=-0.500e+02 Z=-5370.6108
      X=-2286.3863 Y=-0.519e+02 Z=-5392.8419 ! X=-2234.4326 Y=-0.533e+02 Z=-5414.5744
      X=-2182.2725 Y=-0.543e+02 Z=-5435.8065 ! X=-2129.9106 Y=-0.549e+02 Z=-5456.5361
      X=-2077.3518 Y=-0.550e+02 Z=-5476.7614 ! X=-2033.1619 Y=-0.546e+02 Z=-5493.3194
      X=-1988.8401 Y=-0.536e+02 Z=-5509.5209 ! X=-1944.3891 Y=-0.522e+02 Z=-5525.3648
      X=-1899.8120 Y=-0.502e+02 Z=-5540.8501 ! X=-1855.1116 Y=-0.477e+02 Z=-5555.9758
      X=-1810.2907 Y=-0.447e+02 Z=-5570.7408 ! X=-1765.3524 Y=-0.412e+02 Z=-5585.1443
      X=-1720.2995 Y=-0.371e+02 Z=-5599.1852 ! X=-1675.1349 Y=-0.326e+02 Z=-5612.8628
      X=-1629.8616 Y=-0.275e+02 Z=-5626.1760 ! X=-1584.4825 Y=-0.220e+02 Z=-5639.1241
      X=-1539.0006 Y=-0.159e+02 Z=-5651.7062 ! X=-1493.4188 Y=-0.937e+01 Z=-5663.9214
      X=-1447.7400 Y=-0.233e+01 Z=-5675.7691 ! X=-1401.9673 Y=+0.520e+01 Z=-5687.2483
      X=-1356.1036 Y=+0.132e+02 Z=-5698.3584 ! X=-1333.1385 Y=+0.169e+02 Z=-5703.7748
      X=-1310.1517 Y=+0.201e+02 Z=-5709.0987 ! X=-1287.1437 Y=+0.228e+02 Z=-5714.3299
      X=-1264.1148 Y=+0.251e+02 Z=-5719.4684 ! X=-1241.0654 Y=+0.268e+02 Z=-5724.5141
      X=-1217.9959 Y=+0.280e+02 Z=-5729.4670 ! X=-1194.9066 Y=+0.288e+02 Z=-5734.3268
      X=-1171.7979 Y=+0.290e+02 Z=-5739.0936 ! X=-1142.8853 Y=+0.287e+02 Z=-5744.9211
      X=-1113.9439 Y=+0.279e+02 Z=-5750.6030 ! X=-1084.9741 Y=+0.264e+02 Z=-5756.1390
      X=-1055.9769 Y=+0.245e+02 Z=-5761.5292 ! X=-1026.9529 Y=+0.219e+02 Z=-5766.7732
      X=-997.90287 Y=+0.188e+02 Z=-5771.8711 ! X=-968.82752 Y=+0.152e+02 Z=-5776.8226
      X=-939.72761 Y=+0.110e+02 Z=-5781.6276 ! X=-901.86228 Y=+0.560e+01 Z=-5787.6550
      X=-863.95830 Y=+0.415e+00 Z=-5793.4344 ! X=-826.01731 Y=-0.453e+01 Z=-5798.9655
      X=-788.04093 Y=-0.923e+01 Z=-5804.2482 ! X=-750.03078 Y=-0.137e+02 Z=-5809.2822
      X=-711.98849 Y=-0.179e+02 Z=-5814.0673 ! X=-673.91570 Y=-0.219e+02 Z=-5818.6032
      X=-635.81403 Y=-0.256e+02 Z=-5822.8898 ! X=-609.41988 Y=-0.281e+02 Z=-5825.7114
      X=-583.01322 Y=-0.304e+02 Z=-5828.4133 ! X=-556.59458 Y=-0.327e+02 Z=-5830.9955
      X=-530.16451 Y=-0.348e+02 Z=-5833.4579 ! X=-503.72356 Y=-0.368e+02 Z=-5835.8006
      X=-477.27226 Y=-0.387e+02 Z=-5838.0234 ! X=-450.81116 Y=-0.405e+02 Z=-5840.1263
      X=-424.34080 Y=-0.422e+02 Z=-5842.1093 ! X=-397.86173 Y=-0.438e+02 Z=-5843.9723
      X=-371.37448 Y=-0.452e+02 Z=-5845.7152 ! X=-344.87961 Y=-0.466e+02 Z=-5847.3382
      X=-318.37766 Y=-0.479e+02 Z=-5848.8410 ! X=-291.86917 Y=-0.490e+02 Z=-5850.2238
      X=-265.35468 Y=-0.501e+02 Z=-5851.4864 ! X=-238.83475 Y=-0.510e+02 Z=-5852.6288
      X=-212.30991 Y=-0.519e+02 Z=-5853.6510 ! X=-185.78071 Y=-0.526e+02 Z=-5854.5530
      X=-159.24769 Y=-0.532e+02 Z=-5855.3348 ! X=-132.71140 Y=-0.538e+02 Z=-5855.9964
      X=-106.17239 Y=-0.542e+02 Z=-5856.5376 ! X=-79.631206 Y=-0.546e+02 Z=-5856.9586
      X=-53.088379 Y=-0.548e+02 Z=-5857.2594 ! X=-26.544462 Y=-0.550e+02 Z=-5857.4398
      X=-0.0000000 Y=-0.550e+02 Z=-5857.5000 !
MESH COMPLETE !SET G=0,-1,0,0 D=28 C=0.05,0.05,1,0.01,0.01,10.0,0.5 N=1.0
CHANGE STRUCTURE
BUILD N=1,240,1 D=0
RESTRAINTS ! 1,81,01 R=0,0,0,0,0,0 ! 82 R=1,1,1,1,1,1 !
RESTRAINTS ! 1 R=0,1,0,0,0,0 ! 41 R=0,1,1,1,0,0 ! 81 R=1,0,0,0,1,1 !
STRESS N=1,2 F=2660000,0 D=0.25
CHANGE COMPLETE
SOLVE U=2 !OUTPUT P=0
LOADING
    N=57 F=0,-48000,0,-47.72E+5,0,+5.2E+5
    N=65 F=0,-48000,0,-47.87E+5,0,+3.5E+5
    N=73 F=0,-48000,0,-47.97E+5,0,+1.7E+5
```

