NASA Contractor Report 4040

DYCAST—A Finite Element Program for the Crash Analysis of Structures

A. B. Pifko, R. Winter,
and P. L. Ogilvie

NASA

NASA Contractor Report 4040

DYCAST—A Finite Element Program for the Crash Analysis of Structures

A. B. Pifko, R. Winter,
and P. L. Ogilvie

Grumman Corporation Research Center
Bethpage, New York

NASA

National Aeronautics
and Space Administration

Scientific and Technical
Information Branch

1987

FOREWORD

This report describes the DYCAST computer program. DYCAST (DYnamic Crash Analysis of Structures) is a finite element program developed for structural crash simulation. As such it has as its basis the capability to perform nonlinear structural dynamic finite element analysis.

The development of DYCAST was conducted by the Grumman Aerospace Corporation, Bethpage, New York, under partial support for NASA, Langley Research Center under Contract NAS1-13148. The work was performed in the Applied Mechanics Laboratory of the Corporate Research Center with support from Grumman Data Systems. While a number of people have worked on the project during the development of DYCAST, the principal contributors are Dr. Allan B. Pifko, who served as the project's principal investigator; Robert Winter, who leads our effort associated with applications of DYCAST to practical vehicle crash simulations; and Patricia L. Ogilvie, who has the principal programming responsibilities for DYCAST. Dr. Hyman Garnet was primarily responsible for the development of the DYCAST plate element and Jacques Crouzet-Pascal has been involved with the demanding tasks associated with a number of vehicle crash simulators.

DYCAST was developed as part of a joint NASA/FAA program in General Aviation Crashworthiness. NASA's primary role in this program is the Airframe and Component Design Technology. This encompasses four general areas that are currently being addressed: full scale crash simulation testing, nonlinear crash impact analysis, crashworthy design concepts, and the development of crash resistant seats and restraint systems. DYCAST was developed in response to the second of these task, crash impact analysis.

The late Dr. Robert G. Thomson, Branch Head, Impact Dynamics Branch was in charge of the overall program. His quiet determination and sensitive leadership influenced all who were associated with the project. Dr. Robert J. Hayduk, while he was Group Leader, Impact Dynamics Branch, led the NASA effort in qualifying DYCAST for a member of practical light aircraft component crash tests. Dr. Edwin L. Fasanella of PRC Kentron played a significant role in DYCAST qualification for aircraft seat analysis as well as for transport fuselage sections as part of the FAA/NASA Controled Impact Demonstration (CID) impact test. Thanks go also to Barbara J. Durling and Martha P. Robinson who were "always there" when we needed assistance and advice in installing and running DYCAST on the Langley Computer facilities. We also acknowledge Edward Widmayer of the Boeing Commercial Airplane Co. for many helpful suggestions during the course of his work on the CID transport model.

This document consists of six sections.

Section 1 gives an overview of crash simulation methods and the theoretical basis of DYCAST. Sections 2-4 contain the bulk of the information necessary to prepare input data. Our aim in writing three sections was to present material with increasing levels of detail. Section 2, therefore, presents a brief introduction and overview of DYCAST input and should be read before proceeding to the other sections.

Section 3 was designed to be the major sequential instruction set. Once all the features of DYCAST are understood it may be the only section that the user must refer to when preparing input. This sections points to pages in Section 4, when necessary, which contain discussion of some theory as well as sample input cards. Most details are reserved for Section 4. In this manner we have kept the "cook book" section (Section 3) as brief and at the same time as readable as possible. Therefore, while you use Section 3 we maintain the following: "if you don't understand it, don't use it," and proceed to Section 4 for further details.

Section 5 contains instructions for using the DYCAST pre- and post-processors. Section 6 presents a number of sample problems and results. These were intended to give a complete picture of some DYCAST data decks as well as some simple check cases, and will supplement Sections 2-4.

One question logically arises when finite element methods for crash simulation are discussed: how detailed a model is required to simulate the salient features of a crash while still permitting the resulting analysis to be economically viable? We will not attempt to answer this question here but merely state that some expertise will be necessary in "the art" of modelling a vehicle for a crash analysis. This modelling art will require an understanding of the problem and the phenomena sought, as well as that key ingredient, engineering judgement.

TABLE OF CONTENTS

1.0 INTRODUCTION

Crashworthiness is being increasingly emphasized as a structural design requirement for occupant carrying vehicles. This requirement has been expressed recently in the form of military standards for U.S. Army troop carrying aircraft and federal motor vehicle safety standards for passenger automobiles. Consequently, by contract or by law, the crash impact condition has been added to the traditional set of structural design criteria. The goal of crashworthy design is to produce vehicles that, during a specified crash event, will reduce the dynamic forces experienced by the occupants to specified acceptable levels, while maintaining a survivable envelope around them. Generally, the structure outside of this envelope must absorb and dissipate most of the impact energy in a well controlled manner in order to fulfill this goal. In order to meet crashworthiness criteria with a minimum of effort and time, it is essential that adequate crashworthiness evaluation methods be used as early as possible in the design process.

This document describes the DYCAST Program (DYnamic Crash Analysis of STructures), a dynamic nonlinear finite element program which was developed to meet these requirements. DYCAST is an outgrowth of the PLANS system (Ref 1 and 2) of finite element programs for static nonlinear structural analysis that was originally developed by Grumman for the Langley Research Center of NASA.

Usage of DYCAST for the crash simulations of structures has been reported in Ref 3 - 9, 28-30.

Current techniques for structural crashworthiness evaluation can be characterized as experimental, hybrid, or fully theoretical. These methods have been discussed in Ref 3 and 10 - 12 and can be summarized as follows:

o Experimental -   crash tests of actual full scale vehicles or scale models

o Hybrid   -   a combined experimental and numerical method in which the structure is divided into a number of relatively large sections or subassemblies that are treated as beam/nonlinear spring elements. The crush behavior of these components as represented by the varying stiffness characteristics of the elements are determined externally by test or separate analysis

o Theoretical -   the finite element method in which the structure is divided into natural components, i.e., beam, stringer, skin panels, etc. The varying stiffness characteristics are calculated internally and depend interactively on the loading path, the material properties, and the changing shape and position of the structure.

Each of the methods outlined has its virtues and faults. Tests can provide the best accuracy and realism but can be costly and time consuming. Nevertheless, some tests are absolutely essential. For example, the full scale tests at NASA Langley Research Center are providing essential insight into many general aspects of the light aircraft situation (Ref 3 and 12). These will direct the efforts of researchers and designers into the most meaningful areas, and are providing data for verifying mathematical methods (Ref 5). Small scale model tests may also be useful, depending on the compromises between small size and realistic construction detail. Scale model tests on automobiles have been shown to yield useful results (Ref 13). At early stages of design, however, test articles may not be available for destructive evaluation.

While some impact tests will always be required to verify actual performance, theoretical crash simulation can reduce the number of tests. In this sense theoretical crash simulation can be viewed as a numerical experiment in which a discretized model of a structure is subjected to crash conditions. This method is advantageous in that once the model has been developed and validated, it can be used as many times as necessary. Consequently the effect of any design change or the sensitivity to changing any structural component can be assessed in a timely and cost effective manner.

The basic difference between hybrid and theoretical simulation models is in the manner in which they represent the details of the actual structural stiffness and mass characteristics. In the hybrid method, the vehicle is modelled by a relatively small number of lumped masses connected by nonlinear springs or beam elements. Representative structural sections are built or cut from existing vehicles and tested statically for their crush characteristics, which provide the nonlinear stiffnesses for the model. Such component crush tests can evaluate the behavior of any material or special type of construction. Alternatively, the deformation may be

approximated by analytical estimates, a detailed static finite element analysis, or educated guesses.

The external generation of crush data input can in itself be costly and time consuming. In addition the data are usually derived by varying only one force or moment at a time, whereas the actual nonlinear deformation takes place under combinations of several load components that are not known in advance. Thus it cannot be assumed that the accuracy in one particular case will be as good for a variety of impact orientations and velocity vectors because the loading combinations on the structure will vary. The number of structural elements in the model must be limited because of the engineering effort required to generate their nonlinear stiffnesses. Consequently, hybrid methods usually require less computer time than finite element methods so that, if stiffness approximations can be made, the method is suitable for providing preliminary information or gross estimates of vehicle response.

The computational problem associated with finite element crash simulation is formidable, requiring consideration of several interdisciplinary areas that include nonlinear structural mechanics, numerical analysis, and computer sciences. The solution involves: the appropriate theory to treat large elastic-plastic deformation, techniques to handle nonlinear boundary conditions required by variable contact/rebound, a library of finite elements appropriate for crash simulation, and accurate and efficient numerical time integration methods. Although investigations are still underway in each of these areas, theories have reached a sufficient level of maturity to be implemented into a program for crash simulation.

Given that there is a sufficient understanding of the theoretical aspects of crash analysis, the most vexing question associated with finite element methods is a pragmatic one. How detailed a model is required to simulate the salient features of a crash, while still permitting the resulting analysis to be made economically viable?

Experience has shown that, while an accurate, versatile computer code is essential for an adequate crash analysis, it is not enough. Some expertise in the "art" of modelling a vehicle for a nonlinear dynamic analysis is also required, in order to produce sufficiently accurate results with a minimum of time and cost. A thorough understanding of the capabiliites of the theory, and sufficient experience to know what will and will not work, is required by the analyst who prepares the model and its input data for the computer code.

This problem of modelling efficiency is much more acute in large-deflection nonlinear analysis than in the linear cases, because the solution involves a sequence of incremental steps, each one similar to a complete linear analysis in itself. Thus, a dynamic event requiring hundreds or thousands of time increments can be prohibitively costly, unless the model is reduced to the minimum complexity required to produce sufficiently accurate results.

This can often be accomplished by using some of the notions from the hybrid technique. That is, a hybrid element whose nonlinear stiffness is externally supplied can be used to model special energy-absorbing devices or structural components whose deformation characteristics are already known.

Included among these are components that have an adequate finite element representation but whose required detail would prohibitively increase the computational magnitude of the analysis.

Generally, finite element crash analysis cannot be done efficiently in a data vacuum but should use all available information, such as past impact tests on similar vehicles and existing component crush data. It is towards this end that the NASA Langley Research Center program (Ref. 3) of full scale and component tests of aircraft components are providing essential insight into the light aircraft situation.

Static crush tests on selected individual components or subassemblies can be useful to guide the modelling choices, but caution must be exercised since there are cases in which static collapse modes do not agree with the dynamic modes. Some steel structures that collapse statically after much plastic yielding can be greatly stiffened in a dynamic crush by the increase in the material's yield stress due to strain rate sensitivity. In addition, the effects of inertial forces due to added mass can significantly change the local behavior in some sensitive cases.

In the early use of nonlinear finite element models for crash analysis, "purely" theoretical approaches were attempted,[4] in which all the behavior was modelled using the finite elements. However, in the solution of practical problems involving actual vehicle structures, it quickly became apparent that some "hybrid" elements would be required, in which the user specifies the nonlinear stiffness, derived either from test data or a separate analysis. In the simplest case this would involve the modelling of a specific energy-absorbing component by a nonlinear spring with a user-specified crush curve. In the more complex cases, the collapse of a section of structure could be represented by a hybrid element, either because the crush test data were already available, or because the nonlinear behavior of a component would be so complex and so localized that it would require too much computational effort in a small part of the vehicle.

This led to a modelling strategy in which we recognize three distinct behavioral zones in a vehicle structure when preparing a nonlinear finite element model for crash analysis. These are linear behavior, moderately nonlinear behavior, and extremely nonlinear behavior. In the linear behavior zones, no nonlinear behavior is expected, and these zones are modelled as lumped masses or as rigid bodies with finite dimensions, or occasionally with a small number of deformable finite elements. In the moderately nonlinear zones, plasticity, material failure, and large deflections are expected, but the large deformations are not confined to highly localized regions. These zones are represented by a distribution of nonlinear finite elements in sufficient quantity and of the types required to allow for expected modes of deformation and failure. Here, the attempt is made to minimize the complexity while still approximating adequately the necessary stiffnesses. In the extremely nonlinear zones locally large deformations occur, such as; the collapse of a thin-wall hollow beam into accordion bellows-type folds, the complete local flattening of the cross-section of a thin-wall hollow beam to form a weak "hinge" at a bend, and the collapse of a sheet metal panel into very short waves of accordion-type folds. The theoretically accurate modelling of such components requires a large number of plate elements involving thousands of DOF for each collapse zone. The added details of these local collapse models could increase the analysis costs by orders of magnitude. A practical approach for these

components is to model them as simple nonlinear spring elements which require an input curve of force vs displacement or moment vs rotation. Thus, this local hybrid method requires the analyst to specify the expected nonlinear behavior. This method's great advantage is that only one DOF is added for each such nonlinear spring. However, if the conventional hybrid method is used, these nonlinear collapse curves are specified a priori without regard to the interactive effects of other loads acting in combination at the collapse zone. Since these combined loads can greatly reduce the collapse strength, they should somehow be taken into account.

In the case of a collapsing hinge forming in a thin-wall hollow beam, the authors have used a semi-empirical interactive method involving the use of nonlinear rotary springs imbedded between beam elements in a full-vehicle model[29]. The rotary springs are at first "rigidized" and the analysis using DYCAST is begun. The beam elements indicate the instant when lateral collapse begins as a plastic hinge forms. The analysis is then restarted at an earlier time with a revised moment vs rotation curve for the rotary spring element. This revised rotary spring curve rises to the collapse moment, then decays rapidly with increasing rotation angle. The collapse moment is determined interactively by the beam elements in the DYCAST analysis, and the shape of the rotary spring curve is taken from test experience. Typical results with this method in auto crashes predict collapse moments of hollow beams in the range of 10-50% of the theoretical fully plastic limit moment from bending acting alone. This reduced peak moment is primarily caused by the presence of a large compressive force in the beam, acting together with the hinge moment, although the other moments have an effect also. In the case of initially curved hollow rectangular beams, additional moment reduction factors have been found to weaken the collapse strength, based on a series of three-dimensional plate element analyses[31].

The total costs of an analysis are composed of the labor involved in creating the model and evaluating the results, and the costs of using the computer. Although the modelling labor cost can be large, it is rarely discussed in the technical literature, probably because of its variability. A first-time full-vehicle finite element model could require from one to four person-months of effort to prepare and verify, depending on factors such as the convenience of the vehicle geometry data (digital data base or drawings on paper), the use of computer graphics, and the experience of the personnel. In any case, modelling labor costs are dependent on the model size and complexity (quantity of nodes, elements, and DOF). However, after preparation and verification of the finite element model is complete, it can be modified easily, at small cost, enabling the investigation of the effects of structural modifications.

The computational costs are greatly dependent on model size and complexity. At the present time we consider a nonlinear vehicle crash model of 1500 DOF to be large for use on even the fastest scalar computers such as the IBM 370/3081 or CYBER 760. From two to ten restarts could be required to complete such a crash simulation. However, the new vector computers such as the CRAY-1, and the CYBER 205 allow a two to four fold increase in overall computation speed coupled with increased memory size. In the future, improvements in both software and hardware should continue to reduce computer expense to allow more detailed models to be analyzed in smaller time periods.

The methods used to implement plasticity theories into a finite element code by now are well developed and have been reported in many references (see, for example, Ref 14). Here we outline the form of constitutive equations in a general way. Additional details can be found in Ref 1 and 15). DYCAST uses a flow theory of plasticity. Basic to this approach is defining an initial yield criterion as well as flow and hardening rules. The initial yield criterion used is based on Hill's equations for orthotropic material behavior which reduces to the von Mises yield criterion for an isotropic material. From the flow and hardening rules the following incremental relation between the increments of plastic strain and stress is obtained

$$\{\Delta\varepsilon^p\} = [C] \{\Delta\sigma\} \qquad (1.2.1)$$

where the terms of $[C]$ are path dependent quantities that reflect the instantaneous states of stress and hardening of the material and the choice of plasticity theory. DYCAST uses the Prager-Ziegler kinematic hardening theory. Also contained in $[C]$ is a material parameter characterizing the hardening of the material. In the one-dimensional case this is represented by the slope of the stress versus plastic strain curve. This is generalized to multiaxial stress conditions by assuming an effective plastic strain - effective stress relation. Both linear and nonlinear strain hardening options are available with input parameters determining which is chosen. To minimize input requirements for nonlinear hardening, a Ramberg-Osgood representation of the stress-strain data is used.

$$\varepsilon = \frac{\sigma}{E} + \frac{3\sigma}{7E} \left(\frac{\sigma}{\sigma_{0.7}}\right)^{n-1} \qquad (1.2.2)$$

Thus, for this representation of the stress-strain law, two additional material parameters n and $\sigma_{0.7}$ are required.

Another assumption that is used to develop the appropriate equations is that the increment of total strain may be decomposed into an elastic and plastic component,

$$\{\Delta\varepsilon\} = \{\Delta\varepsilon^e\} + \{\Delta\varepsilon^p\} \qquad (1.2.3)$$

where $\{\Delta\varepsilon\}$, $\{\Delta\varepsilon\}^e\}$, and $\{\Delta\varepsilon^p\}$ are the increments in total, elastic, and plastic strains respectively. Equations (1.2.1) and (1.2.3) along with the incremental elastic constitutive relation

$$\{\Delta\sigma\} = [E] \{\Delta\varepsilon^e\} \qquad (1.2.4)$$

lead to the incremental constitutive relations for an elastic-plastic material

$$\{\Delta\sigma\} = [D] \{\Delta\varepsilon\} \tag{1.2.5}$$

and

$$\{\Delta e^p\} = [C][D] \{\Delta\varepsilon\} \tag{1.2.6}$$

where

$$[D] = [E^{-1} + C]^{-1}$$

and $[E]$ contains the usual elastic material parameters. As a final step we state the equivalence between Eq (1.2.5) and

$$\{\Delta\sigma\} = [E] \{\Delta\varepsilon - \Delta\varepsilon^p\} \tag{1.2.7}$$

Explicit forms for the relations in Eq. (1.2.1) that are used in DYCAST are shown in Table 1.1.

The treatment of multiaxial elastic ideally plastic behavior requires that the following conditions be satisfied:

o   The stress increment vector must be tangent to the loading surface

o   The plastic strain increment vector must be normal to the loading surface, where the loading surface is the representation in stress space of the initial yield function or the subsequent yield function after some plastic deformation has occurred.

The first condition provides a linear relationship among the components of stress increment. Thus, one of the components may be expressed in terms of the others. In a matrix form, this can be written as

$$\{\Delta\sigma\} = [\bar{E}] \{\overline{\Delta\sigma}\} \tag{1.2.8}$$

where $\{\overline{\Delta\sigma}\}$ represents the independent stress components.

The normality condition provides a linear relation among the various components of the plastic strain increment. This condition is derived from the flow rule and provides a linear relationship in which each of the components of plastic-strain increment can be written in terms of any one component. This relationship may be represented in the following form

$$\{\Delta\varepsilon^p\} = [\bar{E}] \{\tilde{\Delta\varepsilon}\} \tag{1.2.9}$$

where $\{\Delta\varepsilon\}$ is the independent plastic strain increment.

The independent increments of stress and plastic strain can be combined and written as the components of a vector, $\{\Delta\omega\}$ (see Ref 8), so that Eq. (1.2.8) and 1.2.9) can be written, respectively, as

$$\{\Delta\sigma\} = [\bar{E}] \{\Delta\omega\}$$

$$\{\Delta\epsilon^p\} = [\tilde{E}] \{\Delta\omega\} \qquad (1.2.10)$$

Examples of the explicit form of $[\bar{E}]$ and $[\tilde{E}]$ are given in Table 1.2 with $\Delta\sigma_x$, $\Delta\epsilon_x$, $\Delta\gamma_{xy}$ as the dependent quantities. Combining the above equations with Eq (1.2.3) and (1.2.4), we can form the following relation for the independent quantities

$$\{\Delta\omega\} = [E^*]^{-1} \{\Delta e^T\} \qquad (1.2.11)$$

where

$$[E^*] = [E]^{-1} [\bar{E}] + [\tilde{E}]$$

At times, due to nonproportional loading or other reasons, local unloading may occur. The unloading criterion, which is checked in the program for every load increment, is given by

$$\{m\}^t \{\Delta\sigma\} \geq 0 \quad \text{for loading or neutral loading} \qquad (1.2.12)$$

$$\{m\}^t \{\Delta\sigma\} < 0 \quad \text{for unloading}$$

where $\{m\}$ is a vector of stress dependent quantities that are based on the current loading function. Specific relations for the unloading conditon are shown in Tables 1.1 and 1.2. Equation (1.2.12) is essentially a test to determine if the stress increment vector is either tangent to or outwardly pointing from the current loading surface. The values of $\{\Delta\sigma\}$ used in this calculation are obtained from the "elastic" stress-strain relations for that increment. They are the actual $\{\Delta\sigma\}$ if unloading is detected. If the unloading criterion is not met, however, the $\{\Delta\sigma\}$ values are determined from the plasticity constitutive relations. When unloading is detected at a point, all further stress and strain increments are elastic until reloading is detected using an appropriate yield criterion.

TABLE 1.1 [C] Matrix for Various Stress States

PLANE STRESS - ISOTROPIC

$$[C] = \frac{1}{D}\begin{vmatrix} m_1^2 & & \text{symmetric} \\ m_2 m_1 & m_2^2 & \\ m_3 m_1 & m_3 m_2 & m_3^2 \end{vmatrix} \qquad \begin{aligned} m_1 &= \bar{\sigma}_x - \tfrac{1}{2}\bar{\sigma}_y \\ m_2 &= \bar{\sigma}_y - \tfrac{1}{2}\bar{\sigma}_x \\ m_3 &= 3\bar{\tau}_{xy} \end{aligned}$$

$$D = \tfrac{3}{2} c\sigma_o^2 \ , \quad \sigma_o = \text{yield stress} \ , \quad \bar{\sigma}_{ij} = \sigma_{ij} - \alpha_{ij}$$

yield function: $\qquad f = \bar{\sigma}_x^2 + \bar{\sigma}_y^2 - \bar{\sigma}_x\bar{\sigma}_y + 3\bar{\tau}_{xy}^2 - \sigma_o^2 = 0$

unloading criterion: $\ m_1 d\sigma_x + m_2 d\sigma_y + m_3 d\tau_{xy} < 0$

PLANE STRESS - ORTHOTROPIC

$$[C] = \frac{1}{D}\begin{vmatrix} m_1^2 & & \text{symmetric} \\ m_1 m_2 & m_2^2 & \\ m_1 m_3 & m_2 m_3 & m_3^2 \end{vmatrix} \qquad \begin{aligned} m_1 &= 2(G+H)\bar{\sigma}_x - 2H\bar{\sigma}_y \\ m_2 &= 2(F+H)\bar{\sigma}_y - 2H\bar{\sigma}_x \\ m_3 &= 4N\bar{\tau}_{xy} \\ m_4 &= -2F\bar{\sigma}_y - 2G\bar{\sigma}_x \end{aligned}$$

$$D = c\left(m_1^2 + m_2^2 + \frac{m_3^2}{2} + m_4^2\right) \ , \quad \bar{\sigma}_{ij} = \sigma_{ij} - \alpha_{ij}$$

yield function: $\qquad f = (G+H)\bar{\sigma}_x^2 + (F+H)\bar{\sigma}_y^2 - 2H\bar{\sigma}_x\bar{\sigma}_y + 2N\bar{\tau}_{xy}^2 = 1$

unloading criterion: $\ m_1 d\sigma_x + m_2 d\sigma_y + m_3 d\tau_{xy} < 0$

$$G+H = 1/X^2, \quad H+G = 1/Y^2, \quad F+G = 1/Z^2, \quad 2N = 1/T^2$$

X,Y,Z are yield stresses in tension in principal directions, T is yield stress in shear in principal directions

ONE NORMAL STRESS - TWO SHEAR COMPONENTS

$$[C] = \frac{1}{D}\begin{vmatrix} m_1^2 & & \text{symmetric} \\ m_2 m_1 & m_2^2 & \\ m_3 m_1 & m_3 m_2 & m_3^2 \end{vmatrix} \qquad \begin{aligned} m_1 &= \bar{\sigma}_x \\ m_2 &= 3\bar{\tau}_{xy} \\ m_3 &= 3\bar{\tau}_{xz} \end{aligned}$$

$$D = \tfrac{3}{2} c\sigma_o^2 \ , \quad \sigma_o = \text{yield stress} \ , \quad \bar{\sigma}_{ij} = \sigma_{ij} - \alpha_{ij}$$

yield function: $\qquad f = \bar{\sigma}_x^2 + 3\bar{\tau}_{xy}^2 + 3\bar{\tau}_{xz}^2 - \sigma_o^2 = 0$

unloading criteria: $\ m_1 d\sigma_x + m_2 d\tau_{xy} + m_3 d\tau_{xz} < 0$

TABLE 1.2 $[\bar{E}]$ and $[\tilde{E}]$ Matrices for Various Stress States

PLANE STRESS - ISOTROPIC

$$[\bar{E}] = \begin{bmatrix} 0 & -m_1 & -m_2 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad , \quad [\tilde{E}] = \begin{bmatrix} 1 & 0 & 0 \\ m_1 & 0 & 0 \\ m_2 & 0 & 0 \end{bmatrix}$$

$$m_1 = (\sigma_y - \tfrac{1}{2}\sigma_x) \,/\, (\sigma_x - \tfrac{1}{2}\sigma_y)$$

$$m_2 = 3\tau_{xy} \,/\, (\sigma_x - \tfrac{1}{2}\sigma_y)$$

PLANE STRESS - ORTHOTROPIC

$$[\bar{E}] = \begin{bmatrix} 0 & -m_1 & -m_2 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad , \quad [E] = \begin{bmatrix} 1 & 0 & 0 \\ m_1 & 0 & 0 \\ m_2 & 0 & 0 \end{bmatrix}$$

$$m_1 = \left((G+H)\sigma_y - H\sigma_x\right) \,/\, \left((F+H)\sigma_x - H\sigma_y\right)$$

$$m_2 = 2L\tau_{xy} \,/\, \left((F+H)\sigma_x - H\sigma_y\right)$$

$$G+H = 1/X^2, \quad H+F = 1/Y^2, \quad F+G = 1/Z^2, \quad 2N = 1/T^2$$

X,Y,Z are yield stresses in tension in principal directions, T is yield stress in shear in principal directions.

ONE NORMAL STRESS - TWO SHEAR COMPONENTS

$$[\bar{E}] = \begin{bmatrix} 0 & -m_1 & -m_2 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad , \quad [\tilde{E}] = \begin{bmatrix} 1 & 0 & 0 \\ m_1 & 0 & 0 \\ m_2 & 0 & 0 \end{bmatrix}$$

$$m_1 = 3\tau_{xy}/\sigma_x \quad , \quad m_2 = 3\tau_{yz}/\sigma_x$$

1.3 DYCAST FORMULATION - DEVELOPMENT OF EQUATIONS OF MOTION

The approach implemented in DYCAST is the updated Lagrangian formulation (Ref 16 - 18) for geometric nonlinearity. The derivation of the governing equations based on this approach follows that originally presented in Ref 16 and 17 for static analysis. The essentials of this method are that the solution is obtained incrementally, starting from a reference state, C_R, defined at time t for which the states of stress, strain, and deformation are known. The next state, C_c, termed the current state at time t + Δt is assumed to be incrementally adjacent to C_R. The problem then reduces to solving for the incremental quantities, $\{\Delta u\}$ $\{\Delta \sigma\}$, $\{\Delta \epsilon\}$ which are the increments in displacement, stress and strain in going from C_R to C_c. These quantities are all referenced to C_R so that $\{\Delta \sigma\}$ and $\{\Delta \epsilon\}$ represent the second Piola Kirchoff stress and Green's strain tensor respectively.

Once these quantities are obtained the coordinates of all points are updated by

$$\{x\} = \{X\} + \{\Delta u\} \qquad (1.3.1)$$

where $\{X\}$ is the coordinate location in C_R and the stress measure is transformed to C_c so that C_c is now the reference state for the next increment.

Based on these concepts, the equations of motion can be developed using he principle of virtual work. If D'Alembert forces are treated as body forces, we can write

$$\int_{V_R} \{S+\Delta\sigma\}^t \, \delta\{\Delta\epsilon\} dv + \int_{V_R} \rho\{\ddot{u}+\ddot{\Delta u}\}^t \delta\{\Delta u\} dv$$

$$= \int_{S_R} \{T + \Delta T\}^t \, \delta\{\Delta u\} \, ds \qquad (1.3.2)$$

where ρ is the mass density, $\{S\}$ and $\{T\}$ are stresses and surface tractions referred to C_R, and $\{\ddot{u}\}$ is the acceleration at time t in configuration, C_R. The incremental quantities $\{\Delta\epsilon\}$, $\{\Delta\sigma\}$, $\{\Delta u\}$, and $\{\ddot{u}\}$ are unknowns obtained in going from C_R to C_c. The dot notation refers to differentiation with respect to time.

The strain increment can be separated into a linear and nonlinear component,

$$\{\Delta\epsilon\} = \{\Delta e\} + \{\Delta\eta\} \qquad (1.3.3.)$$

where each component of the vector can be written as

$$\Delta e_{ij} = \frac{1}{2} \left(\Delta u_{i,j} + \Delta u_{j,i} \right)$$

$$\Delta \eta_{ij} = \frac{1}{2} \Delta u_{i,j} \Delta u_{j,i} \tag{1.3.4}$$

Substituting Eq (1.3.3, 1.2.7) into Eq (1.3.2) and neglecting terms that are cubic and quadratic in the displacement increment yields the following functional

$$\int_{V_R} \{\Delta e - \Delta \varepsilon^p\}^t \; [E] \delta \{\Delta e\} dv + \int_{V_R} \rho \{\Delta \ddot{u}\}^t \; \delta \{\Delta u\} dv + \int_{V_R} \{S\}^t \; \delta \{\Delta \eta\} \; dv$$

$$= \{r\} + \int_{S_R} \{\Delta T\}^t \; \delta \; \{\Delta u\} \; ds \tag{1.3.5}$$

where the residual load vector is

$$\{r\} = \int_{S_R} \{T\}^t \; \delta \{\Delta u\} ds - \int_{V_R} \{S\}^t \; \delta \{\Delta e\} dv$$

$$- \int_{V_R} \rho \{\ddot{u}\}^t \; \delta \{\Delta u\} dv \tag{1.3.6}$$

and represents the equations of motion in configuration C_R. In principle $\{r\}$ should be identically equal to zero. However, as will be seen, as a result of linearizing Eq (1.3.5), $\{r\}$ in practice will not be zero. Its approximation, i.e., that $\{r\}$ is less than some prescribed error bound, will serve as a basis for an iterative procedure used to satisfy the equations of motion. Equation (1.3.5) is used to develop the matrix equations of motion once the finite element assumptions are made for the displacement field in terms of nodal variables. Writing these symbolically in matrix form as

$$\{\Delta u\} = [N(x)] \; \{\Delta \bar{u}(t)\} \tag{1.3.7}$$

for each element, substituting into Eq (1.3.5) and performing the appropriate variation of displacement increments yields

$$\left(\int_{V_R} [B]^t[D][B]dv + \int_{V_R} [\Omega]^t[S][\bar{\Omega}]dv\right)\{\Delta u\}$$

$$+ \left(\int_{V_R} [N]^t \rho[N]dv\right)\{\Delta\ddot{\bar{u}}\} = \{r\} + \int_{S_R} [N]^t\{\Delta T\}ds \qquad (1.3.8)$$

where [B] maps nodal displacements to the linear component of strain, $\{\Delta e\}$; $[\Omega]$ is a matrix arising from $\{\Delta\eta\}$ and contains first derivatives of [N]; and [S], $\{S\}$ are a matrix and vector of cauchy stress referred to C_R. The matrix [D] arises from the identity of Eq (1.2.5) and (1.2.7). In the absence of plasticity it is equal to the matrix of elastic constants, [E].

The terms of Eq (1.3.8) are:

$$[k_t] = \int_{V_R} [B]^t[D][B]dv \text{ , the tangent stiffness matrix;}$$

$$[k_g] = \int_{V_R} [\Omega]^t[S][\Omega]dv \text{ , the initial stress stiffness matrix;}$$

$$[m] = \int_{V_R} [N]^t\rho[N]dv \text{ , the consistent mass matrix;}$$

$$\{\Delta p\} = \int_{S_R} [N]^t\{\Delta T\}ds \text{ , the incremental consistent load vector; and}$$

$$\{r\} = \int_{S_R} [N]^t\{T\}ds - \int_{V_R} [B]^t\{S\}dv - [m]\{\ddot{u}\} \text{ , the residual load vector.}$$

Summing these integrals over every finite element with respect to a common global coordinate system leads to the global equations of incremental motion:

$$[K_t + K_g]_n \{\Delta U\}_{n+1} + [M]\{\Delta\ddot{U}\}_{n+1} = \{\Delta P\}_{n+1} + \{R\}_n . \qquad (1.3.9)$$

Here the subscripts n and n+1 refer to configuration C_r and C_c respectively, and the change to capital letters indicates that all quantities are now referenced to the global system.