```
        N=81 F=0,-24000,0,-24.00E+5,0,+000000 !
SOLVE U=2 !OUTPUT P=0
LOADING
        N=57 F=0,-48000,0,-47.72E+5,0,+5.2E+5
        N=65 F=0,-48000,0,-47.87E+5,0,+3.5E+5
        N=73 F=0,-48000,0,-47.97E+5,0,+1.7E+5
        N=81 F=0,-24000,0,-24.00E+5,0,+000000 !
SOLVE U=2 !OUTPUT P=0
LOADING
        N=57 F=0,-48000,0,-47.72E+5,0,+5.2E+5
        N=65 F=0,-48000,0,-47.87E+5,0,+3.5E+5
        N=73 F=0,-48000,0,-47.97E+5,0,+1.7E+5
        N=81 F=0,-24000,0,-24.00E+5,0,+000000 !
SOLVE U=2 !OUTPUT P=0
LOADING
        N=57 F=0,-24000,0,-23.86E+5,0,+2.6E+5
        N=65 F=0,-24000,0,-23.94E+5,0,+1.8E+5
        N=73 F=0,-24000,0,-23.99E+5,0,+0.9E+5
        N=81 F=0,-12000,0,-12.00E+5,0,+000000 !
SOLVE U=2 !OUTPUT P=0
LOADING
        N=57 F=0,-24000,0,-23.86E+5,0,+2.6E+5
        N=65 F=0,-24000,0,-23.94E+5,0,+1.8E+5
        N=73 F=0,-24000,0,-23.99E+5,0,+0.9E+5
        N=81 F=0,-12000,0,-12.00E+5,0,+000000 !
SOLVE U=2 !OUTPUT P=0
LOADING
        N=57 F=0,-24000,0,-23.86E+5,0,+2.6E+5
        N=65 F=0,-24000,0,-23.94E+5,0,+1.8E+5
        N=73 F=0,-24000,0,-23.99E+5,0,+0.9E+5
        N=81 F=0,-12000,0,-12.00E+5,0,+000000 !
SOLVE U=2 !OUTPUT P=0
LOADING
        N=57 F=0,-12000,0,-11.93E+5,0,+1.3E+5
        N=65 F=0,-12000,0,-11.97E+5,0,+0.9E+5
        N=73 F=0,-12000,0,-11.99E+5,0,+0.4E+5
        N=81 F=0,-06000,0,-12.00E+5,0,+000000 !
SOLVE U=2 I=100 !OUTPUT P=0
LOADING
        N=57 F=0,-12000,0,-11.93E+5,0,+1.3E+5
        N=65 F=0,-12000,0,-11.97E+5,0,+0.9E+5
        N=73 F=0,-12000,0,-11.99E+5,0,+0.4E+5
        N=81 F=0,-06000,0,-12.00E+5,0,+000000 !
SOLVE U=2 I=100 !OUTPUT P=0
STOP
```

```
START ;
TITLE
 EXAMPLE 5.4  - KETCHUM SEGMENTAL ANALYSIS EXAMPLE
               THREE SPAN SEGMENTAL CANTILEVER CONSTRUCTION OF A STRAIGHT
               POST-TENSIONED BOX GIRDER BRIDGE.
               CAST IN PLACE SEGMENTAL CONSTRUCTION WITH TRAVELERS, 7 DAY
               CYCLE
CONCRETE PARAMETERS N=1 ! 1 M=ACI T=1.0 W=150
MESH INPUT
NODES N=44
     01 X=0    Y=-46.74 ! 02 X=180  Y=-40.75 ! 03 X=360  Y=-34.50
     04 X=540  Y=-34.50 ! 05 X=720  Y=-34.50 ! 06 X=788  Y=-34.61
     07 X=948  Y=-35.37 ! 08 X=1108 Y=-36.65 ! 09 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 ! 44 X=3420 Y=+00.00 !
LOCAL SYSTEM N=42
     01 K=01,02,44 ! 02 K=02,03,44 ! 03 K=03,04,44 ! 04 K=04,05,44 ! 05 K=05,06,44
     06 K=06,07,44 ! 07 K=07,08,44 ! 08 K=08,09,44 ! 09 K=09,10,44 ! 10 K=10,11,44
     11 K=11,12,44 ! 12 K=12,13,44 ! 13 K=13,14,44 ! 14 K=14,15,44 ! 15 K=15,16,44
     16 K=16,17,44 ! 17 K=17,18,44 ! 18 K=18,19,44 ! 19 K=19,20,44 ! 20 K=20,21,44
     21 K=21,22,44 ! 22 K=22,23,44 ! 23 K=23,24,44 ! 24 K=24,25,44 ! 25 K=25,26,44
     26 K=26,27,44 ! 27 K=27,28,44 ! 28 K=28,29,44 ! 29 K=29,30,44 ! 30 K=30,31,44
     31 K=31,32,44 ! 32 K=32,33,44 ! 33 K=33,34,44 ! 34 K=34,35,44 ! 35 K=35,36,44
     36 K=36,37,44 ! 37 K=37,38,44 ! 38 K=38,39,44 ! 39 K=39,40,44 ! 40 K=40,41,44
     41 X=0,-1,0 Y=+1,0,0 Z=0,0,1 ! 42 X=1,0,0 Y=0,1,0 Z=0,0,1 !
MATERIAL MODELS N=1,1,1 P=1,2,1
     1 M=MILD N=2 I=2 S=150E3 E=0.005172
       N=1 C=150E3,0.005172 ! N=2 C=150E4,0.05172
     1 M=CONC N=1 I=2 R=1.0,1.0
       N=1 C=-3.44,-0.004
     1 M=PRES N=1
       N=1 C=81E4,0.0289284 !
CONCRETE PROPERTIES N=2
     1 F=5000 W=155/1728 D=1 S=0.0008 C=2.5
     2 F=1.0E-04 W=155/1728 D=1 S=0.0008 C=2.5 !
MILD STEEL N=1 ! 1 D=1
FSECTION PROPERTIES N=145 M=15,1 P=1
     01 C=0,0 D=0.01,534750.0
       N=1,3095,0 ! N=2,2376,0 !
     02 C=0,0 D=0.01,534750.0
       N=1,3133,0 ! N=2,2414,0 !
     03 C=0,0 D=0.01,534750.0
       N=1,3236,0 ! N=2,2517,0 !
     04 C=0,0 D=0.01,534750.0
       N=1,3387,0 ! N=2,2668,0 !
     05 C=0,0 D=0.01,534750.0
       N=1,3748,0 ! N=2,3029,0 !
     06 C=0,0 D=0.01,534750.0
       N=1,4282,0 ! N=2,3563,0 !
     07 C=0,0 D=0.01,534750.0
```