Equation (1.3.9) must be integrated in time in order to evaluate $\{\Delta U\}$ and $\{\Delta\ddot{U\}}$. Both explicit and implicit algorithms are implemented in DYCAST. These are discussed in the following sections.

Much attention has been given to methods for obtaining solutions to Eq (1.3.9). References 19 - 22 discuss methods for nonlinear dynamic analysis. The starting point for these is the choice of an appropriate scheme to integrate Eq (1.3.9) in time. Various methods for both linear and nonlinear structural analysis have been surveyed in Ref 23 and 24. We will not attempt to repeat the survey of these procedures here, but rather make some general comments on the integrators used in DYCAST.

One measure used to evaluate a time integrator is the size of the allowable time step that can be used to yield accurate solutions. At the outset we state that a significant factor affecting time step size for a nonlinear dynamic analysis is the degree of nonlinearity active in the analysis. That is, the time step must be small enough so that the assumptions intrinsic to the governing equations, i.e., plasticity theory and geometric nonlinearity, must not be violated. Because the nonlinearities may vary during an analysis, it is our view that an integrator implemented in a general purpose code for nonlinear dynamic analysis should be a variable time step procedure.

A variable time step procedure is one that enables the time step to be changed at different instants of the response, generally subject to stability and accuracy requirements. Such a procedure has obvious advantages over one with a constant time step, particularly in complex problems arising in practical applications because the system nonlinearities and dynamic response are varying continuously throughout the response history. This is particularly true for problems typical in crash simulation. Based on these comments variable time step integrators have been implemented in DYCAST. These are an explicit Modified Adams integrator (Ref 25), and the implicit Newmark-b (Ref 26) and Wilson-r (Ref 27) methods. Also implemented is a constant time step central difference explicit integrator (Ref 23).

The choice of which of these methods to use is clearer for linear problems than for nonlinear ones, with implicit methods overwhelmingly used for gross structural dynamics and explicit methods for problems where high frequency response is significant, as in the treatment of wave propagation effects. Explicit integrators are generally conditionally stable with the critical time step inversely proportional to the highest frequency in the discrete model. Implicit integrators are generally unconditionally stable for linear problems and tend to filter out the higher frequency response. This allows for larger time steps, the choice of which is controlled by the modes necessary to predict the essential features of the response. The "best" implicit method is one that can filter the unwanted higher frequency response without substantially altering the response in the lower frequency range of interest.

For linear problems using a constant time step, both methods lead to coefficient matrices that are constant through the entire response spectrum. The complicating factor for nonlinear problems is a consequence of the change in stiffness due to plasticity and geometric nonlinearities. In this case the explicit method leads to a constant coefficient matrix, but the implicit method may require the frequent reformulation of the coefficient matrix. The choice of which method to use in this case involves tradeoffs between a greater quantity of smaller less

costly time steps for explicit integration versus a lesser quantity of larger but relatively more costly time steps for implicit integration. Reference to the term "costly" here is related to the degree of complexity and magnitude of subsidiary computations during each time step.

It is convenient to recast Eq (1.3.9) into the following form when implementing an explicit time integration method.

$$[M]\{\Delta\ddot{U}\}_{n+1} = \{\Delta P\}_{n+1} + \{R\}_n + \{\Delta f\}_{n+1} \qquad (1.5.1)$$

where

$$\{\Delta f\}_{n+1} = [K_t + K_g]\{\Delta U\}_{n+1}$$

is a vector of incremental internal forces. The implication here is that these operations are performed on the element level rather than on the assembled arrays. Alternatively (Ref 23), $\{\Delta f\}_{n+1}$ can be obtained directly from the corresponding integral quantities in Eq (1.3.9). An expression for this vector is obtained directly from previously calculated variables by making use of the discrete time integrator so that the solution reduces to calculating a right hand side to Eq (1.5.1) and then solving for $\{\Delta\ddot{U}\}_{n+1}$. The term $\{R\}_n$ in Eq (1.5.1) represents an imbalance force that arises due to the linearization of the equations of motion in the nth step. It is carried forward as a correction to the $n+1^{th}$ step.

The widely used constant step central difference technique has been implemented in DYCAST. However, our preference is to use a variable time step (Ref 25), Modified Adams - Predictor - Corrector method. This integrator automatically varies the time step to reflect current system stiffness and dynamic response. Our experience with this method has been that it chooses time steps near those required by the central difference method.

The Modified Adams procedure is based on substituting a predictor solution for $\{\Delta U\}_{n+1}$ into Eq (1.5.1)

$$\{\Delta U\}_{n+1}^{Pred} = \{\Delta U\}_n^{dev} + \Delta t \{\dot{U}\}_n + \frac{\Delta t}{2}\{\dot{U}_n - \dot{U}_{n-1}\} \qquad (1.5.2)$$

Equation (1.5.2) is the Taylor series expansion for $\{U\}_{n+1}$ with the backwards difference used for the acceleration and $\{\Delta U\}_n^{dev}$ is the difference between the nth predictor and corrector solutions, $\{U_n^{Cor} - U_n^{Pred}\}$. Once $\{\Delta\ddot{U}\}_{n+1}$ is obtained from Eq (1.5.1), the corrector solution is generated based on a forward difference for the third term in Eq (1.5.2)

$$\{U\}_{n+1}^{Cor} = \{U\}_n + \Delta t \{\dot{U}\}_n + \frac{\Delta t}{2} \{\dot{U}_{n+1} - \dot{U}_n\}$$

$$(1.5.3)$$

$$\{\dot{U}\}_{n+1}^{Cor} = \{\dot{U}\}_n + \Delta t \{\ddot{U}\}_n + \frac{\Delta t}{2} \{\ddot{U}_{n+1} - \ddot{U}_n\}$$

An error criterion is used to ensure that the difference between the predictor and corrector solutions satisfies some prescribed error value. In practice, the convergence criterion usually fails on the difference between the predictor and corrector velocities. This is

$$\{\dot{U}\}_{n+1}^{Cor} - \{\dot{U}\}_{n+1}^{Pred} = \frac{\Delta t}{2} \{\Delta\ddot{U}_{n+1} - \Delta\ddot{U}_n\}$$

$$(1.5.4)$$

and the error criterion is defined using a velocity error ratio, as

$$\delta < \frac{\Delta t}{2} \frac{\Delta\ddot{U}_{n+1} - \Delta\ddot{U}_n}{\dot{U}_{n+1}} < \epsilon.$$

$$(1.5.5)$$

Whenever the error ratio is larger than the upper limit, the time step is halved. Conversely, the time step is doubled whenever the error ratio is smaller than the lower bound. It can be seen from Eq (1.5.5) that the error criterion limits the rate of change of acceleration.

In that sense, the time step control is based solely on system dynamics. The nonlinearities affect the time step only insofar as they affect system dynamics. Thus, the explicit algorithm in DYCAST depends on the small time steps necessary when using an explicit integrator to enforce the system nonlinearities.

Because of the ease in obtaining solutions to Eq (1.5.1), the computation time for an explicit method becomes strongly dependent on the element level stress-strain recovery and the formation of Δf. Computer costs are therefore directly tied to the number of elements in the discrete model and the number of time steps necessary in the analysis. This leads to the major drawback of explicit methods, namely, that more refined models have an increased frequency spectrum, requiring, for numerical stability, a smaller time step. Consequently, there is a complementary effect caused by a larger set of elements in combination with smaller time steps. Because of this situation a break-even point occurs when the economics of simpler calculations are overridden by the requirement of an increasing quantity of ever smaller time steps.

1.17

A variable time step implicit solution algorithm, based on the Newmark-β family of integrators is implemented in DYCAST. The recurrence relations for this method are

$$\{\ddot{U}\}_{n+1} = \frac{1}{\beta \Delta t^2} \{\Delta U\}_{n+1} - \frac{1}{\beta \Delta t} \{\dot{U}\}_n - (\frac{1}{2\beta} - 1)\{\ddot{U}\}_n$$

$$(1.6.1)$$

$$\{\Delta U\}_{n+1} = \Delta t \{\ddot{U}\}_n + \gamma \Delta t \{\Delta \ddot{U}\}_{n+1}$$

The parameters β and γ affect the integration accuracy and stability. For linear problems it can be shown that when $\gamma \geq 0.5$ and $\beta \geq 0.25 (0.5 + \gamma)^2$ the integrator is unconditionally stable. The case for which $\gamma = 0.5$, $\beta = 0.25$ represents the constant average acceleration method originally proposed by Newmark.

Substituting Eq (1.6.1) into Eq (1.3.9) yields

$$[\bar{K}]_n \{\Delta U\}_{n+1} = \{\Delta P\}_{n+1} + \{Q^d\}_{n+1} + \{R\}_n \qquad (1.6.2)$$

where

$$[\bar{K}]_n = [K_t + K_g + \frac{M}{\beta \Delta t^2}]_n$$

$$\{Q^d\}_{n+1} = [M] \{\frac{\dot{U}n}{\beta \Delta t} + \frac{\ddot{U}n}{2\beta}\}$$

Equation (1.6.2) is solved in two ways in DYCAST. The first is a simple incremental method with a one-step equilibrium correction where

$[\bar{K}]_n$, $\{\Delta P\}_{n+1}$, $\{Q^d\}_{n+1}$ and $\{R\}_n$ are formed at the beginning of the step. The

unknown $\{\Delta U\}_{n+1}$ is then calculated from Eq (1.6.2). The vector $\{R\}_n$ is the

imbalance force from the previous step and prevents drifting from the true solution due to the linearization of Eq (1.3.9).

This procedure is effective as long as the nonlinearities are not large in the current step. Equation (1.3.9) can be solved iteratively at each time by requiring that the equations of motion be satisfied to within some preset tolerance. In this form Eq (1.6.2) becomes

$$[\bar{K}]_n \{\Delta U\}^i_{n+1} = \{\Delta P\}_{n+1} + \{Q^d\}_{n+1} + \sum_{j=0}^{i} \{R\}^j_{n+1} \qquad (1.6.3)$$

In this equation i signifies the iteration and

$$\{R\}^j_{n+1} = \{P\}_{n+1} - \{F\}^j_{n+1} - [M] \{\ddot{U}\}^j_{n+1} \qquad (1.6.4)$$

where the terms are defined in Eq (1.3.6) and are, respectively, the vector of external forces, the vector of internal forces, and the inertia forces, all evaluated at the end of the step.

Since $\{F\}^j_{n+1}$ is evaluated taking into account all system nonlinearities, it serves as a feedback device to the linearized Eq (1.6.2). The coefficient matrix in this procedure is formed only at the beginning of the step and held constant during the iterations. The solution algorithm is therefore classified as a Modified Newton procedure.

When j = o, Eq (1.6.3) reduces to Eq (1.6.2) since

$$\{R\}^o_{n+1} = \{R\}_n$$

There are a number of ways to define convergence. DYCAST uses the following criterion:

$$\frac{\Delta U^i_{n+1} - \Delta U^{i-1}_{n+1}}{\bar{U}_{n+1}} < \varepsilon \qquad (1.6.5)$$

where $\bar{U}_{n+1} = |U_{n+1}|_{max}$, the maximum displacement within the model.

A variable time step procedure is defined by requiring that the number of iterations in each time step be less than a prescribed value. If this criterion is violated the time step is halved. Conversely if the solution converges in one iteration for a prescribed number of steps the time step is increased by a factor of 1.5. An upper bound for the time step is user specified. The static analysis procedure in DYCAST follows the same procedures as outlined above with the dynamic terms suppressed.

The basis for the derivation of the elements in DYCAST are in Ref 1. Some details are also in Section 4.3 of this report. There are currently six element types available for structural modelling, as described below.

Membrane Triangles - The membrane family of triangular elements implemented includes

    o   three-node constant strain

    o   six-node linear strain

    o   four- and five-node transitional elements.

The terms constant, linear, and transitional refer to the strain distributions that exist in the element as a consequence of choosing an assumed displacement field.

Stringer Element - This element is used to represent a one-dimensional axial force structural member. Two stringer elements are included: a two-node element developed from a linear axial displacement field and a three-node element developed from a quadratic axial displacement field.

Beam Element - There are two nodes with six degrees of freedom at each node, three displacements, and three rotations. The element is based on a linear axial displacement field and cubic transverse displacement. In the completely elastic case, the beam stiffness matrix involves elastic material properties and integrated quantities that depend on the cross section, the area and moments of inertia. Once points on the beam are plastic, these integrals must be numerically evaluated. Some details of the analysis are described below.

The beam linear strain component based on Kirchoff's hypothesis, neglecting warping of a cross section and using linearized curvatures is

$$\{\Delta e\} = [Y] \{\Delta\chi\} \qquad\qquad (1.7.1)$$

where

$$[Y] = \begin{bmatrix} 1 & 0 & z & -y \\ 0 & y & 0 & 0 \\ 0 & -z & 0 & 0 \end{bmatrix}$$

$$\{\Delta\chi\} = \{ \Delta U,_x \quad \Delta\beta_{x,x} \quad \Delta\beta_{y,x} \quad \Delta\beta_{z,x} \}$$

and y, z, are coordinate locations in the cross section, $\Delta U,_x$ $\Delta \beta_x,_x$ $\Delta \beta_y,_x$ $\Delta \beta_z,_x$ are the increments of axial strain, twist, and curvatures.

Making the assumptions for the displacement field,

$$\{\Delta \chi\} = [\phi]\ \{\Delta \bar{U}\} \qquad\qquad (1.7.2)$$

where $[\phi]$ is based on a linear function for ΔU and $\Delta \beta_x$ and a cubic for V and W.

The further assumption is made that the material stiffness properties in the plastic state vary linearly in the axial coordinate,

$$[D] = [D]_i\ (1 - \xi) + [D]_j\ \xi \qquad\qquad (1.7.3)$$

where i,j denote quantities at the two nodes, $\xi = x/\ell$ and x, ℓ are the axial coordinate and length respectively. With these assumptions, the stiffness matrix component, $[k_t]$, becomes

$$[k_t] = \ell \int_o^1 ([\phi]^t\ (1-\xi)\ \int_A [Y]^t[D]_i[Y]dA$$

$$(1.7.4)$$

$$+\ \xi \int_A^o [Y]^t[D]_j\ [Y]\ dA\ )\ [\phi]\ d\xi$$

The area integrals in Eq (1.7.4) are evaluated numerically using Gauss-Legendre integration. To accomplish this, the shape of the cross section must be known a priori and the state of stress and strain must be evaluated at each integration point in the cross section. Towards this end user-defined arbitrary cross-sections and 12 pre-defined cross sections can be specified. These are shown in Table 4.3.1. An additional point to be made is that the torsional shearing stresses are neglected in all the thin-walled open sections. However, because warping is neglected in Eq (1.7.1) for the closed sections, the numerical integration can overpredict the torsional stiffness. Because of this the terms in the second and third row of [Y] are multiplied by a "knock down" factor

$$\eta = \sqrt{\frac{J}{I_{yy} + I_{zz}}} \qquad\qquad (1.7.5)$$

where J is the user-specified torsional rigidity of the section and I_{yy}, I_{zz} are the section bending moments of inertia.

Nonlinear Spring Element - A nonlinear spring element becomes an important element in modelling a complex structure. It can be used

o   to simulate structural sections for which the axial load versus elongation behavior (or moment versus rotation) has been obtained either by a crush test or by some other means

o   to simulate an energy absorbing device

o   as a gap element to approximate variable contact/rebound

o   any combination of the above

The force versus elongation for this element is specified in tabular form. In general, nonlinear spring elements dissipate energy by unloading stiffly from their last deformation state along some specified unloading slope, thereby accumulating non-recoverable permanent deformation. Upon reloading, the path is along the unloading line to the previous maximum deformation state, at which point deformation continues along the originally specified load versus deflection curve.

The undamped nonlinear spring element in principle can be used as a gap element in order to simulate variable kinematic constraints that describe contact/rebound. However, in practice, use of a gap element in a dynamic analysis leads to high frequency oscillations because a large stiffness is associated with a small nodal mass. To surmount this difficulty viscous damping must be introduced to the nonlinear spring when used as a gap element. The damping coefficient is dependent on the current stiffness so that before contact, i.e., zero stiffness, the damping coefficient is zero. Spurious rebound can be further prevented by a "capture" technique in which both the stiffness and damping parameter are maintained as long as the contact point oscillates within a certain tolerance of the actual contact position. This technique has been effective for a number of sample problems.

Plate Bending Element - A three-node triangular plate bending element has been implemented in DYCAST. This element is based on the combination of the constant strain triangle for in-plane behavior and an implied cubic variation for the transverse deflection. The classical Kirchoff hypothesis, that plane sections remain plane, is only forced in a discrete sense along the element edges as well as the vertices. This results in a general plate bending element with three-translational and three rotational degrees of freedom at each node.

Ground Contact Element - This element is used to simulate contact between a node and a rigid plane. The perpendicular distance between the plane and the node is monitored until contact is made. Once contact is made the element acts as a nonlinear spring to enforce the contact condition.

REFERENCES

1. A.B. Pifko, H.S. Levine, and H. Armen, Jr., "PLANS - A Finite Element Program for Nonlinear Analysis of Structures, Volume I - Theoretical Manual," NASA CR-2568 (1975).

2. A. Pifko, H. Armen, Jr., A. Levy, and H. Levine, "PLANS - A Finite Element Program for Nonlinear Analysis of Structures Volume II - User's Manual," NASA CR 145244 (1977).

3. R.G. Thomson and R.C. Goetz, "NASA/FAA General Aviation Crash Dynamics Program - A Status Report," AIAA/ASME/ASCE/AHS 20th Structures, Structural Dynamics, and Materials Conference," A Collection of Technical Papers on Design and Loads, 224-232 (1979).

4. R. Winter, A.B. Pifko, and H. Armen, "Crash Simulation of Skin-Frame Structure Using a Finite Element Code," SAE Paper 770484 (1977).

5. R. Hayduk, R.G. Thomson, G. Whittlin, and M.P. Kamat, "Nonlinear Structural Crash Dynamic Analysis," SAE Paper 790588 (1979), SAE Business Aircraft Meeting, Wichita, KS, (April 1979).

6. R. Winter, A.B. Pifko, and J.D. Cronkhite, "Crash Simulation of Composite and Aluminum Helicopter Fuselages Using a Finite Element Program," AIAA, J. of Aircraft, Vol 17, No. 8, p 591 (August 1980).

7. R. Winter, M. Mantus, and A.B. Pifko, "Finite Element Analysis of a Rear-Engine Automobile," SAE Paper 811306 (1981), SAE Fourth International Conference on Vehicle Structural Mechanics, Detroit, MI, (18-20 November 1981).

8. H.D. Carden and R.J. Hayduk, "Aircraft Subfloor Response to Crash Loadings," SAE Paper 810614, SAE Business Aircraft Meeting, Wichita, KS (7-10 April 1981).

9. E. Alfaro-Bou, M.S. Williams, and E.L. Fasenella, "Determination of Crash Test Pulses and Their Application to Aircraft Seat Analysis," SAE Paper 810611, Business Aircraft Meeting, Wichita, KS (7-10 April 1981).

10. K.J. Saczalski and W.D. Pilkey, "Techniques for Predicting Vehicles Crash Impact Response," Aircraft Crashworthiness, (edited by K. Saczalski, G.T. Singley III, W.D. Pilkey, and R.L. Huston), Univ. Press of Virginia, Charlottesville, VA, 467-484 (1975).

11. K.J. Saczalski, "Modelling and Computational Solution Procedure of Prediction of Structural Crash-Impact Response," Finite Element Analysis of Transient Nonlinear Structural Behavior, (edited by T. Belytschko, J.R. Osias, and P.V. Marcal), ASME, AMD-Vol 14, 99-118 (1975).

12. E. Alfaro-Bou, R.J. Hayduk, R.G. Thomson, and V.L. Vaughan, "Simulation of Aircraft Crash and Its Validation," Aircraft Crashworthiness, (edited by K. Saczalski, G.T. Singley III, W.D. Pilkey, and R.L. Huston), Univ. Press of Virginia, Charlottesville, VA, 485-498 (1975).

13. R.S. Holmes, J.K. Gran, J.D. Colton, "Developing a New Vehicle Structure with Scale Modeling Techniques," Measurement and Prediction of Structural and Biodynamic Crash-Impact Response, (edited by K.J. Saczalski and W.D. Pilkey), ASME, 17-32 (1976).

14. H. Armen, "Plastic Analysis," Structural Mechanics Computer Programs, (edited by W. Pilkey, K. Saczalski and H. Schaeffer), Univ. Press of Virginia, Charlottesville, VA, 103-122 (1974).

15. H. Armen, Jr., A. Pifko, H. Levine, "Finite Element Analysis of Structures in the Plastic Range, NASA CR-1649 (February 1971).

16. L.A. Hofmeister, G.A. Greenbaum, and D.A. Evensen, "Large Strain Elasto-Plastic Finite Element Analysis, AIAA J., 9, 1248-1254 (1971).

17. H.S. Levine, H. Armen, R. Winter, and A. Pifko, "Nonlinear Behavior of Shells of Revolution Under Cyclic Loading," J. Comput. Structures, 3, 589-617 (1973).

18. K.J. Bathe, E. Ramm, and E.L. Wilson, "Finite Element Formulations for Large Deformation Dynamic Analysis," Int. J. Num. Meth. Engineering, 9, 353-386 (1975).

19. J.F. McNamara, "Solution Schemes for Problems of Nonlinear Structural Dynamics," J. Pressure Vessel Technology, ASME, 96, 96-102 (1974).

20. D.P. Mondkar and G.H. Powell, "Finite Element Analysis of Nonlinear Static and Dynamic Response," Int. J. Num. Meth. Engineering, 11, 499-522 (1977).

21. J.A. Stricklin and W.E. Haisler, Comments on Nonlinear Transient Structural Analysis, Finite Element analysis of Transient Nonlinear Structural Behavior, (edited by T. Belytschko, J.R. Osias and P.V. Marcal), ASME, AMD-Vol 14, 157-178 (1975).

22. T. Belytschko and D.F. Schoebeule, "On the Unconditional Stability of a Implicit Algorithm for Nonlinear Structural Dynamics," J. Appl. Mech., 97, 865-869 (1975).

23. T. Belytschko, "Transient Analysis," Structural Mechanics Computer Program, (edited by W. Pilkey, K. Saczalski and H. Schaeffer), Univ. Press of Virginia, Charlottesville, VA, 255-276 (1974).

24. J.R. Tillerson, "Selecting Solution Procedures for Nonlinear Structural Dynamics," The Shock and Vibration Digest, 7, (1975).

25. H. Garnet and H. Armen, "A Variable Time Step Method for Determining Plastic Stress Reflections from Boundaries," AIAA J., 13, 532-534 (1975).

26. N. Newmark, "A Method of Computation for Structural Dynamics," J. Eng. Mech. Div., ASCE, 85, 67-94 (1959).

27. E.L. Wilson, L. Forhoomand, and K.J. Bathe, "Nonlinear Dynamic Analysis of Complex Structures," Earthquake Eng. and Struct. Dyn., 1, 241-252 (1973).

28. Hayduk, R.J., Winter, R., Pifko, A.B.; Fasanella, E.L., "Application of DYCAST- A Nonlinear Finite Element Computer Program - To Aircraft Crash Analysis"; International Symposium on Structural Crashworthiness, University of Liverpool, England, September 1983. In Structural Crashworthiness, Ed. by N. Jones and T. Wierzlicki, Butterworth and Co., Boston 283-307, 1983.

29. Winter, R., Crouzet-Pascal, J., and Pifko, A.B., "Front Crash Analysis of A Steel Frame Auto Using A Finite Element Computer Code" SAE paper 84-0728, proceedings of the Fifth Internation Exposition on Vehicular Structural Mechanics, SAE, April 1984.

30. Winter, R. and Pifko, A.B., "Finite Element Crash Analysis of Automobiles and Helicopters," Proceedings of the International Conference on Structural Impact and Crashworthiness, Imperial College, London, July 1984. In Structural Impact and Crashworthiness, Vol 2, Ed. by J. Morton, Elselvier Applied Science Publishers, New York, 278-309, 1984.

31. Crouzet-Pascal, J., Winter, R., and Gentzlinger, R., "Crash Resistance of Curved Thin-Walled Beams", Recent Advances In Engineering Mechanics And Their Impact On Civil Engineering Practice, Vol 1, Ed. by Chen, W.F. and Lewis, A.D.M., ASCE, New York, 656-659, 1983.

This page intentionally blank

2.0 OVERVIEW OF PROGRAM INPUT

This section presents a general introduction to the card input for DYCAST. An 80 character card image format is used.

The input begins with a title card that allows for any 80 character title (specified in columns 1-80). This title serves as a page heading for subsequent computer output. The input data following this card is divided into a number of functional groups, each describing a specific type of input information. These groups are briefly described below and schematically shown in Fig. 2.3, (Page 2.6). Each input group must be read in the specific order listed below and shown in Fig. 2.1. In each group, there are usually data subgroups that begin with a key word. An index to all key words is in Table 2.3. The order of these key word subgroups within the functional group can be varied. However, if the key word subgroup contains more than one card, the sequence of those cards following the key word is fixed. In general, each group is delineated with a section end card. This is the alphanumeric SEND, left justified on the input card in columns 1 through 4. After the initial title card the input groups are as follows:

Group A - Program Control Parameters and Options

Group B - Node Specification

> This group defines an allowable set of node point identification numbers.

Group C - Element Connectivity

> Defines each element by specifying its type, (i.e., beam, triangle ... , etc.) identification number, and connecting node points.

Group D - Node Point Coordinates

> Defines the location of each node point in a global cartesian coordinate system.

Group E - Node Point Single and Multipoint Constraints

> Defines boundary conditions, nodal constraint equations, applied displacements, and applied accelerations.

Group F - Node Point Initial Conditions

> Defines node point initial displacements and velocities. This section may be omitted if all initial displacements and velocities are zero.

Group G - Added Inertia

Defines node point structural or nonstructural concentrated masses and mass moments of inertia. This group is omitted when there are no nonstructural concentrated masses and a consistent mass representation is to be used.

Group H - Element Material and Section Properties

Section properties include element thickness, area, and moment of inertia where applicable. Material properties include such quantities as Young's modulus, Poisson's ratio, and quantities defining the material nonlinearities such as yield stress, hardening parameters, and failure strains.

Group I - Applied Load and Time Function

Defines the spatial and time distribution of Applied Loads, if any. This group is omitted if there are no applied loads or applied displacements.

The last card in the deck is an alphanumeric STOP or END left justified on an input card in columns 1 through 4. STOP indicates that the job is complete and END indicates another problem input file follows.

Some general rules were used in designing the input. These rules are:

o   Most data cards that specify a new item of data begin with a "key-word" of up to five characters, left justified in their appropriate field. For example, cards specifying element connectivity for membrane triangles begin with TRIM and single point constraints begin with SPC. An index of all the key words is in Table 2.3, p. 2.7.

o   With one exception (Group B), all groups of data cards (see Fig. 2.1) end with a card containing the key word group delimiter, SEND (Section End).

o   A "$" in column 1 of an input card is used to specify a comment line. That line is ignored in the operational input stream. This allows the user to insert descriptive headings and notes, as well as temporarily deleting a line while leaving it in place.

o   The data file ends with one of two key words. If END is used, another problem data file follows. If STOP is used, as will probably be the case for most problems, the job ends.

o   Generally, two formats for input are used, E15.0 for floating point input (fields of 15) and I5 for integer or fixed point input (fields of 5). The fixed point (integer) data must be right justified. The floating point data can be written in several forms. For example 10.0 can be input as:

   10.0 any place in the field, or 1.0 E+01, 1.0 E+1,
   1.0E1, or 10, where the entry is right justified in the field.

o   There are a number of places in the program where lists of applicable node or element numbers must be specified with a set of data. In these cases the nodes or elements are specified by entering the appropriate numbers on the input cards in fields of five. However, for this purpose the user can also utilize a shorthand form of the input. That is, specifying m and $-n$ consecutively is the equivalent of the specification of nodes or elements m, $m+1$, $m+2$, ...n and specifying m, $-p$, and $-n$ consecutively is the equivalent of the specification of nodes (elements) m, $m+p$, $m+2p$ ...$m+kp$ where $m+kp$ is the highest integer in the sequence less than or equal to n. For example, the specification of nodes 1 through 100 is written as 1-100 and nodes, 1, 3, 5, ... 99 as 1 - 2 - 99. This card input appears in fields of 5 (I5 format). Any number of continuation cards may be used. A blank I5 field ends the specification. If any such card ends in the last field, a blank card must follow.

o   Each input card set is described in the following section by first stating its key word and then, in tabular form, describing the

    (a)   FORTRAN format

    (b)   columns on the input card reserved for the data

    (c)   descriptive symbols or names of the data

    (d)   brief comments - reference is made here where necessary, to a more in-depth discussion in Section 4.

o   Physical units must be consistent in the system used. For example, if inches, seconds, and lb are units for length, time, and force, then lb sec^2/in, lb/in^2, and in/sec^2 must be the units for mass, stress, and acceleration . Units for several systems are shown in Table 2.1. The generic units for each input are indicated in parenthesis in the text, with (L) referring to length, (T) referring to time, (F) referring to force, and (M) referring to mass.

o   Some input parameters have default values already stored in the program, as noted in the text. If the entry is left blank, the default value will be used; otherwise, the entered value is used. Unless otherwise stated, the default values are zero.

o   In most data input groups, duplicate inputs are not accepted, so that the last entry value for a particular parameter will replace any previous value. The only exception is Group G, Added Inertia, in which the values of inertia assigned to each node are added to any values input previously. This accumulation feature is a convience to the user. Inertia can be subtracted by assigning a negative number. Inertia can also be changed during the course of an analysis, at a restart.

Table 2.1. Consistent Sets of Units

Length Displacement (L)	Time (T)	Force ($F=ML/T^2$)	Energy, Moment (FL)	Mass ($M=FT^2/L$)	Rotary Inertia (ML^2)	Accel. (L/T^2)	Stress (F/L^2)
in	sec	lb	lb in	lb sec^2/in	lb sec^2 in	in/sec^2	lb/in^2
ft	sec	lb	lb ft	lb sec^2/ft	lb sec^2 ft	ft/sec^2	lb/ft^2
cm	sec	dyne	dyne cm	gm	gm cm^2	cm/sec^2	dyne/cm^2
m	sec	N	N m	kg	kg m^2	m/sec^2	N/m^2
mm	sec	kg	kg mm	kg sec^2/mm	kg sec^2 mm	mm/sec^2	kg/mm^2
mm	sec	N	N mm	N sec^2/mm	N sec^2 mm	mm/sec^2	N/mm^2*

---

*=M Pa

TABLE 2.2 MAXIMUM QUANTITIES FOR INPUT DATA

INPUT ITEMS	MAXIMUM QUANTITY
Time functions: ACELT + PTME + PTM2 + PTM3	36 tables, 50 points per table
Multipoint constraint equations: APPL + MPC	200 dependent nodes, 500 coefficients
Beam element loads : BMLO	100 sets
Beam and stringer element sections: CSEC + HCIR + HREC + HSEC + ISEC + LSEC + LSEG + MSTG + SCIR + SREC + TSEC + TWD + ZSCR + ZSEC	200 sets
Plate and membrane element material properties: MAT1 + MAT2 + MAT3	20 sets
Beam element material properties: MBM	50 sets
Contact element and nonlinear spring properties: PGRD + PSPR	60 tables, 20 points per table
Single point constraints: SPC	200 different 6-digit NBND code words
Plate element loads: SURF	100 sets
Plate and membrane element thickness: THIK	100 sets

- - - - -

Note: 1. "Table" means a function defined by pairs of numbers (for example; force and displacement, load and time).

2. "Set" means input data followed by a list of applicable elements.

3. There are no limits to the quantities of nodes or elements, except those imposed by the user to maintain economy of computational expense (CPU) and modelling labor.