```
  N=1,4851,0 ! N=2,4132,0 !
08 C=0,0 D=0.01,534750.0
  N=1,5461,0 ! N=2,4742,0 !
09 C=0,0 D=0.01,534750.0
  N=1,6111,0 ! N=2,5392,0 !
10 C=0,0 D=0.01,534750.0
  N=1,6807,0 ! N=2,6088,0 !
11 C=0,0 D=0.01,534750.0
  N=1,7545,0 ! N=2,6826,0 !
12 C=0,0 D=0.01,534750.0
  N=1,8328,0 ! N=2,7609,0 !
13 C=0,0 D=0.01,534750.0
  N=1,9153,0 ! N=2,8434,0 !
14 C=0,0 D=0.01,534750.0
  N=1,10021,0 ! N=2,9302,0 !
15 C=0,0 D=0.01,534750.0
  N=1,10931,0 ! N=2,10212,0 !
16 C=0,0 D=0.01,534750.0
  N=1,11881,0 ! N=2,11162,0 !
17 C=0,0 D=0.01,534750.0
  N=1,12861,0 ! N=2,12142,0 !
18 C=0,0 D=0.01,534750.0
  N=1,13601,0 ! N=2,12882,0 !
19 C=0,0 D=0.01,534750.0
  N=1,13831,0 ! N=2,13112,0 !
20 C=0,0 D=0.01,534750.0
  N=1,4050,0 ! N=2,3331,0 !
21 C=0,0 D=0.01,534750.0
  N=1,3300,0 ! N=2,2581,0 !
22 C=0,0 D=0.01,534750.0
  N=1,3091,0 ! N=2,2372,0 !
23 C=0,0 D=0.01,534750.0
  N=1,3091,0 ! N=2,2372,0 !
24 C=0,0 D=0.01,334080.0
  N=1,4320,0 ! N=2,3360,0 !
25 C=0,0 D=0.01,56486.7
  N=1,+1311,0 ! N=2,-0102,0 ! N=3,-1514,0 ! N=4,-2926,0 ! N=5,-4338,0
  N=6,-5751,0 !
26 C=0,0 D=0.01,57280.0
  N=1,+1339,0 ! N=2,-0094,0 ! N=3,-1526,0 ! N=4,-2958,0 ! N=5,-4390,0
  N=6,-5822,0 !
27 C=0,0 D=0.01,59413.3
  N=1,+1415,0 ! N=2,-0071,0 ! N=3,-1556,0 ! N=4,-3041,0 ! N=5,-4527,0
  N=6,-6019,0 !
28 C=0,0 D=0.01,62513.3
  N=1,+1527,0 ! N=2,-0039,0 ! N=3,-1599,0 ! N=4,-3162,0 ! N=5,-4724,0
  N=5,-6287,0 !
29 C=0,0 D=0.01,65573.3
  N=1,+1850,0 ! N=2,+0211,0 ! N=3,-1429,0 ! N=4,-3068,0 ! N=5,-4708,0
  N=6,-6347,0 !
30 C=0,0 D=0.01,68680.0
  N=1,+2345,0 ! N=2,+0628,0 ! N=3,-1089,0 ! N=4,-2806,0 ! N=5,-4523,0
  N=5,-6240,0 !
31 C=0,0 D=0.01,62154.3
  N=1,+2996,0 ! N=2,+1442,0 ! N=3,-0112,1 ! N=4,-1666,0 ! N=5,-3220,0
  N=6,-4774,0 ! N=7,-6328,0 !
32 C=0,0 D=0.01,66028.6
  N=1,+3557,0 ! N=2,+1907,0 ! N=3,+0256,0 ! N=4,-1395,0 ! N=5,-3046,0
  N=6,-4697,0 ! N=7,-6347,0 !
33 C=0,0 D=0.01,70462.9
```

N=1,+4152,0 ! N=2,+2390,0 ! N=3,+0629,0 ! N=4,-1133,0 ! N=5,-2895,0
N=6,-4656,0 ! N=7,-6418,0 !
34 C=0,0 D=0.01,75434.3
N=1,+4786,0 ! N=2,+2900,0 ! N=3,+1014,0 ! N=4,-0872,0 ! N=5,-2758,0
N=6,-4644,0 ! N=7,-6530,0 !
35 C=0,0 D=0.01,70815.0
N=1,+5581,0 ! N=2,+3811,0 ! N=3,+2041,0 ! N=4,+0270,0 ! N=5,-1500,0
N=6,-3271,0 ! N=7,-5041,0 ! N=8,-6811,0 !
36 C=0,0 D=0.01,76065.0
N=1,+6299,0 ! N=2,+4397,0 ! N=3,+2496,0 ! N=4,+0594,0 ! N=5,-1308,0
N=6,-3210,0 ! N=7,-5111,0 ! N=8,-7013,0 !
37 C=0,0 D=0.01,81740.0
N=1,+7053,0 ! N=2,+5009,0 ! N=3,+2966,0 ! N=4,+0922,0 ! N=5,-1121,0
N=6,-3165,0 ! N=7,-5208,0 ! N=8,-7252,0 !
38 C=0,0 D=0.01,87835.0
N=1,+7845,0 ! N=2,+5649,0 ! N=3,+3453,0 ! N=4,+1257,0 ! N=5,-0939,0
N=6,-3135,0 ! N=7,-5331,0 ! N=8,-7527,0 !
39 C=0,0 D=0.01,94340.0
N=1,+8673,0 ! N=2,+6315,0 ! N=3,+3956,0 ! N=4,+1598,0 ! N=5,-0761,0
N=6,-3119,0 ! N=7,-5478,0 ! N=8,-7836,0 !
40 C=0,0 D=0.01,101240.0
N=1,+9537,0 ! N=2,+7006,0 ! N=3,+4475,0 ! N=4,+1944,0 ! N=5,-0587,0
N=6,-3118,0 ! N=7,-5649,0 ! N=8,-8180,0 !
41 C=0,0 D=0.01,96471.1
N=1,+10577,0 ! N=2,+08165,0 ! N=3,+05753,0 ! N=4,+03341,0 ! N=5,+00930,0
N=6,-01482,0 ! N=7,-03894,0 ! N=8,-06306,0 ! N=9,-08718,0 !
42 C=0,0 D=0.01,101382.2
N=1,+11255,0 ! N=2,+08721,0 ! N=3,+06186,0 ! N=4,+03652,0
N=5,+01117,0 ! N=6,-01418,0 ! N=7,-03952,0 ! N=8,-06487,0
N=9,-09021,0 !
43 C=0,0 D=0.01,102942.2
N=1,+11466,0 ! N=2,+8892,0 ! N=3,+6319,0 ! N=4,+3745,0 ! N=5,+1172,0
N=6,-01402,0 ! N=7,-3976,0 ! N=8,-6549,0 ! N=9,-9123,0 !
44 C=0,0 D=0.01,49333.3
N=1,+2355,0 ! N=2,+1122,0 ! N=3,-0112,0 ! N=4,-1345,0 ! N=5,-2579,0
N=6,-3812,0 !
45 C=0,0 D=0.01,55166.7
N=1,+1532,0 ! N=2,+0153,0 ! N=3,-1226,0 ! N=4,-2606,0 ! N=5,-3985,0
N=6,-5364,0 !
46 C=0,0 D=0.01,56420.0 N=1,+1307,0 ! N=2,-0103,0 ! N=3,-1514,0 ! N=4,-2924,0 ! N=5,-4335,0
N=6,-5745,0 !
47 C=0,0 D=0.01,56420.0
N=1,+1307,0 ! N=2,-0103,0 ! N=3,-1514,0 ! N=4,-2924,0 ! N=5,-4335,0
N=6,-5745,0 !
48 C=0,0 D=0.01,334080.0
N=1,+2400,0 ! N=2,+1440,0 ! N=3,+0480,0 ! N=4,-0480,0
N=5,-1440,0 ! N=6,-2400,0 !
49 C=0,0 D=0.01,157257.0
N=1,-6682,0 ! N=2,-7132,0 !
50 C=0,0 D=0.01,157081.5
N=1,-6763,0 ! N=2,-7213,0 !
51 C=0,0 D=0.01,156600.0
N=1,-6980,0 ! N=2,-7430,0 !
52 C=0,0 D=0.01,155902.5
N=1,-7294,0 ! N=2,-7744,0 !
53 C=0,0 D=0.01,178209.9
N=1,-7425,0 ! N=2,-7942,0 !
54 C=0,0 D=0.01,219182.2
N=1,-7418,0 ! N=2,-8058,0 !