STOP

END OF DATA

SEND

CONC   50.0
GROUP I
    APPLIED LOAD DISTRIBUTION AND TIME HISTORY

SEND

MBM   11.2 E+06   0.333
GROUP H
    ELEMENT MATERIAL AND SECTION PROPERTIES

SEND

CNMI   3.45   E-04
GROUP G
    NODE POINT STRUCTURAL OR NON STRUCTURAL
    CONCENTRATED MASS AND MASS MOMENT OF INERTIA

SEND

IVEL   -528.0
GROUP F
    NODE POINT INITIAL DISPLACEMENT AND VELOCITIES

SEND

SPC 101010
GROUP E
    NODE POINT SINGLE AND MULTIPOINT CONSTRAINTS

SEND

GRDX   10   1   3
GROUP D
    NODE POINT COORDINATES

SEND

BEAM   1   1   2   10
GROUP C
    ELEMENT CONNECTIVITY

SEND

1-100
GROUP B
    NODE SPECIFICATION

SEND

DYNA   0   10   100
GROUP A
    PROGRAM CONTROL PARAMETERS

*** DYCAST EXAMPLE PROBLEM ***
PROBLEM TITLE CARD

Fig. 2.1 Input Data Organization

2.6

TABLE 2.3 INDEX TO KEY WORDS

WORD	GROUP	USAGE	PAGE
ACEL	E	DOF for Applied Accelerations	3.47
ACELT	I	Time History of Applied Accelerations	3.108
ADAM	A	Modified Adams explicit time integrator	3.5
APPL	E	Applied displacements and rotations of nodes	3.47
BAND	A	Re-order node list to reduce bandwidth	3.24
BEAM	C	Beam element connectivity	3.27
BMLO	I	Distributed Loads on beam element	3.99
CDIF	A	Central difference explicit time integrator	3.6
CNM1	G	Concentrated mass and rotary inertia	3.55
CONC	I	Concentrated force and moment at a node	3.98
CONM	A	Consistent nondiagonal mass matrix	3.11
CSEC	H	Section properties for C-section beam elements	3.74,76
DELE	A	Manually delete (fail) members	3.18
DYNA	A	First two control cards	3.5,7
EIGN	A	Free vibrations	3.10
END	–	End a problem input, another problem follows	3.111
EOFF	A	Turns off input data echo	3.21
FAIL	A	Automatic member failure	3.20
GRAV	I	Gravity Loading	3.107
GRDS	C	Contact Element Connectivity	3.28
GRDX	D	X-Coordinate for a set of nodes	3.34
GRDY	D	Y-Coordinate for a set of nodes	3.35
GRDZ	D	Z-Coordinate for a set of nodes	3.36
HCIR	H	Section properties for hollow circle beam elements	3.74,77
HREC	H	Section properties for hollow rectangle beam elements	3.74,78
HSEC	H	Section properties for hat section beam elements	3.74,79
IDSP	F	Initial displacements and rotations	3.50
ISEC	H	Section properties for I-Section beam elements	3.74,80
IVEL	F	Initial velocities	3.52
LSEC	H	Section properties for L-section beam elements	3.74,81
LSEG	H	Section properties for thin rectangle beam elements	3.74,82
LUMP	A	Lumped diagonal mass matrix	3.12
MAT1	H	Isotropic material properties for plane stress elements	3.59
MAT2	H	Orthotropic material properties for plane stress with perfect plasticity	3.61

TABLE 2.3   INDEX TO KEY WORDS (Continued)

WORD	GROUP	USAGE	PAGE
MAT3	H	Orthropic material properties for plane stress with strain hardening	3.64
MBET	H	Angle between local element axes and principal directions of orthotropy	3.69
MBM	H	Material properties for beam elements	3.72
MPC	E	Multipoint constraints	3.46
MPRIN	A	Print member solution data	3.15
MSTG	H	Material and section properties for stringer elements	3.70
MSTR	A	Print stress and strain details for beam elements	3.23
NEWM	A	Newmark-Beta implicit time integrator	3.7
NPRIN	A	Print nodal solution data	3.16
PFTAB	A	Print summary table of plastic and failed elements	3.19
PGRD	H	Contact Element Properties	3.92
PITC	D	Y-Rotation of vehicle coordinate system	3.41
POFF	A	Turns off printout of processed input data	3.22
PSPR	H	Properties for spring elements	3.90
PTME	I	Time function for all applied loads and displacements	3.102
PTM2	I	Time function for all loads at a node	3.103
PTM3	I	Time function for a load component at a node	3.105
REST	A	Restart parameters	3.17
ROLL	D	X-Rotation of vehicle coordinate system	3.42
SCAN	A	Scan input for errors, no solution	3.14
SCIR	H	Section properties for solid circle beam elements	3.74,84
SEND	-	Ends data for a group	----
SPC	E	Single point constraints	3.44
SPNG	C	Spring element connectivity	3.29
SREC	H	Section properties for solid rectangle beam elements	3.74,85
STAT	A	Static analysis	3.9
STOP	-	End a problem input, last card	3.111
STRG	C	Stringer element connectivity	3.30
SURF	I	Distributed loads on plate elements	3.100
THIK	H	Thickness and layers for plate and membrane elements	3.68
TIME	A	Print breakdown of computer time	3.13
TRIM	C	Triangular membrane element connectivity	3.31
TRP2	C	Triangular plate element connectivity	3.32
TSEC	H	Section properties for T-section beam elements	3.74,86
TWD	H	Section properties for beam element constrained to move in one plane	3.74,87

TABLE 2.3   INDEX TO KEY WORDS (Continued)

This page intentionally blank

3.0   INPUT PREPARATION

This section serves as a basic guide for preparing input data for DYCAST. As such, it is our intention to write a concise and easily followed set of instructions. To accomplish this goal the depth of some of the explanations is limited here. Further in-depth explanations are reserved for Section 4 which serves as a more detailed reference. Specific reference is made to these sections as needed.

3.1   PROBLEM TITLE

```
/ Card 1:   Any 80 Character Title /
```

This 80-character title card serves as a heading for all pages of output. Any number of continuation title cards can be specified. This is indicated with three blanks and a comma in columns 77 through 80. The continuation title cards only appear on the first page of the output. Additional problem description can be included on any number of comment cards following the title card. These comment cards begin with a "$" in column 1, and will appear only on the first page of the output.

NOTES:

1.   This card must be the first one in the data file.

2.   A "$" in column 1 of any card in the input stream indicates that it is a comment card, to be ignored by the program.

3.2 GROUP A - PROGRAM CONTROL PARAMETERS AND OPTIONS

Only four cards in this group are required; two DYNA cards, a time integrator card, and the SEND card.

/ Key Word DYNA /

This required card must always follow the title card. (Section 4.1.1)

Format	Columns	Symbol	Comments
A4	1-4	DYNA	Key word
	5,6		Blank
4I1	7-10	NPNTC	Prints diagnostic data (p. 4.1). Enter a 1 in Col. 7 to print load vector, 1 in Col 8 for element mass and stiffness matrices, 1 in Col. 9 for coeff. matrix entries and their stacking indices, 1 in Col. 10 for assembled coeff. matrix. Default = 0000.
I5	11-15	NPRNT	Results printed every NPRNT time steps, (default is last time step, corresp. to FTIME, p. 3.4). Note 1.
I5	16-20	NFORM	Coefficient matrix reformed every NFORM time steps (see Section 4.1.1). Default = 99999. Note 2.
	21-80		Blank

NOTES:

1. NPRNT controls the frequency of printout requested by the TIME, MPRIN, NPRIN, PFTAB, and MSTR cards defined below.

2. The coefficient matrix is the mass matrix for explicit time intgrators (CDIF and ADAM) and is a combination of the mass and stiffness matrices for implicit integrators (STAT, NEWM, and WLSN). NFORM is set to 1 internally for implicit integration. For explicit, use the default.

3. The program is dimensioned for 1000 nodes and 1000 elements (MXNOD = 1000, MXMEM = 1000). The amount of core required is according to the following relations:

$$\text{NCORE } J \geq 22 \text{ (MXNOD)} + 9 \text{ (MXMEM), to process input}$$
$$\text{NCORE } J \geq 10 \text{ (No. of DOF), to enter calculation phase}$$

The program will use the more efficient in-core solution (stack the coefficient matrix in one pass) if:

$$\text{NCORE } J \geq SN - .5S^2 + N + .5S + 2000$$

where S = Maximum semi-bandwidth, N = No. of DOF

This required card must always follow the previous DYNA card.

Format	Columns	Symbol	Comments
A4	1-4	DYNA	Key Word
	6-20		Blank
E15.0	21-35	FTIME	Maximum time (T)
E15.0	36-50	DTIME	Initial time increment. Note 1.
E15.0	51-65	TIC	Initial reference time, default = 0 (T)
	66-80		Blank

NOTES:

1. If no maximum time step (DTMAX) is specified on the implicit integrator cards NEWM, WLSN, or STAT, then DTIME becomes the maximum time step.

2. The succeeding cards in this group can be in any order.

/ Key Word ADAM /

This card is specified <u>only if</u> the Modified Adams explicit time integrator is to be used. (Sections 1.5 and 4.1.3)

Format	Columns	Symbol	Comments
A4	1-4	ADAM	Key word
	5		Blank
E15.0	6-20	EPSIL	Upper bound convergence error. Default is .10 (10%)
E15.0	21-35	DELTA	Lower bound convergence error. Default is .01 (1%)
E15.0	36-50	ERR	Tolerance on convergence test. Default is $1. \times 10^{-4}$ (Section 4.1.3)
	51-80		Blank

NOTES:

1. <u>One</u> of the following key words must be input to specify the integrator or type of solution: ADAM, CDIF, NEWM, WLSN, STAT, EIGN.

This card is specified <u>only if</u> the central difference explicit time integrator is to be used.

Format	Columns	Symbol		Comments
A4	1-4	CDIF	Key word	
	5-80		Blank	

<u>NOTES</u>:

1. <u>One</u> of the following key words must be input to specify the integrator or type of solution: ADAM, CDIF, NEWM, WLSN, STAT, EIGN.

This card is specified <u>only if</u> the Newmark-BETA implicit time integrator is used. (Section 1.6)

Format	Columns	Symbol	Comments
A4	1-4	NEWM	Key word
	5		Blank
E15.0	6-20	BETA	Parameter beta appearing in expression for displacement in Newmark - Beta method (default is .25 if <u>both</u> BETA and GAMA are blank). If <u>GAMA</u> is non-zero and BETA is zero then $\beta = (.5 + \gamma)^2/4$.
E15.0	21-35	GAMA	Parameter gamma appearing in expression for velocity in Newmark-Beta method (default is .5 if field is blank).
E15.0	36-50	TOL	Limit on convergence test for iterations within a time step. (Default is .01) i.e., $$\left\| \frac{\Delta U^i - \Delta U^{i-1}}{U} \right\| < TOL$$ where i = iteration number
E15.0	51-65	INTER	Maximum number of iterations per time step (default is 10). If INTER < 0 the iterations are suppressed and a constant time step is used
E15.0	66-80	DTMAX	Maximum allowable time step (default is original DTIME from second DYNA card p. 3.4) (T)

NOTES:

1. <u>One</u> of the following key words must be input to specify the integrator or type of solution: ADAM, CDIF, NEWM, WLSN, STAT, EIGN.

2. This procedure assumes a consistent mass matrix. If only lumped mass is desired, set the density equal to zero on all material property cards in Group H, and add the lumped mass to the nodes in Group G.

This card is specified <u>only if</u> the Wilson-Theta implicit time integrator is used.

Format	Columns	Symbol	Comments
A4	1-4	WLSN	Key word
	5		Blank
E15.0	6-20	THETA	Parameter theta in Wilson method (default is 1.4)
	21-35		Blank
E15.0	36-50	TOL	Limit on convergence test for iterations within a time step (default is .01), i.e., $$\left| \frac{\Delta U^i - \Delta U^{i-1}}{U} \right| < TOL$$ where i = iteration number
E15.0	51-65	INTER	Maximum number of iterations per time step (default is 10). If INTER < 0 the iterations are suppressed and a constant time step is used
E15.0	66-80	DTMAX	Maximum allowable time step (default is original DTIME from second DYNA card p. 3.4) (T)

NOTES:

1. <u>One</u> of the following key words must be input to specify the integrator or type of solution: ADAM, CDIF, NEWM, WLSN, STAT, EIGN.

2. This procedure assumes a consistent mass matrix. If only lumped mass is desired, set the density equal to zero on all material property cards in Group H, and add the lumped mass to the nodes in Group G.

This card is specified <u>only if</u> a static analysis is to be used. (Section 4.1.5)

Format	Columns	Symbol	Comments
A4	1-4	STAT	Key word
	5-35		Blank
E15.0	36-50	TOL	Limit on convergence test for iterations within a time step (default is .01) i.e.,

$$\left| \frac{\Delta U^i - \Delta U^{i-1}}{U} \right| < TOL$$

where i = iteration number

Format	Columns	Symbol	Comments
E15.0	51-65	INTER	Maximum number of iterations per time step (default is 10). If INTER < 0 the iterations are suppressed and a constant time step is used.
E15.0	66-80	DTMAX	Maximum allowable time step for computations of load increment (default is the original DTIME from the second DYNA card, p. 3.4) (T)

<u>NOTES:</u>

1. <u>One</u> of the following key words must be input to specify the integrator or type of solution: ADAM, CDIF, NEWM, WLSN, STAT, EIGN.

2. Static loads and displacements are applied incrementally by calculating the product of the load distribution factors (Group I) or displacement factors (APPL, Group E) times the time function PTME in Group I. See Section 4.6.

This card is specified <u>only if</u> a free vibration analysis is desired. (Section 4.1.6)

Format	Columns	Symbol	Comments
A4	1-4	EIGN	Key word
	5		Blank
E15.0	6-20	IQ	Number of modes desired, up 100.
E15.0	21-35	NSBAR	Number of times reorthogonalization is to be performed. Usually 2 is sufficient
E15.0	36-50	LPRINT	If set equal to 1 the factored stiffness matrix and mass matrix are printed, otherwise zero or blank
E15.0	51-65	IPRINT	If set equal to 1 intermediate debugging information is printed
E15.0	66-80	EPS	Shift parameter used when the stiffness matrix is singular (Section 4.1.6) Default = 0.

NOTES:

1. One of the following key words must be input to specify the integrator or type of solution: ADAM, CDIF, NEWM, WLSN, STAT, EIGN.

/ Key Word CONM/

This optional card specifies that the structural mass will be calculated using a consistent mass approach. This is the program default condition.

Format	Columns	Symbol		Comments
A4	1-4	CONM	Key word	
	5-80		Blank	

NOTES:

1. The nodal inertias specified in Group G form a diagonal matrix which is added to this structural mass matrix.

This optional card specifies that <u>all</u> the mass properties of the structure have been defined using concentrated (or lumped) masses and inertias. This card denotes a diagonal mass matrix for explicit integration.

Format	Columns	Symbol		Comments
A4	1-4	LUMP	Key word	
	5-80		Blank	

NOTES:

1. Default condition is consistent mass (CONM).

2. Lumped masses are not internally calculated for the elements. They <u>must</u> be input in Group G.

3. Lumped (diagonal) mass matrices are <u>not</u> reformed, regardless of the NFORM value specified on the first DYNA card, P. 3.2.

4. This card will be ignored for implicit integrators (NEWM, WLSN). However, a diagonal (lumped) mass matrix can be used with implicit integrators by specifying the proper nodal masses in Group G <u>and</u> using zero material densities in Group H (the latter to suppress the consistent structural mass matrix).

<u>/ Key Word TIME/</u>

This optional card initiates printout every NPRNT steps (P. 3.2) of computer time (CPU) used at various parts of the computation. (See Section 4.1.2 for an example of this output)

<u>Format</u>	<u>Columns</u>	<u>Symbol</u>		<u>Comments</u>
A4	1-4	TIME	Key word	
	5-80		Blank	

<u>NOTES</u>:

/ Key Word SCAN/

This optional card indicates that the input is to be read and processed but operations will stop just <u>before</u> calculating and forming the stiffness and mass matrices. This feature is used to check input data before a full job execution is tried.

Format	Columns	Symbol		Comments
A4	1-4	SCAN	Key word	
	5-80		Blank	

NOTES:

/Key Word MPRIN/

This optional set of at least two cards specifies the elements for which stress, strain, and load information will be printed every NPRNT steps (p. 3.2). <u>Default condition: If this card is not specified, printout will occur for every element.</u> In order to suppress all element printout use the key word MPRIN followed by a blank card.

Card 1:

Format	Columns	Symbol		Comments
A5	1-5	MPRIN	Key word	
	6-80		Blank	

Card(s) 2:

The second and succeeding card(s) contain the elements to be printed as indicated below.

Format	Columns	Symbol	Comments
16I5	1-80	MEM	Elements for which data is to be printed are specified in fields of 5 (I5 format). The short form notation and also any number of continuation cards may be used. A <u>blank</u> I5 field ends the specification of the elements to be printed.

NOTES:

1. If an element list ends in the last field, a blank card <u>must</u> follow.

2. For beam elements, the default condition is to print a short table for each beam, deleting the stresses and strains through the cross-section. If these beam stress & strain data are to be printed, the MSTR card (p. 3.23) must also be used.

3. On a restart, if this card set is not included, MPRIN defaults to the input used on the initial segment.

This optional set of at least two cards specifies the nodes at which displacement, velocity and acceleration are to be printed every NPRNT steps (p. 3.2). Default condition: If this card is not specified printout will occur for every node. In order to suppress all node printout use the NPRIN followed by a blank card.

Card 1:

Format	Columns	Symbol		Comments
A5	1-5	NPRIN	Key word	
	6-80		Blank	

Card(s) 2:

The succeeding card(s) after the NPRIN card contain the nodes to be printed as indicated below.

Format	Columns	Symbol	Comments
16I5	1-80	NODE	Nodes for which data are to be printed are specified in fields of 5 (I5 format). The short form notation and also any number of continuation cards may be used. A blank I5 field ends the specification of nodes to be printed.

the

NOTES:

1. If a node list ends in the last field of a card, a blank card must follow.

2. On a restart, if this card set is omitted, NPRIN defaults to the input used in the initial segment.

3.16

<u>/ Key Word REST/</u>

This optional card specifies information for creating an external data base or "restart" file, which is required for post-processing or restarting the analysis. (Section 4.1.4, p. 4.6)

Format	Columns	Symbol	Comments
A4	1-4	REST	Key word
	5		Blank
E15.0	6-20	IREST	= 0. No restart file written (default when this card is omitted). = 1. Initiate a restart file, written on unit 21. = 2. Continue from existing restart file. The old file remains unchanged on unit 21. Old and new output data will be written on a new file on unit 22, if NUTAP > 0 below. = 3. Continue from existing restart file. New output data will be added to the existing file on unit 21, if NUTAP > 0 below.
E15.0	21-35	NUTAP	When IREST = 0. or 1., NUTAP is ignored. When IREST = 2. or 3., set NUTAP = 0. for no new restart file, or NUTAP = 1. to create new restart file.
E15.0	36-50	IPRNT	Restart data written every IPRNT time steps to allow restart in a later continuation run. If equal to zero, restart data are written only at the final time. Note 2.
E15.0	51-65	NRSRT	This number defines the starting time for the continuation job from a restart file. It is one of a set of NRSRT numbers, each associated with a specific time, that are printed out in the job that generated the restart file. Restart segments can begin at any time for which an NRSRT number has been specified.
	66-80		Blank

<u>NOTES:</u>

1. Only title card and Group A input data are required with a restart.

2. IPRNT should be chosen to be as large as possible to reduce I/O and storage requirements.

3. Group G data can also be specified with a restart (i.e. added inertia).

__/ Key Word DELE/__

This optional set of at least two cards specifies elements to be failed manually. __This card is optional and only has meaning when a problem is being restarted.__ (Section 4.1.9)

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	DELE	Key word
	5-80		Blank

Card(s) 2:

The second and succeeding card(s) contain the elements to be deleted.

Format	Columns	Symbol	Comments
16I5	1-80	MEM	Elements to be deleted are specified in fields of 5 (I5 format). A blank I5 field ends the specification of the deleted members. (The short form input notation is __not__ permitted here.)

NOTES:

1. If an element list ends the last field of a card, a blank card __must__ follow.

2. A deleted element cannot be subsequently restored. Therefore it is necessary to specify its number here only once. However, it will be convenient to leave the previously specified elements on the card, and add new ones as desired, at each subsequent restart.

3. The mass of a deleted element remains at its nodes. Nodes completely detached from the rest of the structure become separate bodies.

4. Mass can be added or subtracted in Group G at any node on a restart. Mass can be completely deleted on a restart by subtracting its entire value.

If this optional card is input a table is printed every NPRNT steps
(p. 3.2), indicating the elements that have been plastic, failed, or
deleted. (Section 4.1.8)

Format	Columns	Symbol	Comments
A5	1-5	PFTAB	Key word
	6-80		Blank

NOTES:

If this optional card is used, the stiffness and stress at each
integration point of an element will be deleted when its specified failure
criterion has been satisfied. The failure criteria are specified along
with other material properties in Group H. If no failure criteria are
specified for an element, its failure will not be evaluated. If this card
is omitted, no deletion action will be taken, but a warning message will
be printed when complete failure of an element would have occurred. See
p. 4.10.

Format	Columns	Symbol		Comments
A4	1-4	FAIL	Key word	
	5-80		Blank	

NOTES:

1.   Element continues to carry load if the FAIL card is not specified.

<u>/Key Word EOFF/</u>

This optional card will delete the printout (echo) of the user's input data stream, except that the Group A data (Controls) echo will still be printed.

Format	Columns	Symbol	Comments
A4	1-4	EOFF	Key Word
	5-80		Blank

<u>Notes:</u>

/Key Word POFF/

This optional card will delete the printout of the processed input data that follows the echo of input data, except that the processed Group A data (controls) will still be printed.

Format	Columns	Symbol	Comments
A4	1-4	POFF	Key word
	5-80		Blank

Notes:

If this optional card is specified, stress and strain output will be
printed for each integration point in the cross section at both ends of
beam elements for which printout has been requested by the MPRIN card set.
(p. 3.15)

Format	Columns	Symbol		Comments
A4	1-4	MSTR	Key word	
	5-80		Blank	

NOTES:

1. If this card is used, the beam printout will be one beam element per page,
    compared to four per page without this card.

This optional card causes the program to re-order the Group B node list to produce a reduced bandwidth of the coefficient matrix, in order to reduce the core storage requirements and the computer processing time. After finding the optimum (or near-optimum) nodal order for minimizing the bandwidth, the program will use this re-ordered node list as the Group B input and proceed with the problem calculations. The re-ordered Group B input is also written as an output file on unit 7 in the proper format for direct use as input for subsequent problems.

Format	Columns	Symbol	Comments
A4	1-4	BAND	Key word
	5	Blank	
E15.0	6-20	JMAX	Maximum number of nodes connected to any one node. Default = 12.
	21-80		Blank

NOTES:

1. For large problems, this card is essential to reduce computer resource utilization.

2. MPC connectivities are automatically included.

3. This BAND operation also works with SCAN (p. 3.14) as a pre-processor for re-ordering the node list, without proceeding with the solution.

4. The re-ordered node list from unit 7 should be used as input for Group B, p. 3.24, and the BAND card should be deleted for subsequent runs to save CPU time.

<u>/ Key Word SEND/</u>

    <u>This required card ends Group A data.</u>

<u>Format</u>	<u>Columns</u>	<u>Symbol</u>		<u>Comments</u>
A4	1-4	SEND	Key word	
	5-80		Blank	

<u>NOTES</u>:

1.   If this is a restart job only Group A input is required. In this case, this SEND card is followed by a STOP or END card, p. 3.111.

2.   Mass can be changed on a restart run by adding Group G data immediately following Group A, then a final STOP or END card, p. 3.111.

3.3 GROUP B - NODE SPECIFICATION

This required input group defines the allowable set of node numbers. Every node specified in the subsequent input groups must be listed here. While the node numbers can be in any order, they are converted within the program to consecutive internal numbers. Any quantity of cards may be used. See Note 2 below and Section 4.2.

/Key Word	No Key Word for this Group /		
Format	Columns	Symbol	Comments
16I5	1-80	NODE	The list of node numbers in fields of 5 (I5 format). The short form notation (p. 2.3) and blank fields are allowed. Note 1.

NOTES:

1. If the list ends in the last field of any card, a blank card <u>must</u> follow to end this group.

2. This nodal sequence should be optimized in large problems to produce minimum semi-bandwidth. This can be done using the SATELLITE preprocessing program (Section 5.1) or the BAND card, p. 3.24.

3. A SEND card <u>must not</u> be used to end this group.

3.4 GROUP C - ELEMENT CONNECTIVITY

This group requires one card for each element. Input cards may be in any order, although it will be convenient to list them in order of element number. The key word for each card is a four character descriptor defining the element type. The current element library is as follows:

BEAM	–	Beam
GRDS	–	Ground contact element
SPNG	–	Nonlinear spring
STRG	–	Stringer (rod)
TRIM	–	Membrane triangle
TRP2	–	Plate triangle

For details about these elements refer to Section 4.3.

/Key Word BEAM/

One card required for each beam element (Section 4.3.3).

Format	Columns	Symbol	Comments
A4	1-4	BEAM	Key word
	5-10		Blank
I5	11-15	MEM	Element identification number
I5	16-20	NODEI	Node i
I5	21-25	NODEJ	Node j, defines beam + x axis (longitudinal)
I5	26-30	NODEK	Node k, defines beam + y-axis (transverse)
	31-80		Blank

NOTES:

1. Since node k defines the rotational orientation of the beam about its longitudinal axis, its position must be coordinated with the location of the beam y-axis defined in Table 4.3.1 of Section 4.3.3.

2. The three nodes must be separated by finite distances, and must not be colinear.

<u>/Key Word GRDS/</u>

One card required for each ground contact element (Section 4.3.7).

Format	Columns	Symbol	Comments
A4	1-4	GRDS	Key Word
	5-10		Blank
I5	11-15	MEM	Member identification number
I5	16-20	NODE	Contact node
I5	21-25	NDIR	Global direction closest to the normal to the contact plane, 1 = x 2 = y 3 = z
	26-80		Blank

<u>NOTES</u>:

3.28

One card required for each nonlinear spring element (Section 4.3.5)

Format	Columns	Symbol	Comments
A4	1-4	SPNG	Key word
	5-10		Blank
I5	11-15	MEM	Element identification number
I5	16-20	NODEI	Node i
I5	21-25	NODEJ	Node j, defines spring + x axis
I5	26-30	NODEK	Global direction indicator, optional for translational springs, required for the rotational spring, see notes below. Default = 0.
	31-80		Blank

Notes:

1. For translational springs, NODEK = blank, 0,1,2, or 3. If blank or 0, the spring acts on a line between the two nodes i and j. If 1,2, or 3, the spring acts parallel to the global x,y, or z axes between the nodes.

2. For rotational springs, NODEK = 4,5, or 6, and the spring will rotate about the global x,y, or z axes. The positive senses of the relative nodal rotation and of the moment must be coordinated. See Section 4.3.5.

3. A "grounded" spring can be defined by setting NODEJ = 0, thereby anchoring node j to the origin of coordinates. This can be useful, with globally oriented translational springs (Note 1) in defining interaction normal to a fixed barrier or ground plane.

4. Translational springs must have their nodes separated by a finite distance, in order for their directions to be calculated, except for globally oriented springs whose directions are specified.

3.29

<u>/Key Word STRG/</u>

One card required for each stringer element (Section 4.3.2).

Format	Columns	Symbol	Comments
A4	1-4	STRG	Key word
	5-10		Blank
I5	11-15	MEM	Element identification number
I5	16-20	NODEI	Node i
I5	21-25	NODEJ	Node j, defines + x axis
I5	26-30	NODEK	Node k, mid-length node, optional. Default = 0.
	31-80		Blank

<u>Note:</u>

1. The mid-length node is provided to allow compatibility with the mid-side node of an adjacent TRIM element

2. The nodes must be separated by a finite distance.

One card required for each triangular membrane element (Section 4.3.4).

Format	Columns	Symbol	Comments
A4	1-4	TRIM	Key word
	5-10		Blank
I5	11-15	MEM	Element identification number
I5	16-20	NODEI	Node i, first vertex
I5	21-25	NODEJ	Node j, mid-side, between i and k, optional
I5	26-30	NODEK	Node k, second vertex, defines + x axis
I5	31-35	NODEL	Node 1, mid-side between k and m, optional
I5	36-40	NODEM	Node m, third vertex, defines + y axis
I5	41-45	NODEN	Node n, mid-side between m and i, optional
	46-80		Blank

Notes:

1. The three vertex nodes i, k, m are required, and define the triangle's x and y axes system in which its stresses and strains are calculated.

2. If any mid-side node is missing, the strains along that side will be constant, thus a triangle with only the three vertex nodes specified will have constant strain throughout. When a mid-side node is specified, the strain will be linear along that side.

3. The three vertex nodes must be separated from each other by finite distances, and must not be colinear.

One card required for each triangular plate element (Section 4.3.6).

Format	Columns	Symbol	Comments
A4	1-4	TRP2	Key word
	5-10		Blank
I5	11-15	MEM	Element identification number
I5	16-20	NODEI	Node i, first vertex
I5	21-25	NODEJ	Node j, second vertex, defines + x axis
I5	26-30	NODEK	Node k, third vertex, defines + y axis
	31-80		Blank

Notes:

1. The three vertex nodes must be separated from each other by finite distances, and must not be colinear.

<u>/ Key Word SEND/</u>

This required card ends the Group C data.

Format	Columns	Symbol	Comments
A4	1-4	SEND	Key word
	5-80		Blank

3.5 GROUP D - NODE POINT COORDINATES

This input is required for every node listed in Group B. The cards in this group can appear in any order. (Section 4.4)

If a node is a mid-side node of a TRIM or STRG element, the program will calculate the coordinates, ignoring the user's input values. Therefore, the coordinates of mid-side nodes can be omitted.

/ Key Word GRDX/

Indicates X-coordinate of a set of node points.

Format	Columns	Symbol	Comments
A4	1-4	GRDX	Key word
	5		Blank
E15.0	6-20	XG	X-coordinate (L)
12I5	21-80	NODE	The applicable nodes for the above X-coordinate are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. If the last I5 field is non-zero, the next card is assumed to be a continuation card. In this case the first 20 fields are ignored and only the applicable nodes are read. A blank I5 field ends the specification.

NOTES:

1. If a node list ends in the last field of a card, a blank card must follow.

2. The key word and X-coordinate may be repeated on a continuation card.

Indicates Y-coordinate of a set of node points.

Format	Columns	Symbol	Comments
A4	1-4	GRDY	Key word
	5		Blank
E15.0	6-20	YG	Y-coordinate (L)
12I5	21-80	NODE	The applicable nodes for the above Y-coordinate are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. If the last I5 field is non-zero, the next card is assumed to be a continuation card. In this case the first 20 fields are ignored and only the applicable nodes are read. A blank I5 field ends the specification.

NOTES:

1. If a node list ends in the last field of a card, a blank card <u>must</u> follow.

2. The key word and Y-coordinate may be repeated on a continuation card.

/ Key Word GRDZ/

Indicates Z-coordinate of a set of node points.

Format	Columns	Symbol	Comments
A4	1-4	GRDZ	Key word
	5		Blank
E15.0	6-20	ZG	Z-coordinate (L)
1215	21-80	NODE	The applicable nodes for the above Z-coordinate are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. If the last I5 field is non-zero, the next card is assumed to be a continuation card. In this case the first 20 fields are ignored and only the applicable nodes are read. A blank I5 field ends the specification.

NOTES:

1. If a node list ends in the last field of a card, a blank card must follow.

2. The key word and Z-coordinate may be repeated on a continuation card.

<u>/Key Word XSHF/</u>

This optional card indicates the shift of the origin of vehicle coordinates in the direction of the global X axis (Section 4.4.1).

Format	Columns	Symbol	Comments
A4	1-4	XSHF	Key word
	5		Blank
E15.0	6-20	XO	X-Shift (L). Default = 0
	21-80		Blank

NOTES:

This optional card indicates the shift of the origin of vehicle coordinates in the direction of the global y axis (Section 4.4.1).

Format	Columns	Symbol	Comments
A4	1-4	YSHF	Key Word
	5		Blank
E15.0	6-20	YO	Y-shift (L). Default = 0
	21-80		Blank

NOTES:

/Key Word ZSHF/

This optional card indicates the shift of the origin of vehicle coordinates in the direction of the global z axis (Section 4.4.1).