```
55 C=0,0 D=0.01,172814.8
  N=1,-7358,0 ! N=2,-7866,0 ! N=3,-8374,0 !
56 C=0,0 D=0.01,199047.3
  N=1,-7467,0 ! N=2,-8057,0 ! N=3,-8646,0 !
57 C=0,0 D=0.01,224505.7
  N=1,-7634,0 ! N=2,-8305,0 ! N=3,-8975,0 !
58 C=0,0 D=0.01,249353.4
  N=1,-7849,0 ! N=2,-8601,0 ! N=3,-9353,0 !
59 C=0,0 D=0.01,204975.7
  N=1,-8009,0 ! N=2,-8634,0 ! N=3,-9260,0 ! N=4,-9885,0 !
60 C=0,0 D=0.01,222371.1
  N=1,-8307,0 ! N=2,-8993,0 ! N=3,-9680,0 ! N=4,-10366,0 !
61 C=0,0 D=0.01,238998.2
  N=1,-8647,0 ! N=2,-9395,0 ! N=3,-10142,0 ! N=4,-10890,0 !
62 C=0,0 D=0.01,254887.7
  N=1,-9029,0 ! N=2,-9837,0 ! N=3,-10646,0 ! N=4,-11454,0 !
63 C=0,0 D=0.01,270074.8
  N=1,-9451,0 ! N=2,-10320,0 ! N=3,-11190,0 ! N=4,-12060,0 !
64 C=0,0 D=0.01,284353.4
  N=1,-9911,0 ! N=2,-10842,0 ! N=3,-11773,0 ! N=4,-12703,0 !
65 C=0,0 D=0.01,238270.5
  N=1,-10320,0 ! N=2,-11114,0 ! N=3,-11908,0 ! N=4,-12701,0 ! N=5,-13495,0 !
66 C=0,0 D=0.01,245599.5
  N=1,-10703,0 ! N=2,-11532,0 ! N=3,-12361,0 ! N=4,-13190,0 ! N=5,-14019,0 !
67 C=0,0 D=0.01,247800.0
  N=1,-10830,0 ! N=2,-11670,0 ! N=3,-12510,0 ! N=4,-13350,0 ! N=5,-14190,0 !
68 C=0,0 D=0.01,229847.7
  N=1,-4756,0 ! N=2,-5410,0 ! N=3,-6064,0 !
69 C=0,0 D=0.01,190296.6
  N=1,-6326,0 ! N=2,-6870,0 !
70 C=0,0 D=0.01,157275.0
  N=1,-6676,0 ! N=2,-7126,0 !
71 C=0,0 D=0.01,157275.0
  N=1,-6676,0 ! N=2,-7126,0 !
72 C=0,0 D=0.01,334080.0
  N=1,-3360,0 ! N=2,-4320,0 !
73 C=0,0 D=0.01,214
  N=1,2735,0 !
74 C=0,0 D=0.01,214
  N=1,2773,0 !
75 C=0,0 D=0.01,214
  N=1,2876,0 !
76 C=0,0 D=0.01,214
  N=1,3027,0 !
77 C=0,0 D=0.01,214
  N=1,3388,0 !
78 C=0,0 D=0.01,214
  N=1,3922,0 !
79 C=0,0 D=0.01,214
  N=1,4491,0 !
80 C=0,0 D=0.01,214
  N=1,5101,0 !
81 C=0,0 D=0.01,214
  N=1,5751,0 !
82 C=0,0 D=0.01,214
  N=1,6447,0 !
83 C=0,0 D=0.01,214
  N=1,7185,0 !
84 C=0,0 D=0.01,214
  N=1,7968,0 !
```

```
85 C=0,0 D=0.01,214
  N=1,8793,0 !
86 C=0,0 D=0.01,214
  N=1,9661,0 !
87 C=0,0 D=0.01,214
  N=1,10571,0 !
88 C=0,0 D=0.01,214
  N=1,11521,0 !
89 C=0,0 D=0.01,214
  N=1,12501,0 !
90 C=0,0 D=0.01,214
  N=1,13241,0 !
91 C=0,0 D=0.01,1070
  N=1,13471,0 !
92 C=0,0 D=0.01,214
  N=1,3690,0 !
93 C=0,0 D=0.01,214
  N=1,2940,0 !
94 C=0,0 D=0.01,214
  N=1,2731,0 !
95 C=0,0 D=0.01,214
  N=1,2731,0 !
96 C=0,0 D=0.01,668
  N=1,3840,0 !
97 C=0,0 D=0.01,23
  N=1,604,0 ! N=2,-2220,0 ! N=3,-5044,0 !
98 C=0,0 D=0.01,23
  N=1,622,0 ! N=2,-2242,0 ! N=3,-5106,0 !
99 C=0,0 D=0.01,24
  N=1,672,0 ! N=2,-2298,0 ! N=3,-5269,0 !
100 C=0,0 D=0.01,25
  N=1,746,0 ! N=2,-2380,0 ! N=3,-5506,0 !
101 C=0,0 D=0.01,26
  N=1,1030,0 ! N=2,-2249,0 ! N=3,-5527,0 !
102 C=0,0 D=0.01,27
  N=1,1486,0 ! N=2,-1948,0 ! N=3,-5382,0 !
103 C=0,0 D=0.01,29
  N=1,1442,0 ! N=2,-1666,0 ! N=3,-4774,0 !
104 C=0,0 D=0.01,31
  N=1,1906,0 ! N=2,-1395,0 ! N=3,-4696,0 !
105 C=0,0 D=0.01,33
  N=1,2390,0 ! N=2,-1133,0 ! N=3,-4656,0 !
106 C=0,0 D=0.01,35
  N=1,2900,0 ! N=2,-872,0 ! N=3,-4644,0 !
107 C=0,0 D=0.01,38
  N=1,3811,0 ! N=2,270,0 ! N=3,-3271,0 !
108 C=0,0 D=0.01,41
  N=1,4397,0 ! N=2,594,0 ! N=3,-3209,0 !
109 C=0,0 D=0.01,44
  N=1,5009,0 ! N=2,922,0 ! N=3,-3165,0 !
110 C=0,0 D=0.01,47
  N=1,5649,0 ! N=2,1257,0 ! N=3,-3135,0 !
111 C=0,0 D=0.01,50
  N=1,6315,0 ! N=2,1598,0 ! N=3,-3119,0 !
112 C=0,0 D=0.01,54
  N=1,7006,0 ! N=2,1950,0 ! N=3,12240,0 !
113 C=0,0 D=0.01,43
  N=1,8165,0 ! N=2,3341,0 ! N=3,-1482,0 ! N=4,-6306,0 !
114 C=0,0 D=0.01,46
  N=1,8721,0 ! N=2,3652,0 ! N=3,-1418,0 ! N=4,-6487,0 !
```

115 C=0,0 D=0.01,232
N=1,8892,0 ! N=2,3745,0 ! N=3,-1402,0 ! N=4,-6549,0 !
116 C=0,0 D=0.01,20
N=1,1738,0 ! N=2,-729,0 ! N=3,-3195,0 !
117 C=0,0 D=0.01,22
N=1,842,0 ! N=2,-1916,0 ! N=3,-4674,0 !
118 C=0,0 D=0.01,23
N=1,602,0 ! N=2,-2219,0 ! N=3,-5040,0 !
119 C=0,0 D=0.01,23
N=1,602,0 ! N=2,-2219,0 ! N=3,-5040,0 !
120 C=0,0 D=0.01,668
N=1,1920,0 ! N=2,0,0 ! N=3,-1920,0 !
121 C=0,0 D=0.01,63
N=1,-6907,0 !
102 C=0,0 D=0.01,63
N=1,-6988,0 !
123 C=0,0 D=0.01,63
N=1,-7205,0 !
124 C=0,0 D=0.01,62
N=1,-7519,0 !
125 C=0,0 D=0.01,71
N=1,-7684,0 !
126 C=0,0 D=0.01,88
N=1,-7738,0 !
127 C=0,0 D=0.01,52
N=1,-7612,0 ! N=2,-8120,0 !
128 C=0,0 D=0.01,60
N=1,-7762,0 ! N=2,83.51,0 !
129 C=0,0 D=0.01,67
N=1,-7969,0 ! N=2,8640,0 !
130 C=0,0 D=0.01,75
N=1,-8225,0 ! N=2,-8977,0 !
131 C=0,0 D=0.01,82
N=1,-8009,0 ! N=2,-9260,0 !
132 C=0,0 D=0.01,89
N=1,-8307,0 ! N=2,-9680,0 !
133 C=0,0 D=0.01,96
N=1,-8647,0 ! N=2,-101.42,0 !
134 C=0,0 D=0.01,102
N=1,-9029,0 ! N=2,-10646,0 !
135 C=0,0 D=0.01,108
N=1,-9450,0 ! N=2,-11190,0 !
136 C=0,0 D=0.01,114
N=1,27598,0 ! N=2,34391,0 !
137 C=0,0 D=0.01,79
N=1,-10320,0 ! N=2,-11908,0 ! N=3,-13495,0 !
138 C=0,0 D=0.01,82
N=1,-10703,0 ! N=2,-12361,0 ! N=3,-14019,0 !
139 C=0,0 D=0.01,413
N=1,-10830,0 ! N=2,-12510,0 ! N=3,-14190,0 !
140 C=0,0 D=0.01,138
N=1,-5410,0 !
141 C=0,0 D=0.01,76
N=1,-6598,0 !
142 C=0,0 D=0.01,63
N=1,-6901,0 !
143 C=0,0 D=0.01,63
N=1,-6901,0 !
144 C=0,0 D=0.01,668
N=1,-3840,0 !