Format	Columns	Symbol	Comments
A4	1-4	ZSHF	Key word
	5		Blank
E15.0	6-20	Z0	Z-Shift (L). Default = 0.
	21-80		Blank

NOTES:

<u>/Key Word YAW/</u>

This optional card specifies the rotation of the vehicle coordinates about the vehicle z axis (Section 4.4.1).

Format	Columns	Symbol	Comments
A4	1-4	YAW	Key word
	5		Blank
E15.0	6-20	PSI	Rotation of vehicle coordinates about z axis (degrees). Default = 0.
	21-80		Blank

<u>NOTES</u>:

1. First, any vehicle coordinate shifts (XSHF, YSHF, ZSHF) will be done. Then these coordinate rotations will be performed about the vehicle's new z, y, and x axes, in that order. The z-rotation is done first, then the rotation abut the new y axis, and finally the rotation about the new x-axis.

This optional card specifies the initial rotation of the vehicle coordinates about the new vehicle y axis (Section 4.4.1).

Format	Columns	Symbol	Comments
A4	1-4	PITC	Key word
	5		Blank
15.0	6-20	THETA	Rotation of vehicle coordinates about the y axis (degrees). Default = 0.
	21-80		Blank

NOTES:

1. First, any vehicle coordinate shifts (XSHF, YSHF, ZSHF) will be done. Then these initial rotations will be performed about the vehicle's new z, y, and x axes, in that order. The z-rotation is done first, then the rotation about the new y axis, and finally the rotation about the new x-axis.

<u>/Key Word ROLL/</u>

This optional card indicates the rotation of the vehicle coordinates about the vehicle x-axis (Section 4.4.1)

Format	Columns	Symbol	Comments
A4	1-4	Roll	Keyword
	5		Blank
E15.0	6-20	· PHI	Rotation of vehicle coordinates about the x axis (degrees)
	21-80		Blank

<u>NOTES</u>:

1. First, any vehicle coordinate shifts (XSHF, YSHF, ZSHF) will be done. Then these initial rotations will be performed about the vehicle's new z,y, and x axes, in that order. The z rotation is done first, then the rotation about the new y axis, and finally the rotation about the new x-axis.

<u>/ Key Word SEND/</u>

   <u>This required card ends the Group D data.</u>

<u>Format</u>	<u>Columns</u>	<u>Symbol</u>		<u>Comments</u>
A4	1-4	SEND	Key word	
	5-80		Blank	

<u>NOTES</u>:

3.6 GROUP E - NODE POINT SINGLE AND MULTIPOINT CONSTRAINTS

Constraints are specified here. First are the SPC (single point constraints or boundary conditions). Then, in any order; MPC (multipoint constraints or dependencies), APPL (applied displacements), and ACEL (applied accelerations). (Section 4.5) A SEND card is required to end this group, even if there are no constraints.

/ Key Word SPC/

These optional cards specify the single point constraints, and must be specified before other constraints. (Section 4.5.1)

Format	Columns	Symbol	Comments
A3	1-3	SPC	Key word
	4-5		Blank
6I1	6-11	NBND	A 6-digit sequence, in order of DOF, describing a set of constraints at a node: 0 = fixed or dependent 1 = free 2 = applied displacement or acceleration will be specified
	12-20		Blank
12I5	21-80	NODE	The applicable nodes for the above boundary conditions are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. If the last I5 field is non-zero, the next card is assumed to be a continuation card. In this case the first 20 fields are ignored and only the applicable nodes are read. A blank I5 field ends the specification.

NOTES:

1. If a node list ends in the last field of a card, a blank card must follow.

2. Constraints need be input only for those nodes having either a fixed, dependent, or applied displacement. Free DOF default conditions (NBND = 111111) are set for all nodes not listed. (Section 4.5.1).

3. The key word SPC and boundary condition (NBND values) may be repeated on a continuation card.

4. A maximum of 200 different NBND words is allowed.

3.44

/ Key Word MPC, or APPL, or ACEL/

This is a delimiter card, used only if there are MPC, APPL, or ACEL inputs following the SPC cards. The key word on this card should be the same as the key word on the data card that immediately follows.

Format	Columns	Symbol	Comments
A4	1-4		Enter one of the Key words MPC, APPL, or ACEL as required.
	5-80		Blank

NOTES:

See Section 4.5.3 for the details of the multipoint constraints feature.

Format	Columns	Symbol	Comments
A4	1-3	MPC	Key word
	4-5		Blank
I5	6-10	NODED	Dependent node number
I5	11-15	NDOFD	Dependent degree of freedom
	16-20		Blank
I5	21-25	NODEA	Independent node number
I5	26-30	NDOFA	Independent degree of freedom
E15.0	31-45	COEFA	Coefficient in constraint equation
	46-50		Blank
I5	51-55	NODEB	Independent node number (if required)
I5	56-60	NDOFB	Independent degree of freedom (if required)
E15.0	61-75	COEFB	Coefficient in constraint equation (if required)
	76-80		Blank

NOTES:

1. Any number of continuation cards can be used to complete a constraint equation. Continuation cards begin in column 21, but columns 1-20 can be repeated if desired.

2. A maximum of 200 dependent nodes (NODED) and 500 coefficients (COEFA, COEFB, and UAPPL) are allowed, for the MPC and APPL inputs combined.

3. A node having inertia added in Group G, can have no more than one independent constraint coefficient for each of its dependent DOF.

4. If these MPC cards immediately follow the SPC cards, then the first MPC card must be a delimiter, containing only the key word MPC in columns 1-3, followed by the MPC data cards and then any APPL and ACEL cards.

These optional cards specify applied displacements and follow the MPC card set. See Section 4.5.2 for further details.

Format	Columns	Symbol	Comments
A4	1-4	APPL	Key word
	5		Blank
I5	6-10	NODED	Node number at which displacement is applied
I5	11-15	NDOFD	Degree of freedom at which displacement is applied
	16-30		Blank
E15.0	31-45	UAPPL	Magnitude of the applied displacement factor
	46-80		Blank

NOTES:

1. The applied displacement capability is applicable only for static analysis (key word STAT).

2. If these APPL cards immediately follow the SPC cards, then the first APPL card must be a delimiter, containing only the key word APPL in column 1-4, followed by the APPL data cards, and then any MPC and ACEL cards.

3. The complete applied displacement function is the product of the factors specified here and the time function PTME specified in Group I.

4. For every DOF given an applied displacement, an NBND = 2 must be specified on an SPC card.

5. A maximum of 200 dependent nodes (NODED) and 500 coefficients (COEFA, COEFB, and UAPPL) are allowed, for the MPC and APPL inputs combined.

3.47

<u>/Key Word ACEL/</u>

This optional card specifies the DOF which is given an applied
acceleration and identifies the associated table of acceleration versus time
given in Group I. One card for each DOF to be given an acceleration.
See Section 4.6.6.

Format	Columns	Symbol	Comments
A4	1-4	ACEL	Key word
	5		Blank
I5	6-10	NODE	Node given the applied acceleration
I5	11-15	NDOF	DOF given the applied acceleration, 1-6.
	16-20		Blank
I5	21-25	ACELID	Identifying number of the ACELT acceleration vs. time function 3.108 applied to the NODE and NDOF.
	26-80		Blank

<u>NOTES</u>:

1. Every DOF given an applied acceleration must have its NBND = 2 on an SPC
    card.

2. If these ACEL cards immediately follow the SPC cards, then the first ACEL
    card must be a delimiter, containing only the key word ACEL in columns 1-
    4, followed by the ACEL data cards, and then any MPC or APPL cards.

3. An ACELT card set is required in Group I when this ACEL card is used.

/ Key Word SEND/

This card ends the Group E data, and is always required, even if this group is not used, except on a restart run.

Format	Columns	Symbol	Comments
A4	1-4	SEND	Key word
	5-80		Blank

3.7 GROUP F - NODE POINT INITIAL DISPLACEMENTS AND VELOCITIES

Initial displacements, rotations, velocities, and angular velocities are specified with this group of data cards. All initial conditions are preset to a default value of zero, so that only non-zero values need be specified. If all initial displacements and velocities are zero, omit this group and go to p. 3.55. Any number of card sets can be used in any order.

/ Key Word IDSP/

This optional set of at least three cards specifies initial displacements and rotations; the first containing initial displacements, the second initial rotations, and the succeeding card(s) the applicable nodes.

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	IDSP	Key word
	5		Blank
E15.0	6-20	U	Initial displacements in the global X-direction (L)
E15.0	21-35	V	Initial displacements in the global Y-direction (L)
E15.0	36-50	W	Initial displacements in the global Z-direction (L)
	51-80		Blank

Card 2:

Format	Columns	Symbol	Comments
A4	1-4	IDSP	Key word, optional here.
	5		Blank
E15.0	6-20	THETAX	Initial rotation with respect to the global X-direction (radians)
E15.0	21-35	THETAY	Initial rotation with respect to the global Y-direction (radians)
E15.0	36-50	THETAZ	Initial rotation with respect to the global Z-direction (radians)
	51-80		Blank

(continued on next page)

Card(s) 3:

The third and succeeding card(s) contain the applicable nodes.

Format	Columns	Symbol	Comments
16I5	1-80	NODE	The applicable nodes for the above initial displacements and rotations are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A <u>blank</u> I5 field ends the specifications of applicable nodes

NOTES:

1. Input need not be specified for nodes that have all their initial displacements and rotations equal to zero.

2. Displacements and rotations of constrained or dependent degrees of freedom (NBND ≠ 1 on the SPC cards can be input here but will be ignored.

3. If the list of nodes ends in the last field on a card, a blank card <u>must</u> follow.

4. Any number of sets of these cards are allowed, for the case in which different parts of the structure have different initial conditions.

These optional cards specify initial velocities and angular velocities. Each set consists of at least three cards; the first containing initial velocities, the second initial angular velocities and the succeeding card(s) the applicable nodes.

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	IVEL	Key word
	5		Blank
E15.0	6-20	UDOT	Initial velocities in the global X-direction (L/T)
E15.0	21-35	VDOT	Initial velocities in the global Y-direction (L/T)
E15.0	36-50	WDOT	Initial velocities in the global Z-direction (L/T)
	51-80		Blank

Card 2:

(If Card 1 is used, this card must follow, even if blank.)

Format	Columns	Symbol	Comments
A4	1-4	IVEL	Key word, optional here
	5		Blank
E15.0	6-20	THTAXD	Initial angular velocity with respect to the global X-direction (RAD/T)
E15.0	21-35	THTAYD	Initial angular velocity with respect to the global Y-direction (RAD/T)
E15.0	36-50	THTAZD	Initial angular velocity with respect to the global Z-direction (RAD/T)
	51-80		Blank

Card(s) 3:

The third and succeeding card(s) contain the applicable nodes.

Format	Columns	Symbol	Comments
16I5	1-80	NODE	The applicable nodes for the above initial displacements and rotations are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable nodes.

NOTES:

1. Input need not be specified for nodes that have all their initial velocities equal to zero.

2. Velocities of constrained or dependent degrees of freedom (NBND ≠ 1 on the SPC cards) can be input here but will be ignored.

3. If the list of nodes ends in the last field on a card, a blank card must follow.

4. Any number of sets of these cards is allowed, for the case in which different parts of the structure have different initial conditions.

<u>/ Key Word SEND/</u>

This card ends the Group F data, and is required <u>only if</u> this group is used.

Format	Columns	Symbol	Comments
A4	1-4	SEND	Key word
	5-80		Blank

NOTES:

1. Any number of card sets can be made in <u>any</u> order as long as each set contains the proper internal order.

3.8 GROUP G - ADDED INERTIA

Node point lumping of structural or nonstructural concentrated masses and mass moments of inertia can be specified with this optional group of data cards. When there are no concentrated masses or mass moments of inertia, omit this group and go to p. 3.58.

/ Key Word CNM1 /

Each set consists of <u>at least</u> three cards; the first containing the mass and eccentricities, the second mass moments of inertia, and the succeeding card(s) the applicable nodes.

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	CNM1	Key word
	5		Blank
E15.0	6-20	MASS	Concentrated mass at the applicable node ($M = FT^2/L$)
E15.0	21-35	XO	Offset distance of the mass from the node in the global X-direction (L)
E15.0	36-50	YO	Offset distance of the mass from the node in the global Y-direction (L)
E15.0	51-65	ZO	Offset distance of the mass from the node in the global Z-direction (L)
	66-80		Blank

Card 2:

Format	Columns	Symbol	Comments
A4	1-4	CNM1	Key word, optional here
	5		Blank
E15.0	6-20	I11	Mass moment of inertia of the concentrated mass about the global X-direction ($ML^2=FLT^2$)
E15.0	21-35	I22	Mass moment of inertia of the concentrated mass about the global Y-direction ($ML^2=FLT^2$)
E15.0	36-50	I33	Mass moment of inertia of the concentrated mass about the global Z-direction ($ML^2=FLT^2$)
	51-80		Blank

Card(s) 3:

The third and succeeding card(s) contain the applicable nodes.

Format	Columns	Symbol	Comments
16I5	1-80	NODE	The applicable nodes for the above concentrated mass are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable nodes.

NOTES:

1. If the list of nodes ends in the last I5 field, a blank card must follow.

2. A node having inertia added here must have no more than one independent constraint coefficient for each of its dependent DOF, for the MPC input specified in Group E, p 3.44.

3. These items are additive. Inertia may be accumulated at a node over more than one set of cards. A negative value will subtract inertia.

4. Mass can be added or subtracted on a restart. Mass can be deleted, on a restart, by adding negative mass here.

This card ends the Group G data, and is required only if this group is used.

Format	Columns	Symbol		Comments
A4	1-4	SEND	Key word	
	5-80		Blank	

NOTES:

1. If this group is used on a restart, the STOP card follows immediately.

3.9 GROUP H - ELEMENT MATERIAL AND SECTION PROPERTIES

This required group consists of a number of card sets, each set initiated by a key word. These card sets can appear in any order, although inside each set the sequence of the cards is fixed. Data for this group can be classified as material properties or element cross-section properties. Plastic material properties are input in terms of yield stress and parameters that determine the shape of a uniaxial stress versus strain curve. These parameters are defined in Table 3.9.1 below. (Section 4.3.1) Each element <u>must</u> have its properties specified in this group.

Table 3.9.1 Parameters Determining Hardening Behavior

PLAS1	PLAS2	Type of Plasticity
0.0	0.0	Ideally plastic (no strain hardening)
E_t/E	0.0	Linear strain hardening, with E_t equal to the tangent modulus of the stress versus strain curve in the plastic range
n	$\sigma_{0.7}$	Nonlinear strain hardening with the Ramberg-Osgood form of the three parameter stress versus strain curve $$\varepsilon = \frac{\sigma}{E} + \frac{3\sigma_{0.7}}{7E}\left(\frac{\sigma}{\sigma_{0.7}}\right)^n$$

C-2

This set of at least four cards gives the material property specification for isotropic plane stress elements, TRIM and TRP2. (Section 4.3.4.4)

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	MAT1	Key word
	5		Blank
E15.0	6-20	E	Young's modulus (F/L^2)
E15.0	21-35	ANU	Poisson's ratio
	36-80		Blank

Card 2:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	DENSITY	<u>Mass</u> - density ($M/L^3 = FT^2/L^4$)
	21-50		Blank
E15.0	51-65	EPSFT	Failure strain in tension (+). If zero or blank no failure criterion is monitored in either direction
E15.0	66-80	EPSFC	Failure strain in compression (-). Use the negative sign. Ignored if EPSFT is zero or blank.

Card 3:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	PLAS1	Parameter determining plastic hardening behavior defined in Table 3.9.1 and Section 4.3.1. Default = 0.
E15.0	16-30	PLAS2	Parameter determining plastic hardening behavior defined in Table 3.9.1 and Section 4.3.1. Default = 0.
E15.0	36-50	YLDST	Yield stress
	51-80		Blank

3.59

Card (s) 4:

The fourth and succeeding card(s) contain applicable members.

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above material properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last I5 field of a card, a blank card must follow.

2. Up to twenty different sets of plane stress material properties can be specified, totalled over MAT1, MAT2, MAT3.

3. Non-zero tension failure (EPSFT) must be specified if failure is to be monitored in either tension or compression. If compression failure is desired, but tension failure is not, then use a very large strain for EPSFT.

4. The yield stress (YLDST) must not be set to zero.

/ Key Word MAT2/

This set of at least six cards specifies plane stress material properties for orthotropic material with orthotropic ideal plasticity (no strain hardening). (See Section 4.3.4.4). The axes for material properties are the orthotropic material principal directions 1, 2 in the element plane, and are assumed as a default condition to coincide with the element local coordinate axes x, y. If these directions do not coincide, 1 to x and 2 to y, an appropriate transformation angle must be input. This is discussed later in Group H with key word MBET.

This orthotropic input is only for TRIM elements, and is currently <u>not</u> operational for the TRP2 plate element.

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	MAT2	Key word
	5		Blank
E15.0	6-20	EONE	Young's modulus in the material 1-direction (F/L^2)
E15.0	21-35	ETWO	Young's modulus in the material 2-direction (F/L^2)
	36-50		Blank
E15.0	51-65	GONTO	Shear modulus for shear stresses applied in the material 1-2 directions
E15.0	66-80	VONTO	Poisson's ratio n_{12}, defined as $|e_{11}/e_{22}|$ when e_{22} is applied in a uniaxial test

Card 2:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	DENSITY	Mass density ($M/L^3 = FT^2/L^4$)
	21-50		Blank
E15.0	51-65	EPSFT1	Failure strain in tension (+) in the material 1 direction. If zero or blank no failure criterion is monitored in either direction.
E15.0	66-80	EPSFC1	Failure strain in compression (-) in the material 1 direction. The negative sign is required. Ignored if EPSFT1 is zero or blank.

Card 3:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	EPSFT2	Failure strain in tension (+) in the material 2 direction. Ignored if EPSFT1 is zero or blank.
E15.0	21-35	EPSFC2	Failure strain in compression (-) in the material 2 direction. The negative sign is required. Ignored if EPSFT1 is zero or blank.
	36-80		Blank

Card 4:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	SIGOX	Yield stress in material 1-direction (F/L^2)
E15.0	21-35	SIGOY	Yield stress in material 2-direction (F/L^2)
E15.0	36-50	SIGOZ	Yield stress in thickness-direction (F/L^2)
E15.0	51-65	SIGXY	Yield stress for shear applied in the material 1-2 directions
	66-80		Blank

Card 5 (required)

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	16-20	PLAS1	Set equal to zero or blank
E15.0	21-35	PLAS2	Set equal to zero or blank
	36-80		Blank

Card(s) 6:

The sixth and succeeding card(s) contain applicable members.

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above material properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last I5 field of a card, a blank card must follow.

2. Up to twenty different sets of plane stress material properties can be specified, totalled over MAT1, MAT2, MAT3.

3. Non-zero tension failure (EPSFT1) must be specified if failure is to be monitored in either tension or compression in either the material 1- or 2- directions. If no failure is desired in tension in the material 1 direction, but any of the other three failures are desired, then use a very large strain for EPSFT1.

4. EONE and ETWO can not be set exactly equal to zero if pure shear is being simulated (i.e. GONTO non zero). In this case set EONE and ETWO to a small value.

5. All values of the yield stress (card 4) must be non zero.

This set of at least seven cards specifies plane stress material properties for orthotropic materials with orthotropic strain hardening (See Section 4.3.4.4).

The axes for material properties are the orthotropic material principal directions 1,2 in the element plane, and are assumed as a default condition to coincide with the element local coordinate axes, x, y. If these directions do not coincide, 1 to x and 2 to y, an appropriate transformation angle must be input. This is discussed later in Group H in the set with key word MBET.

This orthotropic input is only for TRIM elements, and is currently _not_ operational for the TRP2 plate element.

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	MAT3	Key word
	5		Blank
E15.0	6-20	EONE	Young's modulus in the material 1-direction (F/L^2)
E15.0	21-35	ETWO	Young's modulus in the material 2-direction (F/L^2)
	36-50		Blank
E15.0	51-65	GONTO	Shear modulus for shear stresses applied in the material 1-2 directions
E15.0	66-80	VONTO	Poisson's ratio n_{12}, defined as $|e_{11}/e_{22}|$ when e_{22} is applied in a uniaxial test

Card 2:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	DENSITY	Mass density ($M/L^3 = FT^2/L^4$)
	21-50		Blank
E15.0	51-65	EPSFT1	Failure strain in tension (+) in the material 1 direction. If zero or blank no failure criterion is monitored in in either direction.
E15.0	66-80	EPSFC1	Failure strain in compression (-) in the material 1 direction. Negative sign required. Ignored if EPSFT1 is zero or blank.

Card 3:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	EPSFT2	Failure strain in tension (+) in the material 2 direction. Ignored if EPSFT1 is zero or blank.
E15.0	21-35	EPSFC2	Failure strain in compression (-) in the material 2 direction. Negative sign required. Ignored if EPSFT1 is zero or blank.
	36-80		Blank

Card 4:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	SIGOX	Yield stress in material 1-direction (F/L^2)
E15.0	21-35	SIGOY	Yield stress in material 2-direction (F/L^2)
E15.0	36-50	SIGOZ	Yield stress in thickness-direction (F/L^2)
E15.0	51-65	SIGXY	Shear yield stress in principal 1-2 plane (F/L^2)
	65-80		Blank

Card 5:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	PLAS1X	Same as PLAS1 (Table 3.9.1) but for material 1-direction only, nonzero
E15.0	21-35	PLAS2X	Same as PLAS2 (Table 3.9.1) but for material 1-direction only.
E15.0	36-50	PLAS1Y	Same as PLAS1 (Table 3.9.1) but for material 2-direction only, nonzero
E15.0	51-65	PLAS2Y	Same as PLAS2 (Table 3.9.1) but for material 2-direction only.
E15.0	66-80	PLAS1XY	Same as PLAS1 (Table 3.9.1) but for shear applied in the material 1-2 directions only, nonzero

Card 6:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	PLAS2XY	Same as PLAS2 (Table 3.9.1) but for shear applied in material 1-2 directions only.
	16-80		Blank

Card(s) 7:

The seventh and succeeding card(s) contain applicable members.

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above material properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last I5 field of a card, a blank card must follow.

2. No component of the MAT3 stress-strain curves may be ideally plastic, i.e, PLAS1X, PLAS1Y, PLAS1XY can not be specified as zero. If any one is zero all must be zero and the MAT2 card set should be used.

3. Up to twenty different sets of plane stress material properties can be specified, totalled over MAT1, MAT2, MAT3.

4. Non-zero tension failure (EPSFT1) must be specified if failure is to be monitored in either tension or compression in either the material 1 or 2 directions. If no failure is desired in tension in the material 1 direction, but any of the other three failures are desired, then use a very large strain for EPSFT1.

5. EONE and ETWO can not be set exactly equal to zero if pure shear is being simulated. In this case set EONE and ETWO to a small value.

6. All values of the yield stress (Card 4) must be non zero.

<u>/ Key Word THIK/</u>

This set of at least two cards specifies the element thickness necessary for two dimensional membrane and plate elements, and number of layers through the thickness for plate elements. (Section 4.3.4.5) Each TRIM or TRP2 element <u>must</u> be represented here.

<u>Card 1</u>:

Format	Columns	Symbol	Comments
A4	1-4	THIK	Key word
	5		Blank
E15.0	6-20	THICK	Thickness of membrane or bending elements (L)
E15.0	21-35	NLAY	Number of layers through the element thickness. This input is necessary with plate elements. It is ignored for membrane elements. NLAY must be an even number from 2 to 8
	36-80		Blank

<u>Card(s) 2</u>:

The second and succeeding card(s) contain applicable members.

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above material properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A <u>blank</u> I5 field ends the specification of applicable members for each thickness.

<u>NOTES</u>:

1. If a member list ends in the last I5 field on a card, a blank card <u>must</u> follow.

2. Up to 100 different sets of these cards can be specified.

3. The number of integration points through the thickness is NLAY +1, (from 3 to 9) including the outer surfaces and the mid-surface.

<u>/ Key Word MBET/</u>

This optional set of at least two cards specifies the orientation of axes of material anisotropy for TRIM elements. Not required for any element when the element's material 1 axis coincide with its local x axis. (Section 4.3.4.6)

<u>Card 1:</u>

<u>Format</u>	<u>Columns</u>	<u>Symbol</u>	<u>Comments</u>
A4	1-4	MBET	Key word
	5		Blank
E15.0	6-20	BETF	Angle in <u>degrees</u> between element local X-axis and principal 1-axis for material orthotropy. (Degrees)
	21-80		Blank

<u>Card(s) 2:</u>

The second and succeeding card(s) contain applicable members.

<u>Format</u>	<u>Columns</u>	<u>Symbol</u>	<u>Comments</u>
16I5	1-80	MEM	The applicable members for the above angles are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A <u>blank</u> I5 field ends the specification of applicable members.

<u>NOTES:</u>

1. If a member list ends in the last I5 field on a card, a blank card <u>must</u> follow.

2. An orientation angle can be specified for any TRIM element in a model. If an element is not listed here, its material orientation angle BETF will default to zero.

This set of three or more cards specifies material and section properties for stringer elements (axial force elements). Necessary only when stringer (STRG) elements are used. (Section 4.3.2.4) Each STRG element <u>must</u> be represented here.

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	MSTG	Key word
	5		Blank
E15.0	6-20	E	Young's modulus (F/L^2)
E15.0	21-35	A	Cross sectional area (L^2)
E15.0	36-50	PLAS1	Parameter determining plastic hardening behavior defined in Table 3.9.1 and Section 4.3.1
E15.0	51-65	PLAS2	Parameter determining plastic hardening behavior defined in Table 3.9.1 and Section 4.3.1
E15.0	66-80	YLDST	Yield stress (F/L^2)

Card 2:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	DENSITY	Mass-density ($M/L^3 = FT^2/L^4$)
	21-35		Blank
E15.0	36-50	SEPST	Failure strain in tension (+). If zero or blank no failure criterion is monitored in tension or compression.
E15.0	51-65	SEPSC	Failure strain in compression (-). Negative sign required. Ignored if SEPST is zero or blank.
	66-80		Blank

Card(s) 3:

The third and succeeding card(s) contain applicable members.

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above material properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last I5 field on a card, a blank card must follow.

2. Up to 200 different sets of MSTG and beam section geometry (CSEC, etc.) card sets can be specified.

3. Non-zero tension failure (SEPST) must be specified if failure is to be monitored in either tension or compression. If no tension failure is desired, but compression failure is desired, use a very large strain for SEPST.

/ Key Word MBM/

This set consists of at least three cards. It specifies material properties for beam elements and is necessary _only_ when beam elements are used. (Section 4.3.3)

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	MBM	Key word
	5		Blank
E15.0	6-20	E	Young's modulus (F/L^2)
E15.0	21-35	ANU	Poisson's ratio
E15.0	36-50	PLAS1	Parameter determining plastic hardening behavior defined in Section 4.3 and Table 3.9.1.
E15.0	51-65	PLAS2	Parameter determining plastic hardening behavior defined in Section 4.3 and Table 3.9.1.
E15.0	66-80	YLDST	Yield stress (F/L^2)

Card 2:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	DENSITY	Mass-density $(M/L^3 = FT^2/L^4)$
	21-35		Blank
E15.0	36-50	BEPST	Failure strain in tension (+). If zero or blank no failure criterion is monitored in tension or compression.
E15.0	51-65	BEPSC	Failure strain in compression (-). Negative sign required. Ignored if BEPST is zero or blank.
	66-80		Blank

(continued on next page)

The succeeding card(s) contain the applicable members.

Card(s) 3: Applicable Members:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above beam material properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. Up to 50 different MBM sets can be specified.

2. If a member list ends in the last I5 field on a card, a blank card must follow.

3. Non-zero tension failure (BEPST) must be specified if failure is to be monitored in either tension or compression. If tension failure is not desired, but compression failure is desired, use a very large strain for BEPST.

4. Poisson's ratio on card 1 is used only to calculate the elastic shear modulus $G = E/2(1 + \nu)$.

This set of at least four cards specifies the geometric properties for the special beam cross sections having certain fixed shapes. The listed key words refer to various beam cross sections defined in Section 4.3.3.5 and Table 4.3.1.

The key words are: CSEC, HCIR, HREC, HSEC, ISEC, LSEC, LSEG, SCIR, SREC, TSEC, TWD, ZSCR, ZSEC.

The first two cards have the same format for all the special sections, and will be described here only once.

Card 1:

Format	Columns	Symbol	Comments
A4	1-4		Appropriate special beam section key word from above list.
	5		Blank
E15.0	6-20	A	Cross sectional area (L^2)
E15.0	21-35	IYY	Moment of inertia about centroidal y-axis (L^4)
E15.0	36-50	IZZ	Moment of inertia about centroidal z-axis (L^4)
E15.0	51-65	IYZ	Product of inertia about centroidal y-z axes (L^4)
E15.0	66-80	J	Torsional rigidity (L^4)

NOTES:

1. A, IYY, IZZ and IYZ will be internally calculated if <u>all</u> four are set to zero or blank in the Input data. However, J must always be input, except for the TWD section, where it is ignored.

Card 2:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	YO	Nodal eccentricity in the reference y-direction, from the origin to the node. (L)
E15.0	21-35	ZO	Nodal eccentricity in the reference z-direction, from the origin of axes to the node. (L)
E15.0	36-50	BETA	Orientation angle of beam local axes measured from new reference axes. (Degrees)
E15.0	51-65	YA	Offset of beam axes in the y direction measured from the new reference axes. (L)
E15.0	66-80	ZA	Offset of beam axes in the z direction measured from the new reference axis. (L)

NOTES:

1. The last three entries (BETA, YA, ZA) on this card are used only when the beam cross-section is to be moved relative to the reference axes. Used for combining beam elements, or rotating a beam cross-section from its originally specified position. If BETA, YA, and ZA are all zero or blank, then the reference axes coincide with the local cross-section axes, defined in Table 4.3.1. (See Section 4.3.3.5)

2. These two cards are required for each special beam section. The remaining cards in each set depend on the cross section being specified. (See Table 4.3.1. These remaining cards are described for each section on succeeding pages.

Card 3: CSEC - Channel Section

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	A1	Dimension of upper flange (L)
E15.0	21-35	A2	Dimension of web (L)
E15.0	36-50	A3	Dimension of lower flange (L)
	51-80		Blank

Card 4:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	T1	Thickness of upper flange (L)
E15.0	21-35	T2	Thickness of web (L)
E15.0	36-50	T3	Thickness of lower flange (L)
	51-80		Blank

The fifth and succeeding card(s) contain applicable members.

Card(s) 5:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above section properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last field on a card, a blank card must follow.

2. Up to 200 different beam cross section card sets can be specified, including the MSTG stringer properties.

Card 3:   HCIR - Hollow Circle

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	R	Outer radius (L)
E15.0	21-35	T	Thickness (L)
	36-80		Blank

The fourth and succeeding card(s) contain applicable members.

Card(s) 4:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above section properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1.   If a member list ends in the last field on a card, a blank card must follow.

2.   Up to 200 different beam cross section card sets can be specified, including the MSTG stringer properties.

Card 3:   HREC - Hollow Rectangle

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	A1	Section width (L)
E15.0	21-35	A2	Section depth (L)
E15.0	36-50	T1	Thickness of upper and lower flanges (L)
E15.0	51-65	T2	Thickness of vertical webs (L)
	66-80		Blank

The fourth and succeeding card(s) contain applicable members.

Card(s) 4:

16I5	1-80	MEM	The applicable members for the above section properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1.   If a member list ends in the last field on a card, a blank card must follow.

2.   Up to 200 different beam cross section card sets can be specified, including the MSTG stringer properties.

Card 3:   HSEC - Hat Section

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	B	Dimension of flanges (L)
E15.0	21-35	B1	Dimension of flange tabs (L)
E15.0	36-50	H	Dimension of web (L)
E15.0	51-65	T	Uniform Section thickness (L)
	66-80		Blank

The fourth and succeeding card(s) contain applicable members.

Card(s) 4:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above section properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last field on a card, a blank card must follow.

2. Up to 200 different beam cross section card sets can be specified, including the MSTG stringer properties.

Card 3: ISEC - I-Section

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	A1	Dimension of upper flange (L)
E15.0	21-35	A2	Dimension of web (L)
E15.0	36-50	A3	Dimension of lower flange (L)
	51-80		Blank

Card 4:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	T1	Thickness of upper flange (L)
E15.0	21-35	T2	Thickness of web (L)
E15.0	36-50	T3	Thickness of lower flange (L)
	51-80		Blank

The fifth and succeeding card(s) contain applicable members.

Card(s) 5:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above section properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last field on a card, a blank card must follow.