```
145 C=0,0 D=1000,0.01
  N=1,0,0 !
TSECTION PROPERTIES N=1 R=3
  1 A=696 I=0,1.081E6,0,0 !
TRAVELERS N=3
  1 X=1 E=29E6 W=15E4 N=3
  2 X=1 E=29E6 W=15E4 N=3
  3 X=1 E=29E6 W=15E4 N=2 !
FRAME ELEMENTS N=166
  001,01,02 C=1,20 S=1,092 D=100 G=001,010,3,1,1,+0 J=0,+1,0,+1,1 L=1
  002,01,02 C=1,44 S=1,116 D=100 G=002,011,3,1,1,+0 J=0,+1,0,+1,1 L=1
  003,01,02 C=1,68 S=1,140 D=100 G=003,012,3,1,1,+0 J=0,+1,0,+1,1 L=1
  013,05,06 C=1,01 S=1,073 D=168 G=013,064,3,1,1,-7 J=0,+1,0,+1,1 L=5
  014,05,06 C=1,25 S=1,097 D=168 G=014,065,3,1,1,-7 J=0,+1,0,+1,1 L=5
  015,05,06 C=1,49 S=1,121 D=168 G=015,066,3,1,1,-7 J=0,+1,0,+1,1 L=5
  067,23,24 C=1,18 S=1,090 D=049 G=067,118,3,1,1,+7 J=0,-1,0,-1,1 L=23
  068,23,24 C=1,42 S=1,114 D=049 G=068,119,3,1,1,+7 J=0,-1,0,-1,1 L=23
  069,23,24 C=1,66 S=1,138 D=049 G=069,120,3,1,1,+7 J=0,-1,0,-1,1 L=23
  121,23,42 C=1,019 S=1,091 D=000 L=41
  122,23,42 C=1,043 S=1,115 D=000 L=41
  123,23,42 C=1,067 S=1,139 D=000 L=41
  124,42,43 C=1,024 S=1,096 D=000 L=41
  125,42,43 C=1,048 S=1,120 D=000 L=41
  126,42,43 C=1,072 S=1,144 D=000 L=41
  127,01,02 C=2,145 L=1 G=127,166,1,1,1,0 J=0,0,0,0,1 !
PRESTRESSING STEEL N=1 ! 1 C=.25 W=.0004/12 S=27E4 D=1 R=10 !
TENDONS N=30
; CANTILEVER TENDONS - EACH REPRESENTS 4, 21-STRAND, 1/2" DIAM TENDONS
  1 S=2 M=1 A=12.852 F=1
    1 N=3  G=21,23,1 L=42 ! R=0,.6638,0,0,0,0 S=-20,-7,-7,0,0,0
    2 N=3  G=23,25,1 L=42 ! R=0,.3362,0,0,0,0 S=-7,-7,-20,0,0,0
  2 S=2 M=1 A=12.852 F=1
    1 N=4  G=20,23,1 L=42 ! R=0,.3929,0,0,0,0 S=-20,-7,-7,0,0,0
    2 N=4  G=23,26,1 L=42 ! R=0,.6071,0,0,0,0 S=-7,-7,-20,0,0,0
  3 S=2 M=1 A=12.852 F=1
    1 N=5  G=19,23,1 L=42 ! R=0,.2790,0,0,0,0 S=-20,-7,-7,0,0,0
    2 N=5  G=23,27,1 L=42 ! R=0,.7210,0,0,0,0 S=-7,-7,-20,0,0,0
  4 S=2 M=1 A=12.852 F=1
    1 N=6  G=18,23,1 L=42 ! R=0,.2163,0,0,0,0 S=-20,-7,-7,0,0,0
    2 N=6  G=23,28,1 L=42 ! R=0,.7837,0,0,0,0 S=-7,-7,-20,0,0,0
  5 S=2 M=1 A=12.852 F=1
    1 N=7  G=17,23,1 L=42 ! R=0,.1766,0,0,0,0 S=-20,-7,-7,0,0,0
    2 N=7  G=23,29,1 L=42 ! R=0,.8234,0,0,0,0 S=-7,-7,-20,0,0,0
  6 S=2 M=1 A=12.852 F=1
    1 N=8  G=16,23,1 L=42 ! R=0,.1492,0,0,0,0 S=-20,-7,-7,0,0,0
    2 N=8  G=23,30,1 L=42 ! R=0,.8508,0,0,0,0 S=-7,-7,-20,0,0,0
  7 S=2 M=1 A=12.852 F=1
    1 N=9  G=15,23,1 L=42 ! R=0,.1292,0,0,0,0 S=-20,-7,-7,0,0,0
    2 N=9  G=23,31,1 L=42 ! R=0,.8708,0,0,0,0 S=-7,-7,-20,0,0,0
  8 S=2 M=1 A=12.852 F=1
    1 N=10 G=14,23,1 L=42 ! R=0,.1139,0,0,0,0 S=-20,-7,-7,0,0,0
    2 N=10 G=23,32,1 L=42 ! R=0,.8861,0,0,0,0 S=-7,-7,-20,0,0,0
  9 S=2 M=1 A=12.852 F=1
    1 N=11 G=13,23,1 L=42 ! R=0,.1019,0,0,0,0 S=-20,-7,-7,0,0,0
    2 N=11 G=23,33,1 L=42 ! R=0,.8981,0,0,0,0 S=-7,-7,-20,0,0,0
  10 S=2 M=1 A=12.852 F=1
    1 N=12 G=12,23,1 L=42 ! R=0,.0921,0,0,0,0 S=-20,-7,-7,0,0,0
    2 N=12 G=23,34,1 L=42 ! R=0,.9079,0,0,0,0 S=-7,-7,-20,0,0,0
  11 S=2 M=1 A=12.852 F=1
    1 N=13 G=11,23,1 L=42 ! R=0,.0841,0,0,0,0 S=-20,-7,-7,0,0,0
```

```
        2 N=13 G=23,35,1 L=42 ! R=0,.9159,0,0,0,0 S=-7,-7,-20,0,0,0
      12 S=2 M=1 A=12.852 F=1
        1 N=14 G=10,23,1 L=42 ! R=0,.0773,0,0,0,0 S=-20,-7,-7,0,0,0
        2 N=14 G=23,36,1 L=42 ! R=0,.9277,0,0,0,0 S=-7,-7,-20,0,0,0
      13 S=2 M=1 A=12.852 F=1
        1 N=15 G=09,23,1 L=42 ! R=0,.0716,0,0,0,0 S=-20,-7,-7,0,0,0
        2 N=15 G=23,37,1 L=42 ! R=0,.9284,0,0,0,0 S=-7,-7,-20,0,0,0
      14 S=2 M=1 A=12.852 F=1
        1 N=16 G=08,23,1 L=42 ! R=0,.0666,0,0,0,0 S=-20,-7,-7,0,0,0
        2 N=16 G=23,38,1 L=42 ! R=0,.9334,0,0,0,0 S=-7,-7,-20,0,0,0
      15 S=2 M=1 A=12.852 F=1
        1 N=17 G=07,23,1 L=42 ! R=0,.0623,0,0,0,0 S=-20,-7,-7,0,0,0
        2 N=17 G=23,39,1 L=42 ! R=0,.9377,0,0,0,0 S=-7,-7,-20,0,0,0
      16 S=2 M=1 A=12.852 F=1
        1 N=18 G=06,23,1 L=42 ! R=0,.0585,0,0,0,0 S=-20,-7,-7,0,0,0
        2 N=18 G=23,40,1 L=42 ! R=0,.9415,0,0,0,0 S=-7,-7,-20,0,0,0
; CONTINUITY TENDONS - EACH REPRESENTS 8, 21-STRAND, 1/2" DIAM TENDONS
      17 S=2 M=1 A=25.704 F=1
        1 N=18 G=01,18,1 L=42 ! R=0,.4,.2,0,0,0  S=-46.74,-100,-12,0,0,0
        2 N=6  G=18,23,1 L=42 ! R=0,.5,0,0,0,0   S=-12,-12,-12,0,0,0
      18 S=2 M=1 A=25.704 F=1
        1 N=6  G=23,28,1 L=42 ! R=0,.5,0,0,0,0   S=-12,-12,-12,0,0,0
        2 N=14 G=28,41,1 L=42 ! R=.3,1,0,0,0,0   S=-12,-100,-100,0,0,0
; LOCAL TENDONS - EACH REPRESENTS 2 OR 4, 21-STRAND, 1/2" DIAM TENDONS
      19 S=1 M=1 A=6.426  F=1
        1 N=9  G=01,09,1 L=42 ! R=0,.874,0,0,0,0  S=-46.74,-98.15,-103.34,0,0,0
      20 S=1 M=1 A=6.426  F=1
        1 N=10 G=01,10,1 L=42 ! R=0,.776,0,0,0,0  S=-46.74,-98.15,-107.43,0,0,0
      21 S=1 M=1 A=6.426  F=1
        1 N=11 G=01,11,1 L=42 ! R=0,.698,0,0,0,0  S=-46.74,-98.15,-112.64,0,0,0
      22 S=1 M=1 A=6.426  F=1
        1 N=12 G=01,12,1 L=42 ! R=0,.634,0,0,0,0  S=-46.74,-98.15,-118.91,0,0,0
      23 S=1 M=1 A=6.426  F=1
        1 N=12 G=30,41,1 L=42 ! R=0,1,0,0,0,0     S=-153.64,-100,-100,0,0,0
      24 S=1 M=1 A=6.426  F=1
        1 N=11 G=31,41,1 L=42 ! R=0,1,0,0,0,0     S=-143.58,-100,-100,0,0,0
      25 S=1 M=1 A=12.852 F=1
        1 N=10 G=32,41,1 L=42 ! R=0,1,0,0,0,0     S=-134.42,-100,-100,0,0,0
      26 S=1 M=1 A=12.852 F=1
        1 N= 9 G=33,41,1 L=42 ! R=0,1,0,0,0,0     S=-126.18,-100,-100,0,0,0
      27 S=1 M=1 A=12.852 F=1
        1 N= 8 G=34,41,1 L=42 ! R=0,1,0,0,0,0     S=-118.91,-100,-100,0,0,0
      28 S=1 M=1 A=12.852 F=1
        1 N= 7 G=35,41,1 L=42 ! R=0,1,0,0,0,0     S=-112.64,-100,-100,0,0,0
      29 S=1 M=1 A=6.426  F=1
        1 N= 6 G=36,41,1 L=42 ! R=0,1,0,0,0,0     S=-107.43,-100,-100,0,0,0
      30 S=1 M=1 A=6.426  F=1
        1 N= 5 G=37,41,1 L=42 ! R=0,1,0,0,0,0     S=-103.34,-100,-100,0,0,0
MESH COMPLETE !SET D=56 G=0,-1,0,0 N=1.0
; BUILD PIER AND STARTING SEGMENT
CHANGE STRUCTURE
RESTRAINTS !1,43 R=0,0,1,1,1,0 !1,5 R=0,1,1,1,1,0 !41 R=1,0,1,1,1,1 !
RESTRAINTS !43 R=1,1,1,1,1,1 !
BUILD N=121,126 !BUILD N=061,072 D=35 !BUILD N=147,150 D=35 !
CHANGE COMPLETE !SOLVE !SOLVE A=0 D=63
; 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 !OUTPUT P=0
CHANGE STRUCTURE ! BUILD N=058,060 !BUILD N=073,075 ! BUILD N=146,151,5 !
CHANGE COMPLETE !SOLVE !SOLVE D=70
```

```
; 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 !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=55,57 !BUILD N=76,78 !BUILD N=145,152,7 !
CHANGE COMPLETE !SOLVE !SOLVE D=77
; 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 !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=52,54 !BUILD N=79,81 !BUILD N=144,153,9 !
CHANGE COMPLETE !SOLVE !SOLVE D=84
; 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 !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=49,51 !BUILD N=82,84 !BUILD N=143,154,11 !
CHANGE COMPLETE !SOLVE !SOLVE D=91
; 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 !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=46,48 !BUILD N=85,87 !BUILD N=142,155,13 !
CHANGE COMPLETE !SOLVE !SOLVE D=98
; 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 !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=43,45 !BUILD N=88,90 !BUILD N=141,156,15 !
CHANGE COMPLETE !SOLVE !SOLVE D=105
; 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 !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=40,42 !BUILD N=91,93 !BUILD N=140,157,17 !
CHANGE COMPLETE !SOLVE !SOLVE D=112
; 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 !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=37,39 !BUILD N=94,96 !BUILD N=139,158,19 !
CHANGE COMPLETE !SOLVE !SOLVE D=119
; 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 !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=34,36 !BUILD N=97,99 !BUILD N=138,159,21 !
CHANGE COMPLETE !SOLVE !SOLVE D=126
; 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 !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=31,33 !BUILD N=100,102 !BUILD N=137,160,23 !
CHANGE COMPLETE !SOLVE !SOLVE D=133
; 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 !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=28,30 !BUILD N=103,105 !BUILD N=136,161,25 !
CHANGE COMPLETE !SOLVE !SOLVE D=140
; STRESS TENDON 12, MOVE TRAVELERS, BUILD SEGMENTS 9 AND 36
CHANGE STRUCTURE !STRESS N=12 S=198E3,198E3 D=.25,.25
MOVE N=1 D=9,10,11 !MOVE N=2 D=35,36,37 !CHANGE COMPLETE  !SOLVE !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=25,27 !BUILD N=106,108 !BUILD N=135,162,27 !
CHANGE COMPLETE !SOLVE !SOLVE D=147
; 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 !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=22,24 !BUILD N=109,111 !BUILD N=134,163,29 !
CHANGE COMPLETE !SOLVE !SOLVE D=154

; 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 !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=19,21 !BUILD N=112,114 !BUILD N=133,164,31 !
CHANGE COMPLETE !SOLVE !SOLVE D=161

; 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 !OUTPUT P=0
CHANGE STRUCTURE !BUILD N=16,18 !BUILD N=115,117 !BUILD N=132,165,33 !
CHANGE COMPLETE !SOLVE !SOLVE D=168

; STRESS TENDON 16
CHANGE STRUCTURE !STRESS N=16 S=198E3,198E3 D=.25,.25
CHANGE COMPLETE  !SOLVE A=0.71 !OUTPUT P=0

; BUILD SEGMENTS 1 THRU 4, INSTALL CLOSURE FORMWORK, BUILD CLOSURE SEGMENT 5
CHANGE STRUCTURE !BUILD N=1,12 !BUILD N=127,130 !
MOVE N=3 D=5,6 !BUILD N=13,15 !BUILD N=131 !
CHANGE COMPLETE  !SOLVE !SOLVE D=175 S=3

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

; MOVE CLOSURE FORMWORK TO CENTER SPAN, BUILD CLOSURE SEGMENT 40
CHANGE STRUCTURE !MOVE N=3 D=40,41 !BUILD N=118,120 D=182 !BUILD N=166 !
CHANGE COMPLETE  !SOLVE !SOLVE D=189 S=3

; 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 !SOLVE D=196 S=3

; 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,-2500/12,0,0,0,0 !!SOLVE  !OUTPUT