2. Up to 200 different beam cross section card sets can be specified, including the MSTG stringer properties.

Card 3: LSEC - L-Section

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	A1	Width of flange (L)
E15.0	21-35	A2	Depth of web (L)
E15.0	36-50	T1	Thickness of flange (L)
E15.0	51-65	T2	Thickness of web (L)
	66-80		Blank

The fourth and succeeding card(s) contain applicable members.

Card(s) 4:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above section properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A _blank_ I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last field on a card, a blank card _must_ follow.

2. Up to 200 different beam cross section card sets can be specified, including the MSTG stringer properties.

Card 3:   LSEG - Line Segment (Thin Rectangle)

Format	Columns	Symbol	Comments
A4	1-4	LSEG	Key word, optional here
	5-10		Blank
E15.0	11-25	Y1	Y coordinate of start point (L)
E15.0	26-40	Z1	Z coordinate of start point (L)
E15.0	41-55	Y2	Y coordinate of end point (L)
E15.0	56-70	Z2	Z coordinate of end point (L)
	71-80		Blank

Card 4: This card must follow its associated card 3.

Format	Columns	Symbol	Comments
A4	1-4	LSEG	Key word, optional here
	5-10		Blank
E15.0	11-25	T	Segment thickness (L)
E15.0	26-40	NINT	Number of Gauss integration points in the segment. From 2 to 8 are allowed.
	41-80		Blank

NOTES:

1. This element has no bending stiffness about the axis through the starting and ending points.

The succeeding card(s) contain applicable members.

Card(s) 5:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above section properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last field on a card, a blank card must follow.

2. Up to 200 different card sets of beam cross-sections can be specified, including the MSTG stringer properties.

See p. 4.54.

Card 3: SCIR - Solid Circle

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	R	Section radius (L)
	21-80		Blank

The fourth and succeeding card(s) contain applicable members.

Card(s) 4:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above section properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last field on a card, a blank card must follow.

2. Up to 200 different beam cross section card sets can be specified, including the MSTG stringer properties.

Card 3:   SREC - Solid rectangle

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	A	Section width (L)
E15.0	21-35	B	Section depth (L)
	36-80		Blank

The fourth and succeeding card(s) contain applicable members.

Card(s) 4:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above section properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1.   If a member list ends in the last field on a card, a blank card must follow.

2.   Up to 200 different beam cross section card sets can be specified, including the MSTG stringer properties.

See p. 4.56.

Card 3: TSEC - T-Section

	1-5		Blank
E15.0	6-20	A1	Width of flange (L)
E15.0	21-35	A2	Depth of web (L)
E15.0	36-50	T1	Thickness of flange (L)
E15.0	51-65	T2	Thickness of web (L)
	66-80		Blank

The fourth and succeeding card(s) contain applicable members.

Card(s) 4:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above section properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last field on a card, a blank card must follow.

2. Up to 200 different beam cross section card sets can be specified, including the MSTG stringer properties.

Card 3:   TWD - Two Dimensional Section

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	W	Section width (perpendicular to the plane of motion) (L)
E15.0	21-35	T	Section depth or thickness (L)
E15.0	36-50	NLAY	Defines the number of integration points through the section depth. NLAY is the number of layers through the thickness in a Simpson's rule integration. NLAY must be an even number ≤ 58
	51-80		Blank

The fourth and succeeding card(s) contain applicable members.

Card(s) 4:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above section properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. This beam calculates no bending stiffness in the local y direction, nor torsional stiffness. That is, the internal calculations will define zero values for IZZ, IYZ, and J. Finite values for those parameters can be specified in the first card of this set, but these values will be main-tained only in the elastic range. As soon as any integration point becomes plastic at either node, the internal calculations will take over, and the zero values will be used.

2. If a member list ends in the last field on a card, a blank card must follow.

3. Up to 200 different beam cross section card sets can be specified, including the MSTG stringer properties.

Card 3: ZSCR - Reversed Z-Section

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	A1	Dimension of upper flange (L)
E15.0	21-35	A2	Dimension of web (L)
E15.0	36-50	A3	Dimension of lower flange (L)
	51-80		Blank

Card 4:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	T1	Thickness of upper flange (L)
E15.0	21-35	T2	Thickness of web (L)
E15.0	36-50	T3	Thickness of lower flange (L)
	51-80		Blank

The fifth and succeeding card(s) contain applicable members.

Card(s) 5:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above section properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:
1. If a member list ends in the last field on a card, a blank card must follow.

2. Up to 200 different beam cross section card sets can be specified, including the MSTG stringer properties.

3. The upper and lower flanges must be equal; $A_1 = A_3$, $T_1 = T_3$.

Card 3: ZSEC - Z-Section

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	A1	Dimension of upper flange (L)
E15.0	21-35	A2	Dimension of web (L)
E15.0	36-50	A3	Dimension of lower flange (L)
	51-80		Blank

Card 4:

Format	Columns	Symbol	Comments
	1-5		Blank
E15.0	6-20	T1	Thickness of upper flange (L)
E15.0	21-35	T2	Thickness of web (L)
E15.0	36-50	T3	Thickness of lower flange (L)
	51-80		Blank

The fifth and succeeding card(s) contain applicable members.

Card(s) 5:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above section properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last field on a card, a blank card must follow.

2. Up to 200 different beam cross section card sets can be specified, including the MSTG stringer properties.

3. The upper and lower flanges must be equal; $A_1 = A_3$, $T_1 = T_3$

3.89

This set of at least four cards specifies nonlinear spring properties for spring elements and is necessary only when spring elements are used. Input is in the form of force versus elongation or moment versus rotation tables. (Section 4.3.5 p. 4.81).

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	PSPR	Key word
	5		Blank
E15.0	6-20	NSPRG	The number of points in the table, from 2 to 20.
E15.0	21-35	USPRG	Unloading slope. If zero or blank, unloading follows loading curve (no energy dissipation). If nonzero, unloading follows along the unloading slope. (F/L or FL/L)
E15.0	36-50	FAILT	Tension failure elongation or rotation (+). If zero or blank no failure criterion is monitored in either direction. (L or radians)
E15.0	51-65	FAILC	Compression failure elongation or rotation (-). Use the negative sign. (L or radians)
E15.0	66-80	DMASS	A characteristic mass used to calculate damping. If DMASS is blank or zero there is no damping. (M = FT^2/L) See p. 4.86. (Section 4.3.5.4) Note 3.

NOTES:

1. DMASS defines a special type of nonlinear critical damping that was designed to be used with springs acting as gap elements, to prevent rapid oscillations at the contact point. See p. 4.88.

2. Non-zero tension failure elongation (FAILT) must be specified if failure is to be monitored in either tension or compression. If tension failure is not desired, but compression failure is desired, use a very large force or moment for FAILT.

3. Damping is not operational with the explicit time integrators ADAM and CDIF.

Cards 2:

These cards specify spring force and elongation in pairs, one pair on each
card. There should be a total of NSPRG cards specified, i.e, one for each
point in the table of force versus elongation (or moment vs rotation). A
minimum of two such cards is required, and a maximum of 20 are allowed.

The data must be input in order of increasing elongation, i.e, from largest
negative elongation, through zero, to largest positive elongation. Zero
elongation must be included in this table (but zero force is not required).
There are no restrictions on the variation of forces.

Format	Columns	Symbol	Comments
A4	1-4	PSPR	Key word, optional here
	5		Blank
E15.0	6-20	FORCE	Spring force (F) or moment (FL)
E15.0	21-35	ELONG	Spring elongation (L) or rotation (Radians)
	36-80		Blank

The succeeding card(s) contain the applicable members.

Card(s) 3:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above spring table properties are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last field on a card, a blank card must follow.

2. A total of 60 PSPR and PGRD tables are allowed, with a maximum of 20 points per table.

This set of at least seven cards specifies the properties of all the ground contact springs (GRDS) that will contact a single specified contact plane. One set is required for each contact plane, whenever any GRDS elements are used. (See Section 4.3.7.4 p. 4.108).

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	PGRD	Key word, required here.
	5		Blank
E15.0	6-20	NSPRG	Number of force-displacement points, from 2. to 20.
E15.0	21-35	FRICT	Coefficient of sliding friction. Note 1.
E15.0	36-50	EPSC	Contact tolerance. Default = Infinity. (L)
E15.0	51-65	EPSR	Rebound tolerance. Default = 0. (L)
E15.0	66-80	DMASS	Characteristic mass used to calculate damping force. Note 2. $(M = FT^2/L)$

NOTES:

1. There is no static friction. When the sliding velocity of a node along the contact plane is zero, or the STAT time integrator is used in Group A, the friction force is zero.

2. DMASS defines a "damping mass" for a special type of critical damping designed to be used with gap springs, to prevent rapid oscillations at the contact point. Damping is not operational for the explicit time integrators ADAM and CDIF.

Cards 2:

This subset of three cards specifies the three points that define a contact plane, and are required for every PGRD input set. Each card contains the global coordinates of one such point. The three cards have identical formats, as follows (Section 4.3.7.4):

Format	Columns	Symbol	Comments
A4	1-4	PGRD	Key word, optional here.
	5		Blank
E15.0	6-20	COORDX	Global X coordinate. (L)
E15.0	21-35	COORDY	Global Y coordinate. (L)
E15.0	36-50	COORDZ	Global Z coordinate. (L)
	51-80		Blank

NOTES:

1. The three points define a local X', Y', Z' coordinate system as follows: the local X' axis lies in the plane, from the first point towards the second point. The local Y' axis lies in the plane, is perpendicular to the local X' axis and passes through the third point. The local Z' axis is perpendicular to the X', Y' plane in the positive direction as defined by the right-hand rule. The local Z' axis must be on the same side of the X', Y' plane as the contact nodes. In other words, all the contact nodes must be on the same side of the contact plane X', Y' as the Z' axis. This direction of the Z' axis is determined by the sequence of the three nodes in the input stream, and the user should be careful to insure the proper sequence. See Fig. 4.3.11, p. 4.109.

Cards 3:

This subset of at least two cards contains the number of cards equal to the value of NSPRG on card 1, that is, one card for each data point in the force-displacement curve for the contact springs to the contact surface defined by cards 2. A maximum of 20 cards is permitted. The format of each card is identical, as follows (Section 4.3.7.4):

Format	Columns	Symbol	Comments
A4	1-4	PGRD	Key word, optional here.
	5		Blank
E15.0	6-20	FORCE	Contact force. Use a positive value. Note 1.(F)
E15.0	21-35	ELONG	Contact displacement. Use a positive value. Note 1 (L)
	36-80		Blank

NOTES :

1. These are compression forces and displacements. They will be converted to negative values internally.

2. The ELONG values must increase monotonically from zero with each succeeding card, but the FORCE values can vary without restriction. The curve is extrapolated to infinity from the last input point using the last slope.

The remaining card(s) of this set contain(s) the identification of the GRDS elements for which the above data applies.

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The list of applicable members in fields of 5 (I5 format). The short form notation and any number of cards are allowed. A <u>blank</u> I5 field ends this list.

NOTES:

1. If this list ends in the last field on a card, a blank card <u>must</u> follow.

2. A total of 60 PSPR and PGRD tables are allowed, with a maximum of 20 points per table.

<u>/ Key Word SEND/</u>

This required card ends the Group H data.

Format	Columns	Symbol		Comments
A4	1-4	SEND	Key word	
	5-80		Blank	

<u>NOTES:</u>

3.10 Group I - Applied loading

This optional groups specifies external loads and their associated time functions. These include; point, line, and area loads, time functions for these loads, and gravity loads which have a built-in step-function in time. Also specified here are applied acceleration versus time functions.

The input card sets may appear in any order but within each set, the order is fixed. Section 4.6 describes the applied load capability.

If there are no applied loads or accelerations go to the last card (STOP or END), p. 3.111.

This optional set of at least two cards specifies the concentrated nodal forces and moments. These loads remain attached to their nodes, and maintain their initial directional orientation relative to the global X, Y, Z axes. (Section 4.6.1)

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	CONC	Key word
	5		Blank
I5	6-10	NODE	Node number
E15.0	11-25	FX	Force factor in global X-direction (F)
E15.0	26-40	FY	Force factor in global Y-direction (F)
E15.0	41-55	FZ	Force factor in global Z-direction (F)
	56-80		Blank

Card 2:

Format	Columns	Symbol	Comments
A4	1-4	CONC	Key word, optional here
	5-10		Blank
E15.0	11-25	MX	Moment factor in global X-direction (FL)
E15.0	26-40	MY	Moment factor in global Y-direction (FL)
E15.0	41-55	MZ	Moment factor in global Z-direction (FL)
	56-80		Blank

NOTES:

1. These load factors are multiplied by one of the time functions PTME (p.3.102), PTM2 (p. 3.103, or PTM3 (p. 3.105) to determine the actual loads.

This optional set of at least two cards specifies the linearly varying
distributed line load on a beam element. These loads remain perpendicular to
the beam length axis. (Section 4.6.2)

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	BMLO	Key word
	5-10		Blank
E15.0	11-25	PYI	Force factor per unit length in the <u>local</u> Y direction of the beam at node i (FL)
E15.0	26-40	PYJ	Same as above at node j
E15.0	41-55	PZI	Force factor per unit length in the local Z direction of the beam at node i (FL)
E15.0	56-70	PZJ	Same as above at node j
	71-80		Blank

The second and succeeding card(s) contain applicable members.

Card 2:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above load specifications are specified in fields of 5 (I5 format). The short form notation and any number of continuous cards may be used. A <u>blank</u> I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last field on a card, a blank card <u>must</u> follow.

2. Nodes i and j are the first two nodes defined in Group C for each beam element,
    p. 3.32

3. These load factors are multiplied by one of the time functions PTME (p. 3.102),
    PTM2 (p. 3.103), or PTM3 (p. 3.105) to determine the actual loads.

4. A maximum of 100 BMLO sets are allowed.

This optional set of at least four cards specifies the linearly varying surface loads for triangular bending (plate) elements (TRP2). These loads remain parallel to the plate coordinate axes (see Section 4.6.3)

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	SURF	Key word
	5-10		Blank
E15.0	11-25	PXI	Value of distributed surface load factor (F/L^2) in local element x-direction at vertex node i of the element
E15.0	26-40	PYI	Value of distributed surface load factor (F/L^2) in local element y-direction at vertex node i of the element
E15.0	41-55	PZI	Value of distributed surface load factor (F/L^2) in local z-direction (normal to plate) at vertex node i of the element
	56-80		Blank

Card 2:

Format	Columns	Symbol	Comments
A4	1-4	SURF	Key word, optional here
	5-10		Blank
E15.0	11-25	PXJ	Value of distributed surface load factor (F/L^2) in local element x-direction at vertex node j of the element
E15.0	26-40	PYJ	Value of distributed surface load factor (F/L^2) in local element y-direction at vertex node j of the element
E15.0	41-55	PZJ	Value of distributed surface load factor (F/L^2) in local z-direction (normal to plate) at vertex node j of the element
	56-80		Blank

Card 3:

Format	Columns	Symbol	Comments
A4	1-4	SURF	Key word, optional here
	5-10		Blank
E15.0	11-25	PXK	Value of distributed surface load factor (F/L^2) in local element x-direction at vertex node k of the element
E15.0	26-40	PYK	Value of distributed surface load factor (F/L^2) in local element y-direction at vertex node k of the element
E15.0	41-55	PZK	Value of distributed surface load factor (F/L^2) in local (normal to plate) z-direction at vertex node k of the element
	56-80		Blank

The fourth and succeeding card(s) contain applicable members.

Card(s) 4:

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable members for the above load specification are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

NOTES:

1. If a member list ends in the last field on a card, a blank card must follow.

2. Nodes i, j, k are three nodes, in sequence, defined in Group C for each TRP2 element, p. 3.32.

3. These load factors are multiplied by one of the time functions PTME (p. 3.102), PTM2 (p. 3.103), or PTM3 (p. 3.105) to determine the actual load.

4. A maximum of 100 SURF sets are allowed.

This set of at least two cards specifies the common time function for the applied loads CONC, BMLO, and SURF and displacements APPL in tabular form. This PTME set is required the with APPL input. Each card contains a pair of values representing the value of the function at a given point in time. The function is composed of linear segments connecting these points. Up to 50 points can be specified. These cards need not be in order and can appear any place in input Group I. The TIME = 0 input must be included. This function will assume that the value specified for the largest time is held constant for all subsequent time. (Section 4.6.4).

Format	Columns	Symbol	Comments
A4	1-4	PTME	Key word
	5-10		Blank
E15.0	11-25	FORCE	Value of function at time t (dimensionless)
E15.0	26-40	TIME	Time for the above value (T)
	41-80		Blank

NOTES:

1. The instantaneous values of the applied loads and the applied displacements are determined as the product of the load distribution factors (CONC, BMLO, SURF) or the applied displacement factors (APPL), multiplied by the time function (PTME).

2. This PTME time function is also used in the static cases to vary the applied load or displacements. This is the only time function that can be used with the applied displacements (APPL).

3. When PTM2 or PTM3 time functions are specified for a particular load component at a particular node, they replace the PTME function for that component. If no PTME function is specified, all loads will be zero except for those having PTM2 or PTM3 time functions assigned to them.

4. A combined total of 36 PTME, PTM2, PTM3, and ACELT sets are allowed, with a maximum of 50 time values per set.

This optional set of at least four cards describes a time function to be applied at specified nodes to all six load components associated with the CONC, BMLO, and SURF inputs. Each PTM2 function is input as a table of non-dimensional load factors versus time, as follows:

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	PTM2 •	Key word
	5		Blank
I5	6-10	NPT	Number of points in the table, from 2 to 50.
	11-80		Blank

This is followed by the number of cards equal to the NPT value given in card 1, each card containing one point on the load factor versus time curve. The first card must contain the initial time, t=0, and the time must increase monotonically with the following cards.

Cards 2:

Format	Columns	Symbol	Comments
A4	1-4	PTM2	Key word, optional here.
	5-10		Blank
E15.0	11-25	FORCE	Value of load facotr (dimensionless)
E15.0	26-40	TIME	Time for the above value (T).
	41-80		Blank

The succeeding card(s) contain the applicable nodes.

Card(s) 3:

Format	Columns	Symbol	Comments
16I5	1-80	NODE	Applicable nodes for the above time function are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends this list.

NOTES:

1. The instantaneous values of the applied load components at a node are determined as the product of the spatial load factors (CONC, BMLO, and SURF) and a time function (PTME, PTM2, or PTM3).

2. Only one time function can be used for load components at a node. Therefore, if more than one of the PTM2 or PTM3 functions are assigned to the same node, only the last of these functions appearing in the input sequence will be used. Furthermore, when either a PTM2 or a PTM3 function is specified for a particular load component, it replaces the PTME function (p. 3.102) for that component, even if the PTME appears later in the input stream.

3. A maximum of 36 time functions are allowed for the combined total of PTME, PTM2, PTM3, and ACELT input card sets. There is no limit to the number of applicable nodes.

4. The PTM2 time function cannot be used with the APPL applied displacement input.

5. If the node list ends in the last field of a card, a blank card must follow.

This optional set of at least four cards describes a time function to be applied at specified components of specified nodes. These load components are associated with the CONC inputs. Each PTM3 function is input as a table of non-dimensional load factors versus time, as follows:

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	PTM3	Key word
	5		Blank
I5	6-10	NPT	Number of points in the table, from 2 to 50.
E15.0	11-25	NDOF	The specific globally-oriented load components at nodes to be specified later. 1 = Force in global X direction 2 = Force in global Y direction 3 = Force in global Z direction 4 = Moment about global X axis 5 = Moment about global Y axis 6 = Moment about global Z axis
	26-80		Blank

This is followed by the number of cards equal to the NPT value given in card 1, each card containing one point on the load factor versus time curve. The first card must contain the initial time, t=0, and the time must increase monotonically with the following cards.

Card(s) 2:

Format	Columns	Symbol	Comments
A4	1-4	PTM3	Key word, optional here.
	6-10		Blank
E15.0	11-25	FORCE	Value of load factor (dimensionless).
E15.0	26-40	TIME	Time for the above value (T).
	41-80		Blank

The succeeding card(s) contain the applicable nodes.

Card(s) 3:

Format	Columns	Symbol	Comments
15I5	1-80	NODE	Applicable nodes for the above time function are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends this list.

NOTES:

1. The instantaneous values of the applied load components at a node are determined as the product of the spatial load factors (CONC, BMLO, and SURF) and a time function (PTME, PTM2, or PTM3.

2. Only one time function can be used for load components at a node. Therefore, if more than one of the PTM2 or PTM3 functions are assigned to the same node, only the last of these functions appearing in the input sequence will be used. Furthermore, when either a PTM2 or a PTM3 function is specified for a particular load component, it replaces the PTME function (p. 3.102) for that component, even if the PTME appears later in the input stream.

3. A maximum of 36 time functions are allowed for the combined total of PTME, PTM2, PTM3, and ACELT input card sets. There is no limit to the number of applicable nodes.

4. The PTM3 time function cannot be used with the APPL applied displacement input.

5. If the node list ends in the last field of a card, a blank card must follow.

6. The PTM3 time function is not generally recommended for use with the BMLO or SURF loads, but its use is possible for small rotations. (See Section 4.6.4)

3.106

This optional card causes the imposition at every node of gravity forces (weight) in the direction of one of the global axes, with magnitude equal to the nodal mass times the specified gravitational acceleration. For the elements having distributed mass (BEAM, STRG, TRIM, TRP2), the gravity forces are apportioned to each elements's nodes using consistent load vectors derived from the element's shape function. The total weight resultant remains at the element's center of gravity. See Section 4.6.5.

Format	Columns	Symbol	Comments
A4	1-4	GRAV	Key Word
	5		Blank
I5	6-10	IGRAV	Direction of gravitational acceleration and forces: 1 = global X axis 2 = global Y axis 3 = global Z axis
E15.0	11-25	G	Magnitude of gravitational acceleration. (L/T^2)
	26-80		Blank

NOTES:

1. If gravity acts along a negative global axis, use a negative number for G.

2. In a static analysis, the gravity forces merely add to the applied external forces. In a dynamic analysis, the gravity forces are applied, in effect, as step functions at t=0 with resultant dynamic effects.

3. There are limitations on these gravity forces when applied to beams that are offset from their nodes. See Section 4.6.5.

This set of at least three cards is required whenever an applied displacement ACEL is specified in Group E, p. 3.48. This card set defines an acceleration versus time function to be assigned to specified degrees-of-freedom of specified nodes by means of an identity number which appears here as well as in the ACEL input. See Section 4.6.6.

Card 1:

Format	Columns	Symbol	Comments
A5	1-5	ACELT	Key word
I5	6-15	ACELID	Identity number of this table
E15.0	11-25	NPT	Number of points in the table, from 2 to 50.
E15.0	26-40	ODISP	Initial displacement (L).
E15.0	41-55	OVEL	Initial velocity (L/T).
	56-80		Blank

NOTES:

This is followed by the number of cards equal to the value of NPT given in card 1, each card containing one point on the acceleration vs. time curve. The first card must contain the initial time t=0, and the time must increase monotonically with the following cards.

Cards 2:

Format	Columns	Symbol	Comments
A5	1-5	ACELT	Key word, optional here.
	6-10		Blank
E15.0	11-25	AC	Value of acceleration at following time (L/T^2).
E15.0	26-40	TIME	Time (T).
	41-80		Blank

NOTES:

1. The applicable nodes and DOF are specified on the associated ACEL cards (p. 3.43 in Group E) bearing the identity number of this table.

2. For the special case of constant acceleration for all time, it is permitted to use only one of these cards containing t=0 and the desired constant acceleration value. All other cases must use at least two cards.

3. When the time exceeds the last time value specified here, the acceleration will be held constant at the last value.

4. A maximum of 36 time functions are allowed for the combined total of PTME, PTM2, PTM3, and ACEL sets.

/ Key Word SEND/

This card ends the Group I data, and is required <u>only if</u> this group is used.

Format	Columns	Symbol	Comments
A4	1-4	SEND	Key word
	5-80		Blank

NOTES:

This is the required last card in the input deck. If END is used, the program assumes that a new problem data deck will follow starting with the title card, p. 3.1. If STOP is used, as will probably be the case for most problems, there is no further data in the DYCAST input.

Format	Columns	Symbol	Comments
A4	1-4		Key word STOP or END
	5-80		Blank

4.0 INPUT REFERENCE

4.1 CONTROL PARAMETERS AND OPTIONS - GROUP A

4.1.1 Diagnostic Output (DYNA)

A provision has been made for the user to request certain diagnostic output not usually needed. The current options are to print:

o Load vector

o Element stiffness and mass matrices (globally oriented)

o Coefficient matrix entries and stacking indices for each element.

o Assembled coefficient matrix

This output is triggered by the NPNTC input on the first DYNA card of Group A as follows:

A 1 in column 7 prints the load vector, 1 in column 8 prints the element mass and stiffness matrices, 1 in column 9 prints coefficient matrix entries with their stacking indices, and a 1 in column 10 prints the assembled coefficient matrix.

The first two items are printed only for t = 0, while the last two will appear at every time step.

Also on this card are input data that determines how often in time steps the results are printed (NPRNT) and how often the coefficient matrix is reformed (NFORM). If NFORM is zero the matrix is only assembled at the initial step. If an implicit time integrator is selected in Group A (NEWM, WLSN, STAT), then NFORM is ignored and it is internally set to 1.

Example:

Print all four diagnostic output options. Print all results every 10 time steps and reform the mass matrix every 1000 time steps.

```
0.........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
DYNA  1111   10 1000
```

Examples of intermediate output generated with each option are shown below:

Load Vector

LOAD VECTOR DUE TO APPLIED LOAD

```
 1  1.0000000E+00  1.0000000E+00  1.0000000E+00  1.0000000E+00  1.0000000E+00  1.0000000E+00 -1.0000000E+00 -1.0000000E+00   8
 9 -1.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  16
17  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  24
25  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  32
33  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  40
41  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  48
49  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  56
57  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  64
65  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  72
73  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  80
81  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  88
```

The indices correspond to the degree-of-freedom. The first index for numbers appearing on a line is in the left-most column and the last index in the right-most column.

Element Mass and Stiffness Matrices

ELEMENT MASS MATRIX FOR MEMBER   1

COLUMNS 1-6

```
 2.2118060E-05  0.0000000E+00  0.0000000E+00  0.0000000E+00 -1.4664761E-04 -7.1499831E-06
 0.0000000E+00  1.7652997E-05  9.6203069E-08  1.2796980E-04  0.0000000E+00  0.0000000E+00
 0.0000000E+00  9.6203069E-08  1.5684543E-05  6.2393235E-06  0.0000000E+00  0.0000000E+00
 0.0000000E+00  1.2796980E-04  6.2393235E-06  1.1861716E-03  0.0000000E+00  0.0000000E+00
-1.4664761E-04  0.0000000E+00  0.0000000E+00  0.0000000E+00  2.4428285E-03 -4.6536485E-05
-7.1499831E-06  0.0000000E+00  0.0000000E+00  0.0000000E+00 -4.6536485E-05  3.3950323E-03
```

ELEMENT STIFFNESS MATRIX FOR MEMBER   1

COLUMNS 1-6

```
 1.9632927E+06  0.0000000E+00  0.0000000E+00  0.0000000E+00 -4.9327730E+07 -2.4050336E+06
 0.0000000E+00  1.3059335E+04 -9.2862522E+04  1.9731092E+06  0.0000000E+00  0.0000000E+00
 0.0000000E+00 -9.2862522E+04  1.9831610E+06  9.6201344E+04  0.0000000E+00  0.0000000E+00
 0.0000000E+00  1.9731092E+06  9.6201344E+04  6.6256287E+07  0.0000000E+00  0.0000000E+00
-4.9327730E+07  0.0000000E+00  0.0000000E+00  0.0000000E+00  1.6525318E+09  7.9485267E+07
-2.4050336E+06  0.0000000E+00  0.0000000E+00  0.0000000E+00  7.9485267E+07  2.6148135E+07
```

This example is for a stringer element, which has three DOF for each of two nodes, thus a total of six DOF. These values are already transformed into the global directions.

Coefficient Matrix Entries to be Stacked with Their Stacking Indices

ELEMENT MASS MATRIX ENTRIES TO BE STACKED, WITH THEIR STACKING INDICES, FOR MEMBER   1

```
40762  2.2118060E-05  40991  1.7652997E-05  41219  9.6203069E-08  41220  1.5684543E-05  41447  1.2796980E-04  41448  6.2393235E-06
41449  1.1861716E-03  41674 -1.4664761E-04  41678  2.4428285E-03  41902 -7.1499831E-06  41906 -4.6536485E-05  41907  3.3950323E-03
40757  1.4017013E-06  41669 -5.3707307E-05  41897 -2.6185652E-06  39617  2.2118060E-05  40986 -9.6203755E-08  41214  7.8352332E-06
41442  3.5292248E-06  39846  1.5684543E-05  40987 -7.2385120E-05  41215 -3.5292248E-06  41443 -8.6350154E-04  40074 -6.2393235E-06
40075  1.1861716E-03  40760  5.3707307E-05  41672 -1.1702784E-03  41900 -1.3987811E-04  40301  1.4664761E-04  40304  2.4428285E-03
40761  2.6185652E-06  41673 -1.3987811E-04  41901  1.6918302E-03  40529  7.1499831E-06  40532 -4.6536485E-05  40533  3.3950323E-03
```

This is the contribution of the element to the global left-hand side coefficient matrix, which, for explicit time integrators will be the mass matrix, and for implicit integrators will be a combination of the stiffness and mass matrices (except at t = 0 when only the mass matrix is used).

4.2

The stacking index integers represent the location in a one dimensional array within which the corresponding coefficient is accumulated according to the following scheme (example shown here is for a semi-bandwidth of three):

$$
\begin{bmatrix}
1 & & & & & \\
2 & 3 & & & & \\
4 & 5 & 6 & & & \\
 & 7 & 8 & 9 & & \\
 & & 10 & 11 & 12 & \\
 & & & 13 & 14 & 15 \\
 & & & & \cdot & \cdot & \cdot & \cdot \\
 & & & & & \cdot & \cdot & \cdot \\
\end{bmatrix}
$$
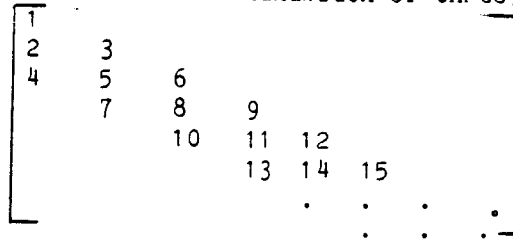
Assembled Coefficient Matrix

ROW	COL	ELEMENT	COL	ELEMENT	COL	ELEMENT	COL	ELEMENT	COL	ELEMENT	COL	ELEMENT
1	1	3.5212E+05										
2	1	3.0462E+04	2	1.0218E+05								
3	1	-3.4467E+04	2	-2.0186E+04	3	2.7487E+05						
4	1	-2.1131E+05	2	-1.9338E+05	3	1.4516E+06	4	4.7778E+07				
5	1	-5.1391E+05	2	-1.2175E+05	3	1.1591E+05	4	1.4213E+07	5	4.2268E+07		
6	1	-2.2889E+06	2	-4.2268E+05	3	3.3306E+05	4	1.5162E+07	5	1.6988E+07	6	8.5359E+07
7	1	1.2835E+05	2	1.7136E+05	3	-1.9167E+05	4	-1.9373E+06	5	2.7833E+06	6	1.2944E+06
8	1	3.8487E+07										
8	7	1.3965E+05	2	7.6416E+06	3	-3.0502E+05	4	-9.7226E+06	5	5.2909E+06	6	-2.2585E+06
9	1	2.9249E+07	2	4.4068E+07								
9	7	5.2153E+05	2	4.0123E+05	3	-2.0477E+05	4	9.0811E+05	5	-7.6921E+06	6	-2.2747E+06
10	7	4.7109E+06	2	3.6315E+06	3	2.2159E+07						
11	1	2.8596E+03	2	-2.9689E+02	3	1.3267E+04	10	2.7918E+05				
12	1	-2.9614E+02	2	4.2438E+03	3	1.9697E+03	10	2.5696E+04	11	2.5755E+04		
13	1	1.3202E+04	2	1.0506E+03	3	-4.4828E+04	10	-4.2202E+04	11	-5.1381E+03	12	2.0451E+05
13	1	-1.8451E+04	13	-1.6830E+03	3	-2.4705E+04	10	-1.1684E+05	11	-5.8016E+03	12	2.4879E+04
14	13	2.9866E+05										
14	1	-1.7128E+03	2	3.5638E+03	3	-2.1946E+03	10	-9.7714E+03	11	2.8306E+03	12	2.8359E+03
15	13	3.8753E+04	14	3.1125E+04								
15	1	-2.4710E+04	2	-2.1501E+04	3	-4.4021E+03	10	2.4879E+04	11	2.0419E+03	12	-3.7524E+04
16	13	-4.0885E+04	14	-1.2341E+04	15	1.9784E+05						
16	1	-1.4726E+05	2	-1.3084E+04	15	3.3268E+04	16	6.6712E+04	5	9.9031E+04	6	8.6511E+05
17	13	-2.2426E+04	14	-6.5086E+03	15	2.7096E+04	4	4.0718E+05				
17	1	-1.3212E+04	2	-3.0585E+03	15	1.3922E+03	4	3.3158E+05	8	1.1846E+05	6	8.2509E+05
18	13	-6.5099E+03	14	8.3416E+03	15	4.4212E+03	16	3.1120E+04	17	1.1005E+05		
18	1	3.3262E+04	2	6.3202E+03	15	-3.9995E+04	4	-6.2050E+05	5	-8.0489E+05	6	-1.7717E+05
19	13	2.7112E+04	14	-2.1438E+05	15	-4.5535E+04	16	-4.4221E+04	17	-1.8222E+04	18	2.8130E+05
19	1	9.2265E+04	2	1.0222E+04	15	-7.3316E+05	4	-1.2822E+07	5	-2.0370E+07	6	4.1523E+06
20	16	-2.1438E+05	17	1.9226E+04	19	1.7287E+06	19	5.8264E+07				
20	1	5.7566E+05	2	-1.2189E+04	3	3.7736E+05	4	9.8158E+06	5	8.8861E+06	6	-1.2474E+07

The output shows the lower triangle of the matrix. The left column contains the row index and the succeeding pairs the column index and corresponding matrix entry.