; STEP THROUGH TIME UP TO 10000 DAYS (27.4 YEARS)
SOLVE D=00300 S=05 !OUTPUT
SOLVE D=10000 S=10 !OUTPUT P=2

STOP
```

```
START ;
TITLE
EXAMPLE 5.5   - TWO SPAN SIMPLY SUPPORTED PRECAST GIRDERS MADE CONTINUOUS AND
                COMPOSITE
                STRAIGHT PRESTRESSED GIRGER BRIDGE LOADED TO FAILURE
                UNITS = LB-IN
CONCRETE PARAMETERS N=1 ! 1 M=ACI W=143.96 T=0.6589
MESH INPUT
NODES N=10
      1 X=000 S=12 ! 2 X=010 ! 3 X=020 ! 4 X=030 ! 5 X=040
      6 X=043      ! 7 X=053 ! 8 X=063 ! 9 X=073 !10 X=083 !
CONCRETE PROPERTIES N=2
      1 F=8000 C=2.5 S=0.0006 W=155/1728
      2 F=8000 C=2.5 S=0.0006 W=0        !
LOCAL SYSTEM N=1 ! 1 !
MATERIAL MODELS N=1,1,1 P=10,2,5
      1 M=CONC I=1 E=0.002,0.004 N=8
      1 M=PRES ! N=1 C=196600,0.00715 ! N=2 C=220000,0.009 ! N=3 C=240000,0.0115
        N=4 C=245000,0.0135 ! N=5 C=270000,0.058
      1 M=MILD I=2 N=2 S=60000 E=2.069E-03
        N=1 C=60000,2.069E-03 ! N=2 C=60000.01,0.03 !
FSECTION PROPERTIES N=6 M=8,1 P=1
      1 C=0,0 D=1.00,60.0 ; Concrete Top Rectangular Flange
          N=1,+01,0 G=1,6 S=1,00 C=1,1 B=1,00 I=1 F=2 !
      2 C=0,0 D=1.00,15.0 ; Concrete Web
          N=1,-07,0 G=1,8 S=1,00 C=1,1 B=1,00 I=1 F=2 !
      3 C=0,0 D=1.00,40.0 ; Concrete Bottom Rectangular Flange
          N=1,-11,0 G=1,4 S=1,00 C=1,1 B=1,00 I=1 F=2 !
      4 C=0,0 D=1.00,2.80 ; Mild Stl Top Rectangular Flange
          N=1,+01,0 G=1,6 S=1,00 C=1,1 B=1,00 I=1 F=2 !
      5 C=0,0 D=1.00,0.15 ; Mild Stl Web
          N=1,-07,0 G=1,8 S=1,00 C=1,1 B=1,00 I=1 F=2 !
      6 C=0,0 D=1.00,1.85 ; Mild Stl Bop Rectangular Flange
          N=1,-11,0 G=1,4 S=1,00 C=1,1 B=1,00 I=1 F=2 !
MILD STEEL PROPERTIES N=1 ! 1 D=1 !
FRAME ELEMENTS N=27
      01,01,02 D=28 C=2,3 S=1,6 L=1 G=01,04,1,1,1,0 ; Top Rectangular Flange Left
      05,05,06 D=28 C=2,3 S=1,6 L=1                  ; Top Rectangular Flange Mid
      06,06,07 D=28 C=2,3 S=1,6 L=1 G=06,09,1,1,1,0 ; Top Rectangular Flange Right
      10,01,02 D=01 C=1,2 S=1,5 L=1 G=10,13,1,1,1,0 ; Web Rectangular Flange Left
      14,05,06 D=28 C=2,2 S=1,5 L=1                  ; Web Rectangular Flange Mid
      15,06,07 D=01 C=1,2 S=1,5 L=1 G=15,18,1,1,1,0 ; Web Rectangular Flange Right
      19,01,02 D=01 C=1,1 S=1,4 L=1 G=19,22,1,1,1,0 ; Bot Rectangular Flange Left
      23,05,06 D=28 C=2,1 S=1,4 L=1                  ; Bot Rectangular Flange Mid
      24,06,07 D=01 C=1,1 S=1,4 L=1 G=24,27,1,1,1,0 ; Bot Rectangular Flange Right !
PRESTRESSING STEEL N=1 ! 1 C=0.25 W=0.0002/12 S=210000 R=10 D=1 !
TENDON GEOMETRY N=2 P=2
      1 S=1 M=1 A=3.5 F=0 ;; STRESSING @ 214ksi
        1 N=05 G=01,05,1 O=0,0,0 L=1 ! R=0.25,0.50,0.25,0,0,0 S=-6.5,-9.0,-6.5 T=0,0,0
      2 S=1 M=1 A=3.5 F=0 ;; STRESSING @ 214ksi
        1 N=05 G=06,10,1 O=0,0,0 L=1 ! R=0.25,0.50,0.25,0,0,0 S=-6.5,-9.0,-6.5 T=0,0,0
MESH COMPLETE !SET D=07 G=0,-1.0,0,0 C=0.05,0.05,1,0.01,0.01,10.0,0.5 N=1.0
;; BUILD SIMPLY SUPPORTED I-BEAMS & ANALYSE DEAD LOAD WITH PRESTRESSING @ 07
DAY
CHANGE STRUCTURE
BUILD N=10,13,01 !BUILD N=15,18,01 !BUILD N=19,22,01 !BUILD N=24,27,01 !
RESTRAINTS ! 01,10,01 R=0,0,1,1,1,0 ! 01,10,09 R=0,1,1,1,1,0 !
RESTRAINTS ! 05,06,01 R=1,1,1,1,1,0 !
STRESS N=01,02,01 F=0.75E+06,0.75E+06 D=0,0
```

```
CHANGE COMPLETE
SOLVE D=07 U=2 !OUTPUT P=1 !SOLVE D=28 U=2 !OUTPUT P=1 !
;; SLAB LOADING CARRIED BY GIRDER @28->56 DAYS
LOADING
N=02,05,01 F=0,-6250,0,0,0,0 ;; Left  Girder
N=01       F=0,-3125,0,0,0,0 ;;
N=06,09,01 F=0,-6250,0,0,0,0 ;; Right Girder
N=10       F=0,-3125,0,0,0,0 !
SOLVE D=28 U=2 !OUTPUT P=1 !SOLVE D=56 U=2 !OUTPUT P=1
;;INSTALL DECK SLAB AND CONTINUITY SEGMENT @ 28 DAYS,
CHANGE STRUCTURE
BUILD N=01,04,01 !BUILD N=06,09,01 !BUILD N=05,14,23 !
CHANGE COMPLETE
SOLVE D=56 U=2 P=4 !OUTPUT P=1
;;ADDITIONAL LOADS
LOADING ;; APPLY AT CENTER OF SPANS
N=03,08,01 F=0,-2500,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
;;ADDITIONAL LOADS
LOADING ;; APPLY AT CENTER OF SPANS
N=03,08,01 F=0,-2500,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
;;ADDITIONAL LOADS
LOADING ;; APPLY AT CENTER OF SPANS
N=03,08,01 F=0,-2500,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
;;ADDITIONAL LOADS
LOADING ;; APPLY AT CENTER OF SPANS
N=03,08,01 F=0,-2500,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
;;ADDITIONAL LOADS
LOADING ;; APPLY AT CENTER OF SPANS
N=03,08,01 F=0,-2500,0,0,0,0 !
SOLVE U=2 !OUTPUT P=1
;; ANALYSE AT ~ 27 YEARS
SOLVE D=00300 S=10   !OUTPUT P=1
SOLVE D=02000 S=7    !OUTPUT P=1
SOLVE D=10000 U=2 S=5 !OUTPUT P=1
STOP
```