4.1.2   Summary of Computer Time (TIME)

A printout of cumulative and incremental computer time, in central precessor unit (CPU) minutes and seconds, can be initiated with a card containing the keyword, TIME, in the Group A data. The output for this option is shown below, where

STACKING	Indicates the time to assemble the coefficient matrix.
SOLUTION	Indicates the time taken to solve the system of equations.
TIME INTEGRATOR	Indicates time taken in the routine that implements the time integration algorithm.
STRESS & STRAIN CALCU-LATION AND ELEMENT FORMATION	The time taken to calculate stresses and strains, implement plastic constitutive relations, reform stiffness matrices, print results, and form any vector of internal loads.

The "total elapsed time" is cumulative only for the current job segment. If a complete analysis involves several restart segments, the printed cumulative time will have to be added manually to those from previous segments.

Note that the CPU time printed at t=0 includes the time to process the input data and perform the preliminary, one time, calculations before the first time step.

	TOTAL ELAPSED TIME THIS SEGMENT	TIME THIS INCREMENT
STACKING	0 MINUTES 7.32 SECONDS	0 MINUTES 0.28 SECONDS
SOLUTION	1 MINUTES 33.95 SECONDS	0 MINUTES 3.60 SECONDS
TIME INTEGRATOR	0 MINUTES 0.0 SECONDS	0 MINUTES 0.0 SECONDS
STRESS + STRAIN CALCULATION AND ELEMENT FORMATION	2 MINUTES 22.37 SECONDS	0 MINUTES 6.32 SECONDS
TOTAL	4 MINUTES 22.08 SECONDS	0 MINUTES 10.28 SECONDS

4.1.3   Error Bounds for Modified Adams (ADAM)

The Modified Adams time integration procedure is based on the use of predicted and corrected values for displacement and velocity. An error check on these quantities determines whether quantities at the current time step have converged. Two error terms that directly control convergence are read as Group A data on the card containing the keyword ADAM. The parameter, EPSIL, is the upper bound error that limits the size of the time step and the second, DELTA, is a lower bound error that prevents the program from seeking time steps that are too small for the desired accuracy. If the error exceeds the upper bound then the time step is halved; if the error is less than the lower bound the time step is doubled. In practice, the upper bound should be at least twice the lower bound in order to prevent excessive halving and doubling.

If the error bounds are not specified then defaults of 10%, 1% are taken.

A last parameter (ERR) on the card is essentially a tolerance that eliminates the possibility that roundoff errors will effect the convergence. The default for this quantity is $1. \times 10^{-4}$.

Example: 15% upper bound, 5% lower bound

```
0........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890123456789  0
ADAM      0.15            0.05
```

4.1.4 Restart (REST)

The restart capability has been designed with a two-fold purpose: to restart an analysis and to postprocess results of the analysis. To accomplish this, an external data base or "restart" file is created and manipulated by the program, with user control by the REST card. First, initial information (model geometry, material properties, etc.) is written on this restart file after the input is processed. Then, a basic set of data consisting only of displacements, velocities, and accelerations are saved on this file at each time step for postprocessing purposes. In addition, a complete set of data (including stresses, strains, forces, etc.) are written at time step intervals IPRNT, at which points the analysis can be restarted. The restart data base file can be stored on any of the user's computer system storage media; for example, tape or disk.

When initially starting the analysis, the restart file is generated by setting IREST = 1.0 and specifying values for IPRNT and KPRINT (NUTAP and NRSRT are ignored). If the entire REST card is omitted, then no restart file is created and no restarts can be made, nor will there be any output data file for post-processing.

When restarting an analysis, IREST = 2 or 3. The two options differ only in that IREST = 2 copies the current restart file before adding to it while IREST = 3 adds new data directly to the existing file. The parameter NUTAP indicates whether the restart file is to be continued, where NUTAP = 0 denotes that no further data is written and NUTAP > 0 denotes that data will continue to be written on the file. The parameter NRSRT, indicates the time step from which the continuation run will be re-started. NRSRT is the time step number printed with an associated time during the run that generated the restart file. IPRNT is again specified, and can be charged on a restart.

The four IREST options 0,1,2,3 will usually require some differences in the Job Control Logic (JCL) for the user's computer system, to access or not access the external files.

Example 1:

An initial restart file is to be generated, with full restart data every ten time steps, and basic output data every step.

```
0........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890
REST    1.0                       10.0
```

Example 2:

Analysis is to be restarted from a previous job at time = 0.0010 sec with full restart data every ten steps. From printout of previous segment, NRSRT = 21 was associated with the desired time.

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
REST      2.0            1.0            10.0            20.
```

The NRSRT number from a previous computer run appeared as below.

■■■
RESTART DATA WRITTEN ON OUTPUT FILE AT TIME = 1.0000000E-03   STEP NUMBER (NRSRT) = 20
■■■

4.1.5　Static Analysis (STAT)

A static analysis can be performed by specifying the keyword STAT in group A data. This keyword has the effect of suppressing the mass contribution in the equations of motion so that the following incremental equation is solved;

$$K_n \, \Delta U_{n+1} = P_{n+1} - f_n \qquad (4.1.1)$$

where

K_n　　　is the incremental stiffness matrix

P_{n+1}　　is the applied load vector

f_n　　　is the vector of internal forces at the start of the load step

ΔU_{n+1}　is the displacement increment

All the input for a dynamic analysis is used in a static analysis with the exception of initial conditions and lumped masses, Groups F and G input respectively.

The maximum load and load increment are controlled by specifying the maximum time and time step on the second DYNA card in Group A input. This is used with the table of load factor versus time, input with the keyword PTME in Group I data. This table is a necessary input with a static analysis. (See Section 4.6.4). The time is used only as a parameter to generate the loads.

The current load level or load increment is obtained from the table by inter polating between any of the input values. One segment can be input if a constant load step procedure is to be used. A variable step procedure can be defined by inputting a number of segments. In this way, for example, large load steps can be taken at lower load levels and smaller steps at higher loadings where the problem nonlinearities may require smaller time steps. The load (time) steps will be further varied, in any case, by the convergence test.

4.1.6 Eigenvalue/Eigenvector Extraction (EIGN)

DYCAST implements an efficient algorithm for determining eigenvalues and eigenvectors of large symmetric matrices. Specifically it implements an existing program called ALARM (Automatic Large Reduction of Matrices to tridiagonal form). ALARM is based upon an algorithm presented by Ojalvo and Newman[*] which reduces a large matrix to an "equivalent" tridiagonal form of much smaller size. It is similar to the eigensolver FEER, which has been implemented in recent versions of NASTRAN.

The strength of the algorithm implemented in ALARM lies in the ability to perform the reduction in such a manner as to obtain the eigenvalues at either end of the original matrix spectrum. In DYCAST this selectivity leads to accurate calculation of the lower frequencies and modes. The size of the reduced matrix is usually approximately 1.5 times the number of desired modes. The number of modes is specified on the EIGN card by the parameter IQ. Internal error checking may increase the order of the reduced matrix until it is rich in the lower modes that are desired. Fundamental to the algorithm is a method that enforces each successive column of the reduction matrix to be orthogonal to all other columns by an iterative method. Experience has indicated that no more than two iterations are necessary for convergence. The number of iterations is specified on the EIGN card as NSBAR.

The algorithm requires the stiffness matrix to be nonsingular. In the event that the stiffness matrix is singular, a shift in [K] by some small amount will be required in order to perform a matrix factorization, i.e.,

$$[K] + \epsilon[M] = [\bar{K}] = [L][L]^t. \qquad (4.1.2)$$

The term, ϵ, appears as EPS in the fifth field of the EIGN card.

---

[*] For full details of the method the reader is referred to "ALARM" - A Highly Efficient Eigenvalue Extraction Routine for Very Large Matrices," by I. U. Ojalvo, The Shock and Vibration Digest, Vol. 7, No. 12, December 1975.

4.1.7 Automatic Element Failure (FAIL)

Material failure can occur when the principal strains equal or exceed the user-specified failure strain values in both tension and compression. These failure strains are input in the Group H material properties card sets (MBM, MSTG, MAT1, MAT2, MAT3). The failure criterion is applied to each stress integration point within an element, provided that a non-zero tension failure strain is specified for that element. If the tension failure strain is not specified for an element, then failure of that element cannot occur and its material will have infinite strain capacity.

In the case of beam and stringer elements (MBM and MSTG inputs), only the longitudinal strains are monitored and compared to the failure strains. For the biaxially strained isotropic membrane and plate elements (MAT1 input), the in-plane principal strains are calculated and compared to the failure strains. In the case of the orthotropic biaxially strained elements (MAT2 and MAT3 inputs), the failure strains vary with direction within the material. The user is required to specify the failure strains in the material's directions of in-plane orthotropy, and the program assumes a smooth elliptical variation with angle between these two orthogonal directions. The in-plane principal strains are then compared to the failure strains in the directions of the principal strains. The failure of nonlinear springs is based on user-specified maximum values (+ and -) of elongation or rotation.

When an integration point fails, the values of stress and stiffness assigned to the finite amount of material surrounding that point are set to zero, and its strain energy is fixed at the current value, thereby becoming totally nonrecoverable. Thus it can be seen that an element having only one integration point, such as a stringer or a membrane, will be suddenly and completely failed whenever its single integration point satisfies the failure criteria. For the beams, when all the integration points at either end node have failed, then the entire beam is failed. A "partial failure" of a beam element is defined as when at least one integration point has failed, but a complete element failure has not occured. At present, the automatic failure feature for plate elements is not operational.

In this way, a partial failure of a beam may progress through its cross-section, steadily reducing its forces, moments, and stiffnesses. This progressive failure may actually reach a stable state, without further failure, if the element is buried within a redundant structure containing other load paths. Since the stiffness of the partially failed element is reduced, its internal forces may transfer to the remaining stiffer load paths, and further progression of the partial failure may cease.

Should the required conditions for complete failure be achieved in an element, then all that element's forces, moments, and stiffnesses become zero. Its strain energy is "frozen" at the previous value, with no subsequent changes. The portion of the total strain energy that was elastic or recoverable, is now arbitrarily made nonrecoverable. Thus the elastic strain energy contained within the failed element is not released to the system to be converted to kinetic energy or redistributed. (In an actual material, some of this elastic strain energy is recovered by the system.)

The action taken when a complete element failure is detected is as follows:

o   If the FAIL card was used in Group A, the element's stiffness and internal forces are automatically deleted for the next time step, and a message is printed that the element has failed and has been deleted.

o   If the FAIL card was not used in Group A, no action is taken, except that a warning message will be printed that the element would have "failed" but has not been deleted.

The inertia (mass and rotary inertia) of a deleted element is not deleted, on the assumption that although the element is no longer capable of carrying load, it is still physically attached to the rest of the structure. If its inertia matrix was a consistent one, it is diagonalized. This has the effect of lumping the inertia at the element nodes.

If desired, the inertia of failed elements (or any inertia in the model) can be deleted by using the added inertia feature of Group G. In that case, negative values of the existing nodal inertias would be manually added to the appropriate nodes at any desired restart time step (NRSRT), thereby removing such inertia from the structure.

If a single node or group of nodes and elements become completely detached from the structure because of element failures, it will be treated as a separate body.

The failure status and the deletion status appear also in the plasticity and failure table whose printout is activated by the PFTAB card, p. 3.19 and 4.13.

The following table summarizes the automatic element failure procedures. Note that elements can be manually deleted on a restart by using the DELE input of Group A.

ELEMENT	FAILURE CRITERION[a]	PARTIAL FAIL	COMPLETE FAIL	ACTION
SPNG	ELONGATION OR ROTATION	NO	YES	ELEMENT DELETED[b] IF FAIL CARD[c] USED. WARNING ONLY, IF NO FAIL CARD
STRG	AXIAL STRAIN	NO	YES	"
TRIM	PRINCIPAL STRAIN	NO	YES	"
BEAM	AXIAL STRAIN	AT EACH STRESS POINT	WHEN ALL STRESS PTS. FAIL AT EITHER END NODE	"
TRP2	NOT CURRENTLY OPERATIONAL			

a. If no failure criterion is given in the Group H input for an element, it will not be examined for failure.

b. Deletion results in zero forces, moments, and stiffnesses, and frozen strain energy.

c. The FAIL card of input Group A applies universally to all elements.

4.1.8 Table of Plastic and Failed Elements (PFTAB)

If the PFTAB card is used, a summary of all elements that are plastic or have failed will be printed at every step for which printout (NPRNT) has been requested on the first DYNA card. An example is shown below. The table contains the element type and number, and columns that show when (time and step number) an element first became plastic, when it completely failed, and when it was deleted. An element is listed as plastic if at least one stress point is in the plastic range (exceeds the yield stress specified in Group H, Sect. 4.3). Elements that have reached a failure criterion will be automatically deleted only if a FAIL card (Sect. 4.1.7) is specified in the Group A data. Elements that are still elastic and have not failed will not appear in this table.

TABLE OF PLASTIC, FAILED AND DELETED MEMBERS
AT TIME = 1.0000000E-03   INCREMENT NUMBER = 26

MEMBER TYPE	MEMBER NO.	PLASTIC TIME	PLASTIC INCREMENT	FAILED TIME	FAILED INCREMENT	DELETED TIME	DELETED INCREMENT
TRIM	52	2.0000E-04	5	2.5000E-04	7	2.5000E-04	7
TRIM	56	1.5000E-04	3	1.5000E-04	3	1.5000E-04	3
TRIM	57	3.2500E-04	10	3.5000E-04	11	3.5000E-04	11
TRIM	58	3.7500E-04	12	4.1250E-04	13	4.1250E-04	13
TRIM	258	1.0000E-03	26				
TRIM	259	7.5000E-04	21				
TRIM	265	1.0000E-03	26				
TRIM	266	4.8750E-04	15	5.2500E-04	16	5.2500E-04	16
TRIM	267	4.8750E-04	15	5.2500E-04	16	5.2500E-04	16
TRIM	268	3.7500E-04	12	4.5000E-04	14	4.5000E-04	14
TRIM	269	4.1250E-04	13	4.8750E-04	15	4.8750E-04	15
TRIM	270	1.0000E-04	2	1.0000E-04	2	1.0000E-04	2
TRIM	271	1.0000E-04	2	1.0000E-04	2	1.0000E-04	2
TRIM	272	1.0000E-04	2	1.0000E-04	2	1.0000E-04	2
TRIM	273	1.0000E-04	2	1.0000E-04	2	1.0000E-04	2
TRIM	274	4.1250E-04	13	6.5000E-04	19	6.5000E-04	19
TRIM	275	4.1250E-04	13	4.8750E-04	15	4.8750E-04	15
TRIM	276	4.5000E-04	14	5.2500E-04	16	5.2500E-04	16
TRIM	277	4.5000E-04	14	5.2500E-04	16	5.2500E-04	16
TRIM	295	1.5000E-04	3	2.0000E-04	5	2.0000E-04	5
TRIM	298	6.5000E-04	19				
TRIM	299	2.5000E-04	7	2.7500E-04	8	2.7500E-04	8
TRIM	300	2.0000E-04	5	2.7500E-04	8	2.7500E-04	8
TRIM	301	1.5000E-04	3	1.7500E-04	4	1.7500E-04	4
TRIM	308	4.5000E-04	14	1.0000E-03	26	1.0000E-03	26
TRIM	311	4.5000E-04	14	1.0000E-03	26	1.0000E-03	26
TRIM	317	9.0000E-04	24				
TRIM	348	8.5000E-04	23				
TRIM	486	1.0000E-04	2				
TRIM	487	1.0000E-04	2				
TRIM	488	1.5000E-04	3				
TRIM	489	1.5000E-04	3				
TRIM	494	6.5000E-04	19				
TRIM	496	7.5000E-04	21				
TRIM	507	7.5000E-04	21				
TRIM	509	8.0000E-04	22				
TRIM	523	4.8750E-04	15				
TRIM	524	7.0000E-04	20				
TRIM	525	4.8750E-04	15	5.6250E-04	17	5.6250E-04	17
TRIM	526	6.5000E-04	19	7.0000E-04	20	7.0000E-04	20
TRIM	527	3.0000E-04	9	5.6250E-04	17	5.6250E-04	17
TRIM	528	3.0000E-04	9	8.5000E-04	23	8.5000E-04	23
TRIM	529	2.5000E-04	7				
TRIM	530	2.7500E-04	8				
TRIM	531	2.5000E-04	7	2.7500E-04	8	2.7500E-04	8
TRIM	532	2.2500E-04	6	2.7500E-04	8	2.7500E-04	8
TRIM	533	1.5000E-04	3	2.0000E-04	5	2.0000E-04	5
TRIM	534	1.0000E-04	2	2.0000E-04	5	2.0000E-04	5

4.1.9   Manual Deletion of Elements (DELE)

Any element can be manually deleted at the beginning of any restart segment. This can be used to simulate the failure of any structural element at some point in the calculations. If it is desired to delete a group of elements, the key word DELE is specified in the Group A data along with a list of elements to be deleted. Once a manual deletion is specified on a restart, it is not necessary to repeat the deletion request for that element at subsequent restarts. However, it will probably be convenient to repeat the previous deletions, and merely add the new ones to the end of the list.

When an element is deleted its stiffness contribution and internal force contribution to overall equilibrium is deleted from the equations of motion. The mass contribution, however, is maintained unless otherwise specified. When an element is deleted, its mass contribution is no longer calculated from a consistent mass matrix, but rather by "lumping" proportional mass components at each node of the element. Inertia can be deleted for a group of deleted elements by specifying negative values of the existing inertia components to be added at the appropriate nodes at a convenient restart time step (NRSRT). Since the inertia are additive, this action will remove such inertia from the structure.

4.1.10 Bandwidth Reduction (BAND)

Use of the optional BAND card in Group A will cause a re-ordering of the Group B node list to reduce the bandwidth of the coefficient matrix, using an algorithm devised by Collins*. This will reduce the core storage requirements and the computer time, and is essential for large problems.

The re-ordered node list will be printed with the processed input data, output on Unit 7 as a card image file for future use in the format required for the DYCAST Group B input, and used directly in the current problem solution.

This re-ordering process includes the MPC specifications as nodal connectivities, along with the elements.

After using BAND the first time, the re-ordered nodal list from Unit 7 should be used to replace the original nodal list on subsequent runs, with the BAND card deleted. This process should be repeated whenever the connectivity of the model is changed, especially for large problems.

The BAND option previously used in the pre-processing program SATELLITE (Section 5) is also available, but is no longer supported.

*R.J. Collins, "Bandwidth Reduction By Automatic Renumbering" International Journal for Numerical Methods In Engineering, Vol. 6, pp. 345-356, 1973.

The Group B input defines an allowable set of external node point numbers. The program uses this information in two ways. First, it sets up a table of allowable node points that is used to check all subsequent node point input. Second, the program converts each external node number to an internal number consecutively in the order that the node appears on the input card. Consequently, the order of the input of external node numbers is completely arbitrary and need not be increasing monotonically. Once the input is read the program operates with the internal numbers which are now numbered from 1 through the maximum number of nodes in the model. In this manner the node ordering and therefore the bandwidth of the stiffness matrix can be easily changed and nodes can be inserted or deleted by changing the external node specification. A bandwidth optimizing subprogram is part of the input preprocessing program SATELLITE. This program takes any order of external node number and prints a reordered list of external node numbers which leads to an optimum bandwidth. (See Section 5.1) Alternatively the BAND option, Group A data p. 3.24 can be used.

All the shorthand notation for fixed point input can be used with this input.

Example 1:

Given a model with 100 nodes numbered consecutively, use shorthand notation:

```
0.......1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
      1-100
```

Example 2:

Given a model with 100 nodes numbered consecutively 1 through 25, 31, 33, 34, 36, 50, 52, 54, ... 170

```
0.......1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
      1  -25  31   33   35   36   50   -2 -170
```

Both shorthand notations are used, where 1-25 indicates 1 through 25, and 50-2-170 indicates 50 through 170 in increments of two.

Example 3:

Given a model with 57 nodes ...

Original order was

```
0.......1.........2.........3.........4.........5.........6.........7........8
1234567890123456789012345678901234567890123456789012345678901234567890
   100 -107  108 1001 2000 4000 6000 3001  200 -207  208 1003 2003 4002 6002 3003
   300 -314 1005 2004 4004 6004 3005 8000 8004  400  406  408  411  414  422
```

The bandwidth optimization from the SATELLITE preprocessing program (Sect. 5.1) or the BAND option indicated the following order:

```
0.......1.........2.........3.........4.........5.........6.........7........8
1234567890123456789012345678901234567890123456789012345678901234567890
   101  100  103 1001  102  105  201  104 2000  108 3001  200  203  308  106  202
  6000  205  303 4000  204 2002  208 3003  301  305 1005  206  304 6002  311  300
   307  306 4002  302 2004  207  309  313 3005  312  408  314 6004 8000  411  400
   310 4004  414  422 8004  406  107  203
```

This new order decreased the bandwidth by a factor of two.

4.17

The theoretical basis of the elements are described in the DYCAST theoretical manual. In this section a brief description of the theory is presented along with an explanation of the input data cards necessary to use each element.

The topics discussed for each element include:

o   Introduction

o   Input Data

o   Element Connectivity

o   Material and Section Properties

o   Loads

o   Description of Output

4.3.1 Plasticity in General (MAT1, MAT2, MAT3, MSTG, and MBM)

The methods used to implement plasticity theories into a finite element code by now are well developed and have been reported in many references. The theoretical basis and the relevant equations that mathematically describe the plasticity theory implemented in DYCAST is described in Section 1. This section outlines the form of the constitutive equations in a general way and describes more fully the input required to specify material properties.

DYCAST makes use of a flow theory of plasticity. Basic to this approach are the definitions of an initial yield criterion, and flow and hardening rules. The initial yield criterion used is based on Hill's equations for orthotropic material behavior which reduces to the von Mises yield criterion for an isotropic material. Use of Hill's equation requires the input of the yield stresses in the three principal directions of orthotropy as well as a yield stress in shear. For an isotropic material, this reduces to the specification of a single quantity, the yield stress, from a uniaxial tensile test.

From the flow and hardening rules the following incremental relation between the increments of plastic strain and stress is obtained.

$$\{\Delta\epsilon^p\} = [C] \{\Delta\sigma\} \qquad (4.3.1)$$

where the terms of [C] are path dependent quantities that reflect the instantaneous states of stress and hardening of the material and the choice of plasticity theory. In DYCAST use is made of the Prager-Ziegler kinematic hardening theory. Also contained in [C] is a material parameter characterizing the hardening of the material. In the one dimensional case this is represented by the slope of the stress versus plastic strain curve. This is generalized to multiaxial stress conditions by assuming an effective stress - effective plastic strain relation. Both linear and nonlinear strain hardening options are available with input parameters determining which is chosen. The Ramberg-Osgood three-parameter representation of the stress-strain data is used for nonlinear hardening, where the plastic strain depends on a power of the stress. This nonlinear relationship is written as

$$\epsilon = \frac{\sigma}{E} + \frac{3\sigma_{0.7}}{7E} \left(\frac{\sigma}{\sigma_{0.7}}\right)^n \qquad (4.3.2)$$

Thus, for this representation of the stress strain curve, two additional material parameters, n, $\sigma_{0.7}$, are required. To determine the values of n and $\sigma_{0.7}$ that best fit the actual stress-strain data, the method suggested by Ramberg-Osgood

4.19

can be used if the strain range is sufficiently small (on the "knee" of the stress-strain curve). For this case

$$n = 1 + \frac{\text{Log}(17/7)}{\text{Log}(\sigma_{0.7}/\sigma_{0.85})} \qquad (4.3.3)$$

The quantities $\sigma_{0.7}$ and $\sigma_{0.85}$ are the stresses at which the curve has a secant moduli of 0.7E and 0.85E, respectively. If the strain of interest is sufficiently large (past the "knee" of the stress-strain curve) so that the parameters as determined by the preceding process do not fit the curve well, then a power law representation to fit the actual data can be used,

$$\varepsilon = \frac{\sigma}{E} + \beta\sigma^n \qquad (4.3.4)$$

where n is the same as before.

Once β and n have been determined to "best fit" the experimental data the value of $\sigma_{0.7}$ is obtained as

$$\sigma_{0.7} = \left(\frac{3}{7E\beta}\right)^{\frac{1}{n-1}} \qquad (4.3.5)$$

Linear strain hardening is treated with a bilinear representation of the stress-strain curve,

$$\varepsilon = \sigma/E, \qquad \text{for} \quad \sigma \leq \sigma_y \qquad (4.3.6)$$

$$\varepsilon = \frac{\sigma_y}{E} + \frac{\sigma - \sigma_y}{E_t} = \frac{\sigma}{E_t} + \frac{\sigma_y}{E_t}\left(\frac{E_t}{E} - 1\right), \text{ for } \sigma > \sigma_y$$

where σ_y is the yield stress and E_t is the slope of the post yield portion of the curve. In this case the hardening parameter is obtained from the ratio E_t/E.

Another assumption that is used to develop the appropriate equations is that the increment of total strain may be decomposed into an elastic and plastic component. This assumption leads to the incremental constitutive relations for the stresses and plastic strains in an elastic-plastic material.

$$\{\Delta\sigma\} = [D]\{\Delta\varepsilon\} \qquad (4.3.7)$$

4.20

and, from Eq (4.3.1),

$$\{\Delta\epsilon^p\} = [C][D]\{\Delta\epsilon\} \qquad (4.3.8)$$

where $\Delta\epsilon$ is the increment in total strain, $[D] = [E^{-1} + C]^{-1}$, and $[E]$ contains the usual elastic material parameters.

Explicit forms for the relations in Eqs. (4.3.7) and (4.3.8) that are used in DYCAST are in Section 1.2. Equations similar in form to Eq (4.3.8) for perfect plasticity are also shown in that manual.

The following table lists the stress-strain curve types and the elements to which they apply. Tension and compression properties are assumed to be the same, except for failure.

Key Word	Element	Material Type, Elastic	Yield Criterion	Material Type, Plastic
MAT1	TRIM TRP2	Isotropic biaxial	Isotropic,	perfect plasticity, linear strain hardening, non-linear strain hardening
MAT2	TRIM TRP2	Orthotropic biaxial	Orthotropic	perfect plasticity
MAT3	TRIM TRP2	Orthotropic biaxial	Orthotropic	linear strain hardening, nonlinear strain hardening
MSTG	STRG	Isotropic biaxial	Isotropic,	perfect plasticity, linear strain hardening, non-linear strain hardening
MBM	BEAM	Isotropic biaxial*	Isotropic,	perfect plasticity, linear strain hardening, non-linear strain hardening

* Torsional shear included for solid and closed hollow cross-sections only, otherwise uniaxial.

Cautionary Note:

In application, the linear elastic stress-strain curve, $\epsilon = \sigma/E$, is followed until the user-specified "yield stress" is exceeded. Then the program switches to Eq (4.3.2). Therefore, the "yield stress" input for the Ramberg-Osgood nonlinear curve should be equal to or less than the stress at which the nonlinear stress-strain curve departs significantly from the elastic line. Otherwise, the nonlinear part of the stress-strain curve will be incorrectly used by the program.

4.3.2   Stringer Element

4.3.2.1   Introduction

This element is used to represent a one dimensional axial force structural member. Two stringer elements are included: a two-node element developed from a constant axial strain assumption and a three-node element developed from a linearly varying strain assumption (Fig. 4.3.1). As shown in Fig. 4.3.1, the indices i,j denote the end point nodes of the two-node stringer and the indices i,j,k denote the end point and mid-length nodes of the three-node stringer. The specification of the axial displacement component u is sufficient to describe fully the elastic linear response of a stringer element since the element has no out-of-plane (normal to the axial direction) stiffness. The normal components, v,w, contribute to the elastic geometrically nonlinear response by coupling the axial force to the out-of-plane deformations This coupling leads to the initial stress stiffness matrix $[k^I]$. It should be noted that displacement components v, w, as shown in Fig. 4.3.1 refer to any two mutually perpendicular directions normal to the stringer axis since the stringer does not have a preferential cross sectional reference system.

(A) CONSTANT STRAIN ELEMENT
   Displacement Assumption
   $U = A_1 + A_2X$

(B) LINEAR STRAIN ELEMENT
   Displacement Assumption
   $U = A_1 + A_2X + A_3X_2$

Fig. 4.3.1   STRINGER

4.22
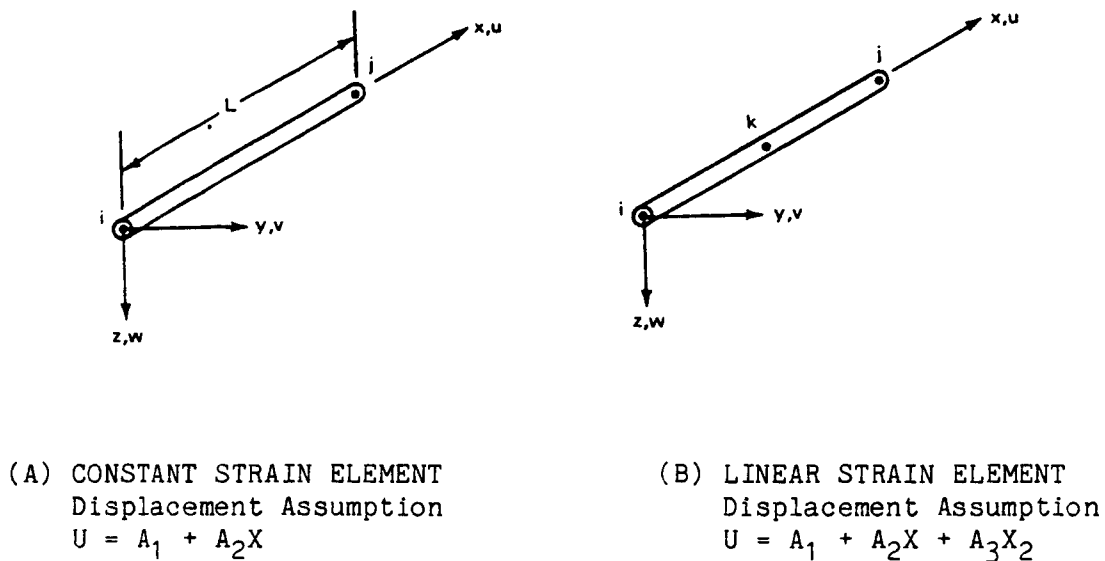
4.3.2.2 Stringer Input Data

The data for the stringer element are specified with the following input types:

Group	Key Word	Required	Comments
C	STRG	Yes	Element identification and connectivity.
H	MSTG	Yes	Element material and section properties.
I	CONC	No	Concentrated forces at nodes.

4.3.2.3 Stringer Connectivity (STRG, Group C)

The nodes of the stringer element are specified with both major (end- point) nodes first, followed by the minor (midside) node. The absence of the midpoint node must be indicated by a zero or blank field in the proper position. The member identification can be any number of up to 5 digits. This number need not be consecutive but must be unique for the job.

Example:
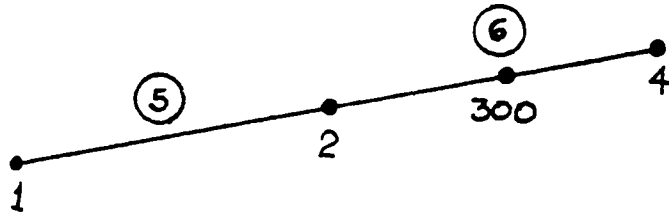

```
0........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890
STRG        5    1    2
STRG        6    2    4   300
```

4.3.2.4 Stringer Material and Section Properties (MSTG, Group H)

The following material and section properties for the axial force stringer are input:

E	Youngs's modulus
A	Cross-sectional area
PLAS1 PLAS2	See Table below
YLDST	Yield stress (See Cautionary Note on p 4.21)
DENSITY	Mass density (mass/unit volume)
SEPST	Failure strain in tension (+)
SEPSC	Failure strain in compression (-)

Three types of plastic behavior can be considered, i.e., perfect plasticity, linear, or nonlinear strain hardening. They are indicated as follows:

Input Parameters		Types of Plasticity
PLAS1	PLAS2	
0	0	Perfect plasticity (zero slope in plastic range)
E_t/E	0	Linear strain hardening, with E_t equal to slope of the uniaxial stress versus strain curve in the plastic range
n	$\sigma_{0.7}$	Nonlinear strain hardening with Ramberg-Osgood representation of the uniaxial stress-strain curve, see p. 4.19.

Stringer material properties should be taken from curves of engineering stress versus engineering strain. The engineering stress should be used because the program multiplies it by the initial undeformed area (A) to get the resultant axial force. The engineering strain should be used because the program converts it to true strain internally (for stringer elements only). Tension and compression properties are assumed to be the same, except for failure.

Note that, for ductile homogeneous materials, actual compression failures rarely occur in the material. The first compression damage is usually from plastic shearing or geometric instability (elastic or plastic buckling or collapse). In these cases, the compression failure strain input can be made very large to prevent failure, or the compression strain can be specified at which the actual damage is expected. Instability can be more accurately determined by using beam and plate elements.

Example:

    50 stringer elements, aluminum alloy

    Area = .1 in^2 for elements 1-25

    Area = 0.15 in^2 for elements 26-50

    All material properties the same

    Density = 0.1 lb/in^3 (2.588 x 10^{-4} lb sec^2/in^4)

    Young's Modulus = 10 x 10^6 psi

    Yield = 36,000 psi

    Linear strain hardening slope = 2 x 10^6 psi

    No failure criterion for elements 1-25

    Failure strain of 10% in tension and compression for elements 26-50

```
0........1.........2.........3.........4.........5.........6.........7.........8
123456789012345678901234567890123456789012345678901234567890123456789012345678 90
MSTG     1.0     E+07    0.1             0.2             0.              36000.
         2.588   E-04                    0.0             0.0
      1  -25
MSTG     1.0     E+07    0.15            0.2             0.              36000.
         2.588   E-04                    0.1             -0.1
     26  -50
```

4.3.2.5 Stringer Loads (Group I)

The following mechanical loads may be applied to the stringer element:

CONC - Concentrated forces applied at specified nodes in the global
       directions in the units of force.

In addition, a table (PTME, or PTM2, or PTM3) of the load magnitude versus time, must be specified.

These loads are discussed in Section 4.6.

4.27

4.3.2.6 Description of Stringer Output

This section shows examples of typical output generated when using the stringer element.

Stringer Connectivity

MEMBER	STRINGER ELEMENTS		
	NODE I	NODE J	NODE K
29	102	2000	
30	2000	202	
31	202	2002	
32	2002	304	
33	304	2004	
34	2004	408	
35	4000	106	
36	206	4000	
37	4002	206	
38	312	4002	
39	4004	312	
40	422	4004	
75	1001	101	
76	201	1001	
77	1003	201	
78	303	1003	
79	1005	303	
80	403	1005	
81	105	3001	
82	3001	205	
83	205	3003	
84	3003	311	
85	311	3005	
86	3005	411	

In the absence of any three-node stringer elements, the third field for nodes contains a blank. The midside external node number appears in that location when a three-node stringer is specified.

Stringer Material Properties

STRINGER SPECIFICATION TYPE   1

YOUNG'S MODULUS	AREA	PLAS1	PLAS2	YIELD STRESS
3.0000000E+04	1.0000000E-04	0.0000000E+00	0.0000000E+00	3.0000000E+04

DENSITY			FAILURE STRAIN	
			TENSION	COMPRESSION
0.0000000E+00			0.0000000E+00	0.0000000E+00

APPLICABLE MEMBERS
301   302   303

This printout is generated when specifying stringer data with cards containing the keyword MSTG.

Stringer Solution Output

The output shown below is for a two-node stringer since results for NODE K (the midside node) are blank. The nodal forces at each node are in the element axial direction and AVERAGE STRAIN and EPS are the total and plastic strain, respectively. The state of plasticity is indicated by the column labeled "PLASTIC". A "NO" indicates an elastic element, "YES" indicates a plastic element, and "UNL" an element that has unloaded elastically from a previously plastic state. Failed elements are deleted from this output.

NODAL FORCES AND AVERAGE STRESS AND STRAIN FOR STRINGER MEMBERS
AT TIME = 2.0000000E-03   INCREMENT NUMBER =   40

MEMBER	PLASTIC	NODE I	NODAL FORCE	NODE J	NODAL FORCE	NODE K	NODAL FORCE	AVERAGE STRESS	AVERAGE STRAIN	AVERAGE PLASTIC STRAIN
301	NO	86	-6.678665E-06	120	6.678665E-06			6.678665E-02	2.226222E-06	0.000000E+00
302	NO	120	2.744217E-04	11	-2.744217E-04			-2.744217E+00	-9.147391E-05	0.000000E+00
303	NO	120	-4.485425E-04	9	-4.485425E-04			4.485425E+00	1.495142E-04	0.000000E+00

4.3.3 Beam Element

4.3.3.1 Introduction

Some general features and limitations of the beam elements are:

o   Initially straight.

o   Two nodes with 6 degrees of freedom per node.

o   Constant cross-section along length.

o   Thirteen pre-formed special cross-sections (L,T,C,etc.), with user specified dimensions.

o   Elastic geometric stiffness quantities (Area, moments of inertia) can be either user-specified or calculated by the program. The torsional stiffness constant must be input.

o   Longitudinal, bending, and torsional deformations. No transverse shear deformation.

o   Interactive yield criterion involves combined normal and shear stresses due to longitudinal forces, bending moments, and torsion for closed thin-walled and solid cross-sections. The shear stresses due to torsion are neglected for open thin-walled sections.

o   Each beam element must be homogenous (one material), but any number of beam elements, each having its own material, can be combined into a beam assembly that acts as a single beam between common end nodes.

o   Cross-sections do not deform, however the reduced bending strength due to distortions of hollow, thin-walled beams can be represented by a nonlinear rotary spring having a user-specified curve of moment vs. rotation.

There are four sets of local axes used in the beam elements as follows (Figs 4.3.2 through 4.3.4):

o   <u>Nodal axes</u> x,y,z - Defined by the nodes, and describe the general position of the beam's initial rotational position about its longitudinal x axis, and is defined by the "pointer" node (Fig 4.3.2). This is described in detail later in Section 4.3.3.3.

o   Cross-sectional axes x'y'z' - Fixed in the cross-section, with origin at its shear center. Usually coincident with the reference axis x"y"z" (default condition) as shown in Fig. 4.3.2. Can be rotated and translated relative to the reference axes as in Figs 4.3.3 and 4.3.4. The location of these axes for each beam cross-section is given in Section 4.3.3.5.

o   Centroidal axes $\bar{x}$ $\bar{y}$ $\bar{z}$ - Parallel to the cross-sectional axes, but through the centroid of area for the cross section.

o   Reference axes x"y"z" - Parallel to the nodal axes, with origin at the shear center, relative to which the cross-sectional axes x'y'z' can be rotated and translated in the transverse plane as in Figs 4.3.3 and 4.3.4. Fig 4.3.2 shows the (default) use of no rotation or translation. Used for changing the rotational orientation of a beam from that defined by the nodal axes, or for combining a number of separate beam elements (different materials allowed) into an assembly that acts as a single beam. The origin of these reference axes must be placed at the shear center of the combined beam assembly.

Note that the cross-sectional axes x'y'z' remain fixed in the cross-section and rotate and translate with it. In the right side of Fig. 4.3.4, the other beam cross-sections that form the combined beam are omitted. The reference axes x"y"z" are centered at the shear center of the combined beam assembly and apply to all the beams that are combined into the assembly, and serve to locate the relative positions of these beams.

In actuality, the reference axes x"y"z" are always present, but are coincident with the cross-sectional reference axes x'y'z' when $y_A=z_A=\beta=0$ in Figs 4.3.3 and 4.3.4. Thus the user can ignore these input parameters and the reference axes until they are needed. The procedure and examples for creating a combined beam assembly are given in Section 4.3.3.5.

These local axes completely define the initial (undeformed) position of a beam element in the structure. As the beam deforms, these axes move together relative to the global coordinate system.

In the following discussion the terms "reference axes" and "shear center" pertain to the cross-section reference axes and their associated shear center unless the new reference axes are used.
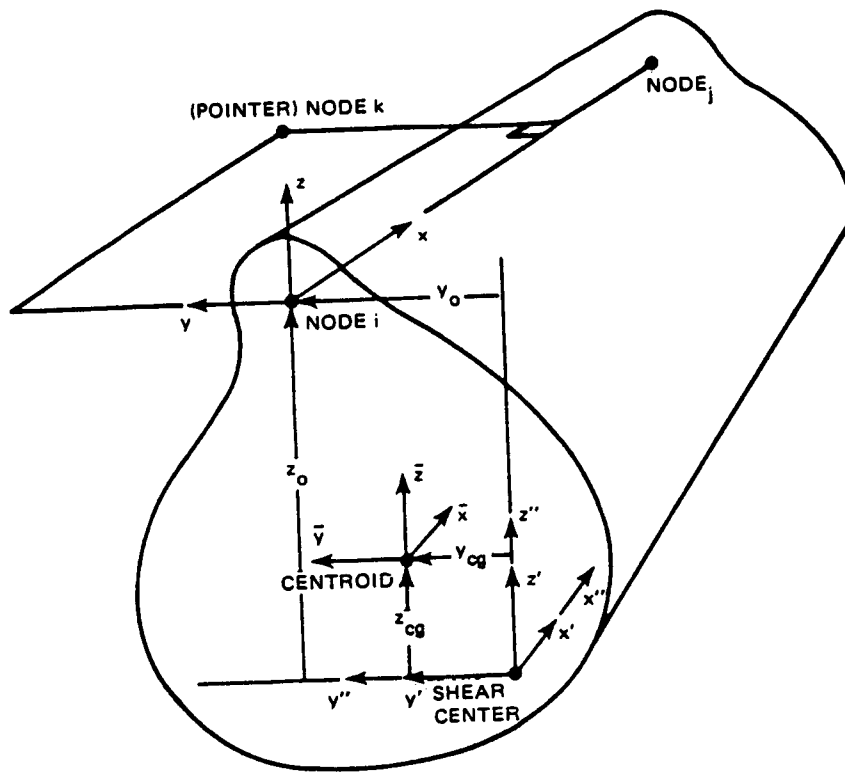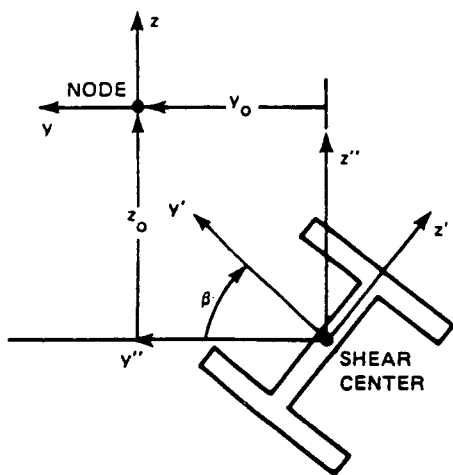
Fig 4.3.2 Basic Axes For Beam Cross-Sections

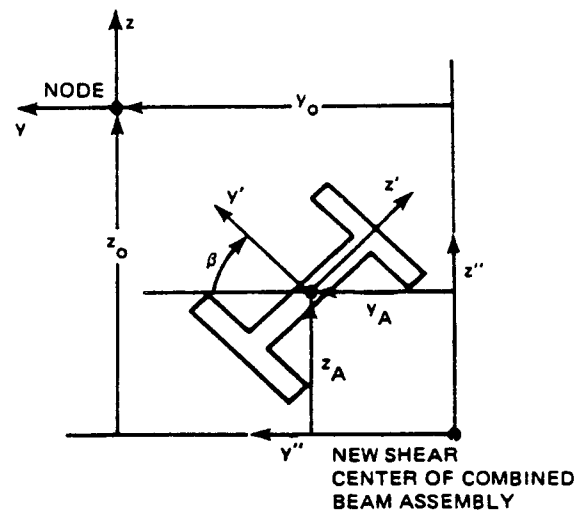Fig 4.3.3 Rotating a Beam
Cross-section

Fig 4.3.4 Rotating and Translating
a Beam Cross-Section as part of a
Combined Beam

4.32

The stiffness properties for an initially straight beam element of arbitrary cross section are based on the assumptions of classical beam theory. These assumptions include:

1) Normals to the centroidal axis remain straight and normal after deformation and their length remains unchanged, i.e., the effect of transverse shear deformation and transverse normal strains are neglected.

2) Longitudinal warping of the cross section out-of-plane is neglected.

3) Transverse deformations within the cross-section are neglected, i.e., cross-sections are rigid (no crippling of thin flanges or webs).

4) Isotropic material, for which $G = E/2(1 + \nu)$ is used in the elastic torsional stiffness. (A special case of a transversely isotropic material is also allowed. See discussion below.)

Although assumption 2) is restrictive for all but circular cross sections, the ability to specify the actual elastic torsional rigidity has been maintained. The basic beam coordinate systems are defined in Fig. 4.3.2. Note that the shear center and centroid do not necessarily coincide, and that the node can be anywhere in the plane. The reference axes at the shear center are parallel to the nodal axes. The geometry is the same at both ends of the beam element. Based on the right-hand rule for moments and rotations, and assumptions 1) and 2) above, the displacements at any point within the beam cross section can be written as

$$u = u_{cg} + (z' - z_{cg}) \beta_y - (y' - y_{cg}) \beta_z \qquad (4.3.9)$$

$$v = v_{sc} - z' \beta_x$$

$$w = w_{sc} + y' \beta_x$$

where u_{cg} is the axial displacement of the centroidal axis; v_{sc}, w_{sc} are the lateral displacements of the shear center in the cross sectional y' and z' directions; β_x, β_y, β_z are the cross sectional rotations about the shear center; and y_{cg}, z_{cg} are the coordinate distances from the shear center to the centroid.

There are four independent displacement components that are prescribed to formulate the beam element stiffness properties. These are the two transverse displacement components v_{sc}, w_{sc} and the axial displacement and twist u_{cg} and β_x. The out-of-plane rotations are related to the transverse displacements in the usual manner by imposing Kirchoff's hypothesis, i.e.,

$$\beta_y = -w_{,x} \quad , \qquad \beta_z = v_{,x} \qquad (4.3.10)$$

The two components of transverse displacement are assumed to be cubic functions that are solutions to the linear homogeneous beam equations while the axial displacement and twist u_{cg}, β_x are linear functions of the axial coordinate.

In the completely elastic case, the beam stiffness matrix involves elastic material properties and integrated quantities that depend on the cross-section, the area, and moments of inertia. Once points on the beam are plastic, these quantities must be numerically integrated. In order to accomplish this, it is necessary to locate "integration points" within the element cross-section. Consequently, the shape of the cross section must be input and the state of stress and strain evaluated at each integration point. Thirteen distinct cross sections can currently be specified. These shapes are discussed in Section 4.3.3.5 below.

Specifically, it is assumed that plastic strains (and therefore plastic material properties) vary linearly between nodes and that the "integration points" are located at Gauss-Legendre integration points in the cross-section at either end. The number and location of the Gauss points for each cross-section are discussed in Section 4.3.3.5 below.

An additional point to be made is that the torsional shearing stresses are neglected in the yield function and the plastic constitutive relations for all the thin walled open sections. Because warping is neglected for the beam sections the numerical integration can over-predict the torsional stiffness. Consequently, these terms are reduced by a "knock down" factor which is the ratio of the input elastic torsional constant to the elastic torsional constant obtained when warping is neglected. Thus, the integrated plastic torsional stiffness is reduced by the same factor that reduces the elastic torsional stiffness.

A transversely isotropic beam is also allowed, for which the transverse (yz) plane is isotropic, but the beam length (x) direction is a principal axis of orthotropy. In that case, E_x is input for E, and the input value of Poisson's ratio should be $\nu = (E_x/2G_{xz})-1$ as a device to have the proper value of G_{xz} calculated internally, for use in the torsional stiffness. However, the isotropic yield criterion will still be applied, so that the plastic behavior of the material will be in error. If such a beam is made of a very brittle material, having little or no plastic strain before failure, then the yield limitation does not apply. A common example of such a brittle elastic material is a resin reinforced by high stiffness uniaxial continuous fibers, such as in a graphite fiber/epoxy resin system.

4.3.3.2 Beam Input Data

The data for the beam element are specified with the following input types:

Group	Keyword	Required	Comments
C	BEAM	Yes	Element identification and connectivity.
H	MBM	Yes	Beam material properties.
	SREC	Yes*	Solid rectangle.
	SCIR	Yes*	Solid circular.
	HREC	Yes*	Hollow rectangular.
	HCIR	Yes*	Hollow circular.
	LSEC	Yes*	L - section.
	TSEC	Yes*	T - section.
	ISEC	Yes*	I - section.
	ZSEC	Yes*	Z - section.
	ZSCR	Yes*	Z-section reserved.
	CSEC	Yes*	Channel - section.
	HSEC	Yes*	Hat - section.
	TWD	Yes*	Planar section.
	LSEG	Yes*	Thin Line Segment.
I	BMLO	No	Linear varying distributed lateral load on a beam element.
	CONC	No	Concentrated forces and moments at nodes.

*Each beam element must be identified with one of the cross sections listed. One specification can be applicable for a number of elements having the same section properties.

4.35

4.3.3.3 Beam Connectivity (BEAM, Group C)

The beam connectivity card begins with the key work BEAM, followed by the beam element number, followed by three node numbers defining node i, node j, and node k in sequence. The use of this three-node sequence is described in the following discussion.

The beam element initial position is described by three distinct node points. The nodal x-axis is defined from node i to node j (Fig. 4.3.2). The nodal y-axis is parallel to the perpendicular from the x-axis through the node k, and the nodal z-axis is perpendicular to the x-y plane. The node k (called the "pointer" node) is used only for the purpose of defining the initial orientation of the beam cross section, and can be an existing node in the structure or a node specified only for this purpose. <u>In the latter case, all degrees of freedom associated with that node must be fixed</u>. Once this additional point is defined, the direction cosine transformations between the nodal and global directions are completely determined. The beam cross-section geometry (shape and position) is specified relative to these nodal axes, as described in general in Section 4.3.3.1 above, and for specific cross-sections in Section 4.3.3.5 below.

```
0........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890123456789 0
BEAM      10  100  105   26
```
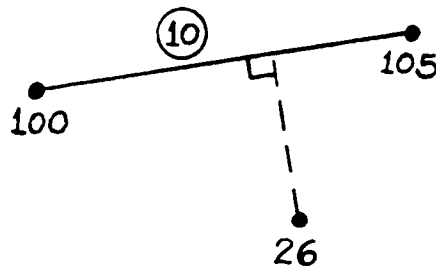
4.3.3.4 Beam Material Properties (MBM, Group H)

The following beam material properties are input.

E — Young's modulus

ANU — Poisson's ratio

PLAS1
PLAS2 — See Table below

YLDST — Yield stress (See Cautionary Note p 4.21)

DENSITY — Mass density (mass/unit volume)

BEPST — Failure strain in tension (+)

BEPSF — Failure strain in compression (-)

Three types of plastic behavior can be considered, perfect plasticity, linear, or nonlinear hardening. They are as follows:

Input Parameters		Types of Plasticity
PLAS1	PLAS2	
0.0	0.0	Perfect plasticity (zero slope in plastic range)
E_t/E	0.0	Linear strain hardening with E_t equal to the slope of the uniaxial stress versus strain curve in the plastic range
n	$\sigma_{0.7}$	Nonlinear strain hardening with Ramberg-Osgood representation of a uniaxial stress-strain curve, see p. 4.19.

Beam material property parameters should ideally be taken from curves of engineering stress versus true strain. Engineering stress should be used because the program does not alter the cross-section area due to Poisson's ratio under axial strains. Using engineering strain will involve strain errors whose magnitude is approximately half the strain (e.g., 10% strain gives an error of 5%), and in many cases this error is acceptable. The material stiffness integrations will be effected only for nonlinear hardening, because the tangent modulus in the plastic range are constant for linear hardening and perfectly plastic materials. Stresses will not be significantly affected by this strain error in the case of perfect plasticity. Therefore, using engineering stress-strain curves should be adequate for most purposes. Tension and compression properties are assumed to be the same, except for failure.

Example 1. Young's Modulus = 10×10^6 psi, Poisson's ratio = 0.25 (shear modulus = 4×10^6 psi)

Perfect Plasticity, No failure strain, density = 2.588×10^{-4} lb $\sec^2/in^4$ (0.1 lb/in^3)

Elements 50 - 100

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
MBM        1.0 E+07    0.25              0.             0.           51000.
           2.588 E-04                0.             0.
      50 -100
```


Example 2.

Same as Example 1, except:

Nonlinear Hardening, n = 10, $\sigma_{0.7}$ = 65,000 psi

Elements 3, 5, 10, 20 -35
```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
MBM        1.0 E+07    0.25             10.         65000.          51000.
           2.588 E-04                0.             0.
      3    5    10    20    -35
```


Example 3.

Same as Example 1, but:

Linear Hardening slope = 2×10^6 psi

Elements 1 - 20

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
MBM        1.0 E+07    0.250            0.2            0.           51000.
           2.588 E-04                0.             0.
       1 -20
```

4.3.3.5 Beam Section Properties (Group H)

The program currently includes thirteen special beam cross sections. Distinct cross sections are required to calculate the extent of plastic behavior within the beam cross section. This is contrary to an elastic analysis where only the integrated section properties, moments of inertia, and area are necessary. Each beam section is specified by a key word. The sections available with their key word and pertinent input quantities are in Table 4.3.1 which starts on p. 4.48 below. For the special cross sections, the moments and product of inertia are calculated and printed out with respect to the <u>section centroidal axes</u> $\bar{y}$ $\bar{z}$ and are defined as (see Fig 4.3.2):

$$I_{\bar{y}} = \int (z' - z_{cg})^2 \, dA \tag{4.3.11}$$

$$I_{\bar{z}} = \int (y' - y_{cg})^2 \, dA$$

$$I_{\overline{yz}} = \int (y' - y_{cg})(z' - z_{cg}) \, dA.$$

Note that in the case where the beam section is only rotated, as in Fig 4.3.3, these geometric quantities are still calculated and printed out as defined above, that is, with respect to the now rotated centroidal or cross-sectional axes. Thus, these values will not change with only a rotation of the section. The program performs the necessary rotational transformations, but these are not printed out. However, when any section is translated using the y_A and z_A in Fig 4.3.4, then all these quantities are calculated with respect to the <u>reference axes</u> y"z", regardless of section shape or whether or not a rotation is specified.

The elastic torsional constant J must be input directly. For calculating the elastic stiffness in torsion, the shear modulus is taken as the isotropic value, $G = E/2(1 + \nu)$. For transverse isotropy, the shear modulus G_{xz} is also calculated internally (Section 4.3.3.1).

In the plastic range, torsional stiffness is initially calculated by numerical integration over the Gauss points, but neglecting the warping function. Thus the result would be similar to $J = I_y + I_z$ which is correct only for the circular cross section. All other sections are less stiff in torsion. To account for this, a torsional stiffness reduction factor is defined from the elastic input quantities as $\eta = J/(I_y + I_z)$ Then, the torsional stiffness terms are always multiplied by η.

The first card of this set begins with one of the beam section key words and contains the cross section properties. If area, moments of inertia, and product of inertia are all input as zero or blank, then these four quantities are calculated internally, based on the section dimensions, by numerical integration using the Gauss integration points for the section (Table 4.3.1 starting on p. 4.48). The numerical integration of moments of inertia neglects the contribution of the moment of inertia of a flange or web component about an axis through its own centroid parallel to its length. This contribution is usually very small for thin walled sections. Thus for thick flanges and webs, where this term is significant, the correct elastic values of I_{yy}, I_{zz}, and I_{yz} should be input directly. This approximation results from using only one integration point across the thickness of flanges and webs and does not apply to solid rectangular or circular cross sections. The input values of these quantities are used only in the elastic range, while the integrated quantities are used in the plastic range. Therefore, a thick-walled section is treated as a thin walled section in the plastic range for calculating the stiffnesses.

If the moments of inertia are input they must be with respect to the proper axes defined by Eq. (4.3.11) or (4.3.12) on p. 4.39 above.

The second card defines eccentricities, y_o, z_o, an orientation angle, β, and two quantities y_A, z_A which define the placement of a beam section relative to the reference axes at the shear center of a combined beam assembly (see Fig 4.3.4).

The nodal eccentricities y_o, z_o are the distances from the cross sectional shear center to the node (see Figs. 4.3.2 through 4.3.4). Eccentricities are the same for each end of the beam. The location of the shear center is shown in Table 4.3.1 for each special section. Note that if a group of beam elements form a combined beam assembly they will all have the same reference axes y"z" (Fig 4.3.4) and the same nodes. Therefore the y_o, z_o inputs will be the same for all beam sections forming the assembly.

The parameters β, y_A, and z_A (called BETA, YA, and ZA in the input) are used to define a new set of reference axes y" z" for a beam element, as shown in Fig. 4.3.4. Fig 4.3.3 shows the use of β alone to rotate a beam element from its originally specified position. In that case, the cross-section and its axes y' z' are rotated through an angle β relative to the stationary y" z" axes which remain parallel to the nodal axes y z. The beam's cross-section stiffness and dimensions (input on the first, third, and subsequent cards of this card set) are unchanged, since they remain assigned to the y' z' axes that rotate with the cross-section. The nodal eccentricities y_o z_o specified on card 2 of the set are also unchanged, since they are now measured from the unmoved y", z" axes. Thus, a beam can be easily rotated about its longitudinal axis to any angle from its original orientation by merely specifying the desired angle β (BETA) on card 2 of the section property input set. The axes y" z" become the new reference axes for the element. This feature can be applied to any beam section, as a simpler alternative to moving its pointer node.

In the more general case, shown in Fig 4.3.4, a beam can be translated as well as rotated relative to the new reference axes y" z". The nodal eccentricities y_o, z_o are now specified relative to the new axes. This feature is used to combine two or more separate beam elements into one single-acting beam. Since each component beam element can have its own material properties (MBM input), this is the way to create a multi-material beam model. The procedure is as follows:

1. Prepare a drawing of the combined cross-section.

2. Locate the node position and the orientation of the nodal axes y z using the pointer (k) node. See Fig 4.3.2.

3. Locate the shear center of the combined cross-section and place the reference axes y" z" there, parallel to the nodal axes. Measure the nodal position (y_0, z_0) from these reference axes.

4. Break up the combined cross-section into sub-sections, according to material type and geometry. Each sub-section must be capable of being defined by one of the cross-section geometries available in Table 4.3.1, and is to be given a separate beam element number.

5. Locate the position and orientation of each sub-section's axes y' z' as shown in Table 4.3.1 starting on p. 4.48. (See Fig 4.3.4 above and Example 4 p. 4.46 below.) Measure the rotational and translational positions (β, y_A, z_A) of each sub-section's y' z' axes from the new reference y" z" axes. Use the right-hand rule for rotation about the beam's longitudinal (x) axis.

6. Prepare a separate BEAM connectivity card (Group C input) for each sub-section beam, having different beam numbers but identical node numbers.

7. Prepare an MBM material properties card set (Group H input) for each separate material used in the combined section. Sub-section beams having the same material can share one MBM set.

8. Prepare a geometric properties card set (Group H input, key words ISEC, TSEC, etc) for each sub-section beam. Note that the nodal eccentricity (y_0, z_0) on card 2 of this set is the same for all sub-sections of a combined beam.

The geometric stiffness quantities A, I_y, I_z, I_{yz}, J will be printed out for each sub-section beam, relative to the new reference axes y" z", and represents each sub-section's contribution to the total section's value of these quantities.

The next card(s) contains cross section dimensions. These are tabulated in Table 4.3.1 and are listed below for each special cross section:

SREC	A	B			Card 3
SCIR	R				Card 3
HCIR	R	T			Card 3
HREC	A1	A2	T1	T2	Card 3
LSEC	A1	A2	T1	T2	Card 3
TSEC	A1	A2	T1	T2	Card 3
ZSEC	A1	A2	A3		Card 3
	T1	T2	T3		Card 4
ZSCR	A1	A2	A3		Card 3
	T1	T2	T3		Card 4
ISEC	A1	A2	A3		Card 3
	T1	T2	T3		Card 4
CSEC	A1	A2	A3		Card 3
	T1	T2	T3		Card 4
HSEC	B	B1	H	T	Card 3
TWD	W	T	NLAY		Card 3
LSEG	Y1	Z1	Y2	Z2	Card 3
	T				Card 4

The two dimensional beam section, TWD, (Table 4.3.1) is used for a beam element restricted to deformation in the local x-z plane defined in Fig. 4.3.2. The input for this section is the width, W, perpendicular to the plane of deformation, the thickness (depth), T, and the number of layers, NLAY, through the thickness. NLAY is used to define Simpson's rule integration points at the boundary of the layers. Up to 58 layers can be specified defining 59 integration points through the thickness. NLAY must be an even number, from 2 to 58.

Example 1.

Solid Rectangular Section
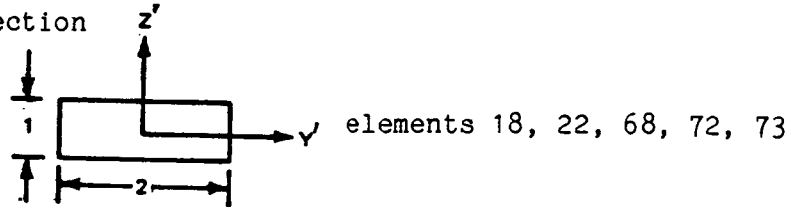
elements 18, 22, 68, 72, 73

```
0.........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890123456789 0
SREC     2.0            0.1667        0.6667        0.           0.458
         0.             0.            0.
         2.0            1.0
    18    22    68    72    73
```

4.43

Example 2.

L-Section

elements 29 - 40,
75-86

```
0.........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
LREC    0.1211          0.01099         0.01099         -0.007087       1.6276 E-04
        0.9375          0.9375          0.
        1.0             1.0             0.0625          0.0625
    29   -40    75    -86
```

C-3

Example 3.

I-Section

elements 1 - 100
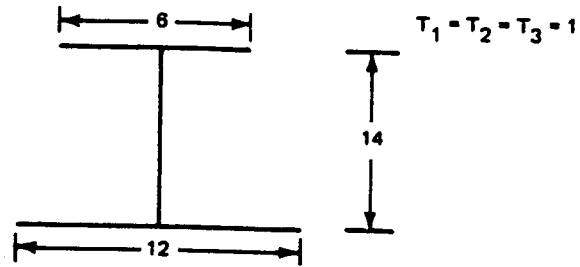
$T_1 = T_2 = T_3 = 1$

dimensions: 6 (top), 14 (height), 12 (bottom)

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
ISEC    30.             855.            163.            0.              10.667
         0.              0.              0.
         6.             14.             12.
         1.              1.              1.
     1 -100
```

4.45

Example 4.

Hollow Rectangular Section

Composed of two LSEC's combined, using the b, Y_A, Z_A inputs.

NOTE: Two beam member cards must be specified in Group C data. The L-sections
      dimensions are:

Section	A1	A2	T1	T2
1	.95	.975	.025	.025
2	.95	.975	.025	.025

The origin (mid thickness at the intersection) of Section 1 is placed at the lower
left corner of the rectangle. The origin of section 2 is placed at the upper right
corner and section 2 is rotated 180°.


```
0........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890123456789 0
LSEC       (four blanks request internal calculation)                 .02317
         0.              0.              0.              -.475         -.475
         .95             .975            0.025           0.025
     1
LSEC       (four blanks request internal calculation)                 .02317
         0.0             0.0             180.            .475          .475
         .95             .975            0.025           0.025
     2
```

TABLE 4.3.1   BEAM SECTION GEOMETRIES

C Section - Key Word CSEC

Input Dimensions: $A_1, A_2, A_3,$
T_1, T_2, T_3

Number of Stress Point: 9
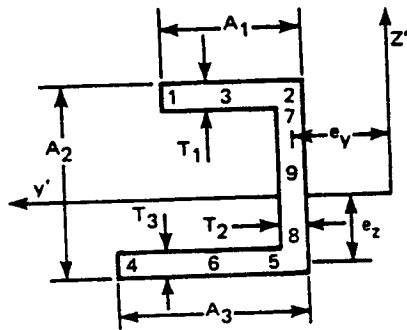
Point No.	Local y', z' coordinates
1	$0.88730\overline{A}_1 + e_y,\quad \overline{A}_2 - e_z$
2	$0.11271\overline{A}_1 + e_y,\quad \overline{A}_2 - e_z$
3	$0.50000\overline{A}_1 + e_y,\quad \overline{A}_2 - e_z$
4	$0.88730\overline{A}_3 + e_y,\quad -e_z$
5	$0.11271\overline{A}_3 + e_y,\quad -e_z$
6	$0.50000\overline{A}_3 + e_y,\quad -e_z$
7	$e_y,\quad 0.88730\overline{A}_2 - e_z$
8	$e_y,\quad -0.11270\overline{A}_2 - e_z$
9	$e_y,\quad 0.5\overline{A}_2 - e_z$

where

$$\overline{A}_1 = A_1 - 0.5T_2, \quad \overline{A}_2 = A_2 - 0.5(T_1 + T_3), \quad \overline{A}_3 = A_3 - 0.5T_2$$

$$e_y = \overline{A}_2{}^2\overline{A}_1{}^2 T_1(2\overline{A}_3 T_3 + \overline{A}_2 T_2)/[4I_y(\overline{A}_1 T_1 + \overline{A}_2 T_2 + \overline{A}_3 T_3)]$$

$$e_z = \overline{A}_2/(1 + I_{z3}/I_{z1})$$

I_y = Moment of inertia about axis parallel to y' through centroid

I_{z1} = Contribution of upper flange to moment of inertia about axis parallel to z' through centroid

I_{z3} = Same as I_{z1}, for lower flange

Shear center at: y' = 0, z' = 0

Centroid at: $y' - e_y = 0.5(\overline{A}_1{}^2 T_1 + \overline{A}_3{}^2 T_3)/(\overline{A}_1 T_1 + \overline{A}_2 T_2 + \overline{A}_3 T_3)$, from middle of web

$z' + e_z = 0.5\overline{A}_2(2\overline{A}_1 T_1 + \overline{A}_2 T_2)/(\overline{A}_1 T_1 + \overline{A}_2 T_2 + \overline{A}_3 T_3)$, from middle of lower flange

TABLE 4.3.1 BEAM SECTION GEOMETRIES (continued)

Hollow Circle - Key Word HCIR

Input Dimensions: R, T

Number of Stress Points: 12
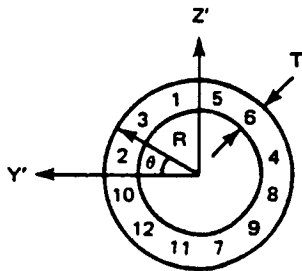
Point No.	Local $\bar{r}$, θ coordinates	
1	$\bar{R}$,	10.14°
2	$\bar{R}$,	79.86°
3	$\bar{R}$,	45.00°

Points in the second through fourth quadrants are located by adding 90°, 180°, 270° to the angles listed above.

stress points
symmetric about y'

where $\bar{R} = R - 0.5T$

Origin at shear center
Centroid at origin: y' = 0, z' = 0

TABLE 4.3.1 BEAM SECTION GEOMETRIES (continued)

Hollow Rectangle - Key Word HREC
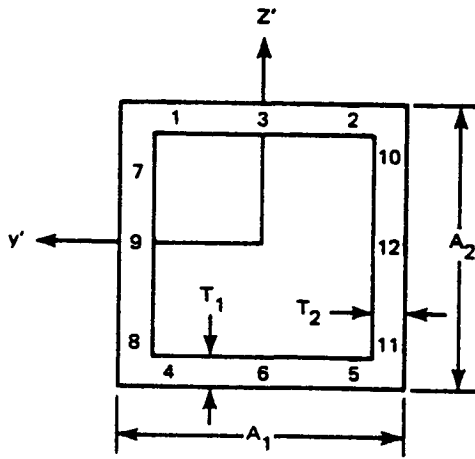
symmetric about y', z'

Input Dimensions: $A_1, A_2, T_1 T_2$

Number of Stress Points: 12

Point No.	Local y', z' Coordinates
1	$0.38730\bar{A}_1$, $0.5A_2$
3	0.0 , $0.5A_2$
7	$0.5A_1$, $0.38730\bar{A}_2$
9	$0.5A_1$, 0.0

where $\bar{A}_1 = A_1 - T_2$,

$\bar{A}_2 = A_2 - T_1$

Origin at shear center
Centroid at origin: y' = 0, z' = 0

TABLE 4.3.1 BEAM SECTION GEOMETRIES (continued)

Hat Section - Key Word HSEC
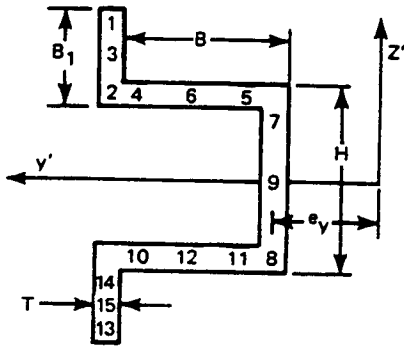
Input Dimensions: B, B_1, H, T

Number of Stress Points: 15

Uniform thickness T,
Symmetry about y'

Point No.	Local y', z' coordinates
1	$e_y + B$, $0.88730B_1 + 0.5H - T$
2	$e_y + B$, $0.11270B_1 + 0.5H - T$
3	$e_y + B$, $0.5B_1 + 0.5H - T$
4	$0.88730B - 0.38730T + e_y$, $0.5(H - T)$
5	$0.11270B + 0.38730T + e_y$, $0.5(H - T)$
6	$0.5B + e_y$, $0.5(H - T)$
7	e_y, $0.38730H$
9	e_y, 0.0

where: $\bar{B} = B - T/2$, $\bar{H} = H - T$

$$e_y = \bar{B}[1 + (\bar{B}/2B_1) - (4/3)(B_1/\bar{H})^2]/[1 + (\bar{H}/6B_1) + (\bar{B}/B_1) + (2B_1/\bar{H})(1 + 2B_1/3\bar{H})]$$

Shear center at: y' = 0, z' = 0

Centroid at: $y' - e_y = 0.5B(B + 2B_1 - T)/(B + B_1 - T + 0.5H)$,   z' = 0

TABLE 4.3.1 BEAM SECTION GEOMETRIES (continued)

I Section - Key Word ISEC

Input Dimensions: A_1, A_2, A_3,
T_1, T_2, T_3

Number of Stress Points: 9
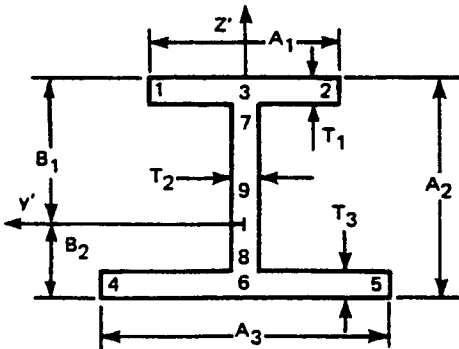
symmetry about z'

Point No.	Local y', z' coordinates
1	$0.38730A_1$, $\overline{B}_1$
3	0.0 , $\overline{B}_1$
4	$0.38730A_3$, $-\overline{B}_2$
6	0.0 , $-\overline{B}_2$
7	0.0 , $0.8873\overline{B}_1 + 0.11271\overline{B}_2$
8	0.0 , $-0.11271\overline{B}_1 - 0.8873\overline{B}_2$
9	0.0 , $0.5(\overline{B}_1 - \overline{B}_2)$

where $\overline{B}_1 = B_1 - 0.5T_1$, $\overline{B}_2 = B_2 - 0.5T_3$

$$\overline{B}_1 = \overline{A}_2/\left[1 + \left(\frac{T_1}{T_3}\right)\left(\frac{A_1}{A_3}\right)^3\right], \quad \overline{B}_2 = \overline{B}_1\left(\frac{T_1}{T_3}\right)\left(\frac{A_1}{A_3}\right)^3, \quad \overline{A}_2 = A_2 - 0.5(T_1 + T_3) = \overline{B}_1 + \overline{B}_2$$

Origin at shear center

Centroid at: $y' = 0$

$$z' = -\overline{B}_2 + [T_1A_1\overline{A}_2 + 0.5T_2(A_2-T_1-T_3)(A_2-T_1)]/[T_1A_1 + T_2(A_2-T_1-T_3) + T_3A_3]$$

4.51

TABLE 4.3.1   BEAM SECTION GEOMETRIES (continued)

L Section - Key Word LSEC

Input Dimensions: A_1, A_2, T_1, T_2

Number of stress points: 6
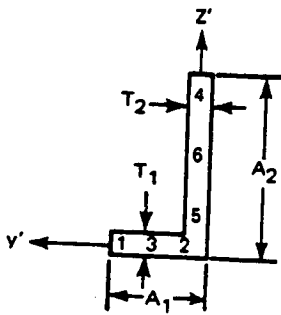
Point No.	Local y', z' coordinate	
1	$0.88730\overline{A}_1,$	0.0
2	$0.11271\overline{A}_1,$	0.0
3	$0.50000\overline{A}_1,$	0.0
4	0.0 ,	$0.88730\overline{A}_2$
5	0.0 ,	$0.11271\overline{A}_2$
6	0.0 ,	$0.50000\overline{A}_2$

where $\overline{A}_1 = A_1 - 0.5T_2$, $\overline{A}_2 = A_2 - 0.5T_1$

Origin at shear center: intersection of flanges

Centroid at: $y' = \dfrac{\overline{A}_1}{2} \cdot \dfrac{T_1\overline{A}_1}{T_1\overline{A}_1 + T_2\overline{A}_2}$, $z' = \dfrac{\overline{A}_2}{2} \cdot \dfrac{T_2\overline{A}_2}{T_1\overline{A}_1 + T_2\overline{A}_2}$

TABLE 4.3.1   BEAM SECTION GEOMETRIES (continued)

Thin Line Segment Beam - LSEG
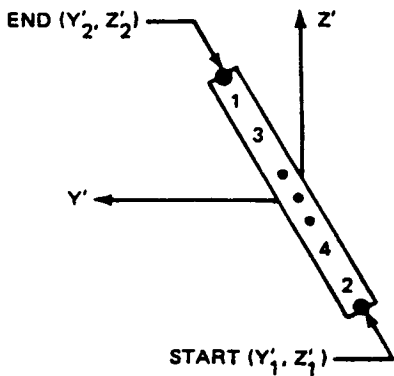
Input Dimensions = Y_1', Z_1', Y_2', Z_2', T.

Number of Stress Points = from 2 to 8.

Location of Stress Points: At Gauss points, numbered in alternating sequence as shown. Locations of points along length depends on number of points used. See a handbook of math function for exact locations.

Origin, shear center, and centroid: at mid-length,

$$Y' = (Y_1' + Y_2')/2$$

$$Z' = (Z_1' + Z_2')/2$$

TABLE 4.3.1 BEAM SECTION GEOMETRIES (continued)

Solid Circle - Key Word SCIR
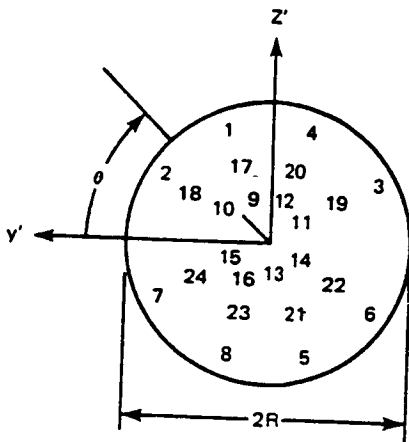
Input Dimension: R
Number of Stress Points: 24

Point No.	Local r, θ coordinates
1	0.88730R, 70.98°
2	0.88710R, 19.02°
9	0.11271R, 19.02°
10	0.11271R, 19.02°
17	0.5R, 70.98°
18	0.5R, 19.02°

Points in second through fourth quadrants
are located by adding 90°, 180°,
270° to the angles listed above.

symmetric about y', z'

Origin at shear center
Centroid at origin: y' = 0, z' = 0

TABLE 4.3.1   BEAM SECTION GEOMETRIES (continued)

Solid Rectangle - Key Word SREC

Input Dimensions: A, B

Number of Stress Points: 16

Point No.	Local y', z' coordinates
1	0.430568A, 0.430568B
3	0.430568A, 0.169991B
9	0.169991A, 0.430568B
11	0.169991A, 0.169991B
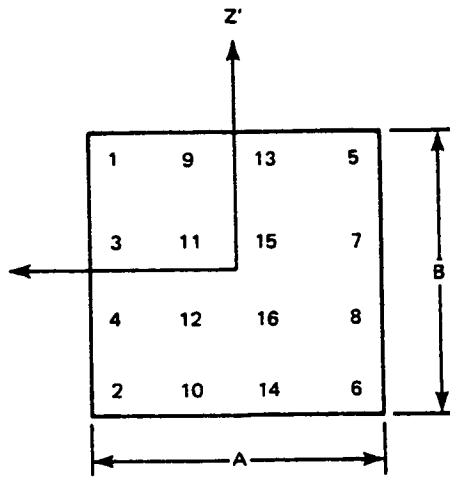
symmetric about y', z'

Origin at shear center
Centroid at orgiin: y' = 0, z' = 0

TABLE 4.3.1   BEAM SECTION GEOMETRIES (continued)

T Section – Key Word TSEC

Input Dimensions: $A_1, A_2, T_1 T_2$

Number of Stress Points: 6
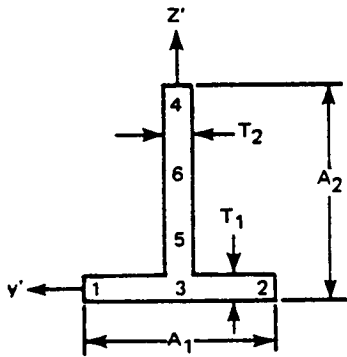
Point No.	Local y', z', coordinate	
1	$0.38730A_1$,	0.0
2	$-0.38730A_1$,	0.0
3	0.0 ,	0.0
4	0.0 ,	$0.88730\bar{A}_2$
5	0.0 ,	$0.11271\bar{A}_2$
6	0.0 ,	$0.50000\bar{A}_2$

where $\bar{A}_2 = A_2 - T_1$

Origin at Shear center: intersection of flanges

$$\text{Centroid at:}\quad y' = 0 , \quad z' = \frac{0.5\bar{A}_2 T_2 A_2}{A_1 T_1 + \bar{A}_2 T_2}$$

4.56

TABLE 4.3.1 BEAM SECTION GEOMETRIES (continued)

Two Dimensional Beam - Key Word TWD

Input Dimensions: W, T, N

Number of Stress Points: from 3 to 59 (odd)

Point locations: evenly spaced from
 upper surface to lower surface, at
 boundaries of an even number N of
 equal thickness layers, from 2 to 58.
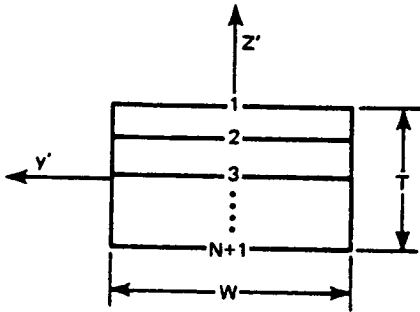
symmetric about y', z'

Origin at shear center
Centroid at origin: y' = 0, z' = 0

TABLE 4.3.1 BEAM SECTION GEOMETRIES (continued)

Z Section reversed - Key Word ZSCR

Input Dimensions: A_1, A_2, T_1, T_2

Number of Stress Points: 9

Equal flanges only
(anti-symmetric about y', z')

Point No.	Local y', z' coordinates)	
1	$0.88730A_1$,	$-0.5\overline{A}_2$
2	$0.11271A_1$,	$-0.5\overline{A}_2$
3	$0.50000A_1$,	$-0.5\overline{A}_2$
7	0.0 ,	$-0.38730\overline{A}_2$
9	0.0 ,	0.0

where $\overline{A}_2 = A_2 - T_1$

Origin at shear center

Centroid at origin: $y' = 0$, $z' = 0$

4.58

TABLE 4.3.1 BEAM SECTION GEOMETRIES (continued)

Z SECTION - KEY WORD ZSEC

Input Dimensions: A_1, A_2, T_1, T_2

Number of Stress Points: 9

Equal flanges only
(anti-symmetric about y', z')

Point No.	Local y', z' coordinates
1	$-0.88730A_1$, $+0.5\overline{A}_2$
2	$-0.11271A_1$, $+0.5\overline{A}_2$
3	$-0.50000A_1$, $+0.5\overline{A}_2$
7	0.0 , $+0.38730\overline{A}_2$
9	0.0 , 0.0

where $\overline{A}_2 = A_2 - T_1$

Origin at shear center
Centroid at origin: $y' = 0$, $z' = o$

4.3.3.6 Beam Loads (Group I)

The following mechanical loads may be applied to the beam element:

CONC   - Concentrated forces and moments applied at specific nodes in the
           global directions.

BMLO   - Distributed line load in the beam local transverse directions.
           A linear variation of the distributed load between nodes is assumed.

These loads are discussed in Section 4.6.

This paragraph shows examples of typical output generated; these are distinct to the beam element.

Beam Connectivity

MEMBER	BEAM ELEMENTS NODE I	NODE J	NODE K
1	100	102	108
2	104	102	108
3	106	104	108
4	108	106	102
5	200	202	208
6	204	202	208
7	206	204	208
8	208	206	202
10	300	304	316
12	308	304	316
14	312	308	316
16	316	312	304
18	400	408	406
21	410	408	406
22	422	410	406
24	414	422	406
25	414	406	422
27	406	400	422
41	6000	108	311
42	208	6000	311
43	6002	208	311
44	316	6002	311
45	6004	316	311
46	414	6004	311
47	8000	316	311
48	8002	8000	311
49	8004	8002	311
50	406	8004	311
51	101	100	108
52	103	101	108
53	105	103	108
54	105	108	101
55	201	200	208
56	203	201	208
57	205	203	208
58	205	208	201
60	303	300	316
63	307	303	316
64	311	307	316
66	311	316	303
68	403	400	406
71	409	403	406
72	411	409	406
73	411	414	406

NOTE: The third node shown is the pointer node that defines the local y direction in the beam cross section.

Beam Material Properties

```
                      BEAM MEMBER MATERIAL PROPERTIES SPECIFICATION  TYPE  4
         YOUNG'S MODULUS  POISSON RATIO  PLAS1          PLAS2           YIELD STRESS
         3.000000E+09     0.000000E+00   0.000000E+00   0.000000E+00    1.000000E+12
                                                           FAILURE STRAINS
         DENSITY                                        TENSION        COMPRESSION
         7.332000E-06                                   0.000000E+00   0.000000E+00
APPLICABLE MEMBERS
    543    801    802    811   -817    525    526
```

The above output is for beam material properties (generated with the key word MBM) for the elements listed below APPLICABLE MEMBERS. The above output specified ideally plastic behavior since both PLAS1 and PLAS2 are zero. Up to 20 distinct beam material properties can be specified using the key word MBM. This number is indicated by the type specification.

Beam Section Properties

```
                            BEAM SECTION PROPERTIES SPECIFICATION TYPE  4      NODAL   OFFSET
                 M O M E N T S   O F   I N E R T I A                          Y0          Z0
        AREA      I-Y          I-Z          I-YZ           J             0.000000E+00  0.000000E+00
   4.474000E-01  4.488818E-02  7.293670E-02  0.000000E+00  7.029300E-03
                              POSITION OF REFERENCE AXIS
                                   YA           ZA
         BETA                  0.000000E+00    0.000000E+00
   0.000000E+00                            T-SECTION
                      A2           A3           T2            T3
                 2.090000E+00  1.000000E+00  9.375000E-02  2.775000E-01
     APPLICABLE MEMBERS
         47    -50
```

The above output is for a T-section. The first line of output contains the beam integrated quantities, the format of which is the same for every beam section. Subsequent lines indicate the particular section as well as the geometric parameters that describe it.

The moments of inertia for the special cross-sections (HREC, etc) Iy, Iz, Iyz are printed with respect to axes at the centroid of the cross-section and parallel to the cross-section axes y', z' shown in Table 4.3.1. If the cross-sectional axes have been moved by using the Y_A, Z_A inputs (see p. 4.40) then the moments of inertia will be printed relative to the new reference axes y" z".

Beam Solution Output

An example of the beam solution output is given below.

```
                    NODAL FORCES AND MOMENTS, STRESSES, STRAINS FOR BEAMS
                    AT TIME = 1.0500000E-03           INCREMENT NUMBER =  27

                                     MEMBER =    84
                                GENERAL CLOSED SECTION
NODE         PLASTIC AND FAIL INDICATORS ( 0=ELASTIC, 1=PLASTIC, -1=UNLOADING, -2=FAILED )
 45    0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
 65    0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
NODE         EPSO           KAPPZY          KAPPYY          KAPPZZ       MAX.STRESS/YIELD   MAX.TENSION      MAX.COMPRESSION
 45     4.8491772E-04  -1.3498106E-04   2.7256417E-06  -5.9901235E-05    (FRACTION)          STRAIN            STRAIN
 65     4.8491772E-04  -1.3498106E-04  -4.7996419E-05  -5.0060592E-05   2.5040767E-01     5.6648043E-04     -1.1540752E-03
NODE         FORCE-X         FORCE-Y         FORCE-Z                      MOMENT-X          MOMENT-Y          MOMENT-Z
 45     4.4793937E+01   4.8838833E+01   2.6969270E+02                  7.1345379E+02     -1.0362652E+04     7.2953986E+03
 65    -4.4793937E+01  -4.8838833E+01  -2.6969270E+02                 -7.1345379E+02     -1.1262942E+04     -3.3792052E+03
                                      S T R E S S                     P L A S T I C   S T R A I N
NODE  PLASTIC    SIG-X         SIG-XY          SIG-XZ         EPS-X         EPS-XY         EPS-XZ        FRACTION OF
 45    NO     7.5985014E+00   3.5295381E-01  -1.2997341E-02  0.0000000E+00  0.0000000E+00  0.0000000E+00  4.9824100E-02   YIELD STRESS
       NO     8.8370947E+00   1.9524191E+01  -2.0594457E-01  0.0000000E+00  0.0000000E+00  0.0000000E+00  2.2845952E-01
       NO     7.7040231E+00   1.9862433E+00  -2.9435439E-02  0.0000000E+00  0.0000000E+00  0.0000000E+00  5.5146537E-02
       NO     8.7315730E+00   1.7890901E+01  -1.8950647E-01  0.0000000E+00  0.0000000E+00  0.0000000E+00  2.1043327E-01
       NO     7.8788791E+00   4.6927050E+00  -5.6674385E-02  0.0000000E+00  0.0000000E+00  0.0000000E+00  7.3989356E-02
       NO     8.5567170E+00   1.5184440E+01  -1.6226753E-01  0.0000000E+00  0.0000000E+00  0.0000000E+00  1.8077515E-01
       NO     8.8995000E+00   8.1075267E+00  -9.1042559E-02  0.0000000E+00  0.0000000E+00  0.0000000E+00  1.0595953E-01
       NO     8.3360962E+00   1.1769618E+01  -1.2789935E-01  0.0000000E+00  0.0000000E+00  0.0000000E+00  1.4395583E-01
       NO    -1.3795251E+01  -1.2565627E+00   1.8313976E+01  0.0000000E+00  0.0000000E+00  0.0000000E+00  2.2652986E-01
       NO     7.1400316E+00  -6.8007764E-02   3.6216591E-01  0.0000000E+00  0.0000000E+00  0.0000000E+00  4.6852954E-02
       NO    -1.2011674E+01  -1.1553040E+00   1.6784576E+01  0.0000000E+00  0.0000000E+00  0.0000000E+00  2.0600686E-01
       NO     5.3564547E+00  -1.6926645E-01   1.8915666E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  4.1083843E-02
       NO    -9.0561766E+00  -9.8751212E-01   1.4250264E+01  0.0000000E+00  0.0000000E+00  0.0000000E+00  1.7220082E-01
       NO     2.4009573E+00  -3.3705836E-01   4.4258781E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  5.2642023E-02
       NO    -5.3271393E+00  -7.7580417E-01   1.1052649E+01  0.0000000E+00  0.0000000E+00  0.0000000E+00  1.3017324E-01
       NO    -1.3280800E+00  -5.4876631E-01   7.6234930E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  8.6960063E-02
 65    NO     7.1115106E+00   3.5295381E-01  -1.2997341E-02  0.0000000E+00  0.0000000E+00  0.0000000E+00  4.6652118E-02
       NO    -1.8003572E-01   1.9524191E+01  -2.0594457E-01  0.0000000E+00  0.0000000E+00  0.0000000E+00  2.5040767E-01
       NO     4.9718364E+00   1.9862433E+00  -2.9435439E-02  0.0000000E+00  0.0000000E+00  0.0000000E+00  3.9518030E-02
       NO    -1.5863898E+01   1.7890901E+01  -1.8950647E-01  0.0000000E+00  0.0000000E+00  0.0000000E+00  2.2754337E-01
       NO     1.4262639E+00   4.6927050E+00  -5.6674385E-02  0.0000000E+00  0.0000000E+00  0.0000000E+00  5.3939710E-02
       NO    -1.2318326E+01   1.5184440E+01  -1.6226753E-01  0.0000000E+00  0.0000000E+00  0.0000000E+00  1.8982645E-01
       NO    -3.0472884E+00   8.1075267E+00  -9.1042559E-02  0.0000000E+00  0.0000000E+00  0.0000000E+00  9.3923820E-02
       NO    -7.8647735E+00   1.1769618E+01  -1.2789935E-01  0.0000000E+00  0.0000000E+00  0.0000000E+00  1.4277138E-01
       NO    -8.5722181E+00  -1.2565627E+00   1.8313976E+01  0.0000000E+00  0.0000000E+00  0.0000000E+00  2.1523273E-01
       NO     7.3025642E+00  -6.8007764E-02   3.6216591E-01  0.0000000E+00  0.0000000E+00  0.0000000E+00  4.7911132E-02
       NO    -7.2197694E+00  -1.1553040E+00   1.6784576E+01  0.0000000E+00  0.0000000E+00  0.0000000E+00  1.9621947E-01
       NO     5.9501155E+00  -1.6926645E-01   1.8915666E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  4.4436706E-02
       NO    -4.9786782E+00  -9.8751212E-01   1.4250264E+01  0.0000000E+00  0.0000000E+00  0.0000000E+00  1.6494957E-01
       NO     3.7090243E+00  -3.3705836E-01   4.4258781E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  5.7790690E-02
       NO    -2.1510280E+00  -7.7580417E-01   1.1052649E+01  0.0000000E+00  0.0000000E+00  0.0000000E+00  1.2621585E-01
       NO     8.8137413E-01  -5.4876631E-01   7.6234930E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00  8.6717321E-02
```

Data are printed for each end node (in the order specificied on the BEAM card; that is, node i then node j) as follows:

o PLASTIC AND FAIL INDICATORS - Status code at each integration point; 0 = elastic, 1 = plastic, -1 = elastic unload from previous plastic state, 2 = failed.

o EPSO - Longitudinal strain; at the centroid for all the special sections, at the shear center for the GSEC, at the origin of the reference axes if the section is part of a combined beam assembly using the YA, ZA inputs.

o KAPPZY - Twist (radian/length) about the x axes.

o KAPPYY - Curvature (radian/length) about the y axes.

o KAPPZZ - Curvature (radian/length) about the z axes.

o MAX. STRESS/YIELD - Maximum longitudinal stress at the integration points divided by the current effective yield stress.

o MAX. TENSION STRAIN - Max. longitudinal tension strain at the integration points.

o MAX. COMPRESSION STRAIN - Max. longitudinal compression strain at the integration points.

4.63

<u>Beam Solution Output</u> (continued from previous page)

      o FORCE X, FORCE Y, FORCE Z - Longitudinal (x) and transverse forces.

      o MOMENT X, MOMENT Y, MOMENT Z - Torsion (about x) and bending moments (about the y and z axes).

The following additional data are printed <u>only</u> if the MSTR card is specified in the Group A input:

      o SIG X, SIG XY, SIG XZ - Longitudinal and shear stresses at each integration point.

      o EPS X, EPS XY, EPS XZ - The plastic part of the Longitudinal and shear strains at each integration point.

Those quantities that depend on the location or orientation of the y and z axes (EPSO, KAPPYY, KAPPZZ, FORCE Y, FORCE Z, MOMENT Y, MOMENT Z, SIGXY, SIGXZ, EPSXY, EPSXZ) are calculated with respect to the centroidal axes.

If the MSTR card is omitted from the Group A input, the short form beam output will be used for all beams, which deletes the stress and strain data at each integration point. However, the maximum stress/yield stress ratio and the maximum strains are given, which should be sufficient in most cases. This default condition will cause the printing of four beam element solutions per page, compared to only one per page if the MSTR card is used, and will greatly reduce the print-out in cases with many beam elements.

In the event that a beam element fails (when all its integration points at either node fail), its solution output is deleted from the print-out.

4.3.4   Membrane Triangles

4.3.4.1   Introduction

The membrane triangles described in these paragraphs are classified into the following three categories:

1)   3-node constant strain

2)   6-node linear strain,

3)   4- and 5- node transitional elements.

The terms constant, linear, and transitional refer to the strain distributions that exist in the element as a consequence of various assumed displacement variations. A brief description of the element associated with these three categories follows.

Constant Strain Triangle (CST) - This well-known plane stress membrane element is one of the most widely used elements for the idealization of membrane structures. Its derivation is based on the assumption of a linear distribution for the in-plane displacements, u and v, and consequently leads to a constant strain state within the element.   Each vertex is allowed three degrees of freedom (the global displace ments u, v, and w) for a total of nine degrees of freedom for the element. Consistent with the total strain distribution, the initial strains (plastic strains) are assumed to be constant within each element.

Linear Strain Triangle (LST) (Fig. 4.3.5) - In regions of high strain gradient, the CST triangle is not sufficiently accurate to be used in a plasticity analysis unless a very fine grid is employed. The linear strain triangle (LST) remedies this shortcoming. The assumption of a quadratic distribution for the in-plane displacements allows for a linear strain variation within the triangle.   Three degrees of freedom at each node (global u, v, w) for each of the six nodes (three vertex and three midside nodes) give this element a total of 18 degrees of freedom. The stresses and strains are evaluated at the centroid of the element.
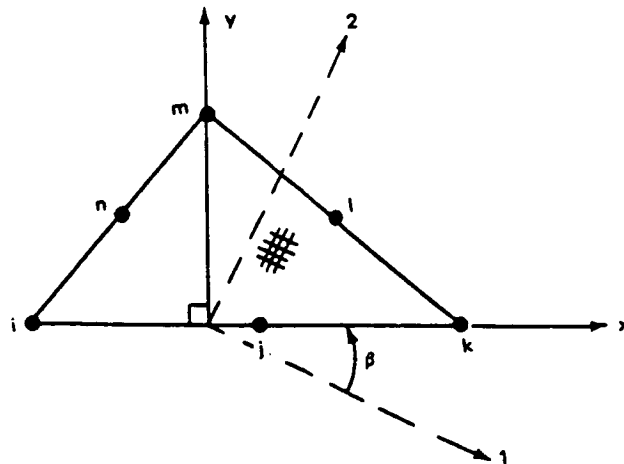
Fig. 4.3.5   Linear Strain Triangle

Transitional or Hybrid Triangles (HST) - In transition regions, i.e., regions in which stresses and strains change from rapidly varying to slowly varying, it becomes convenient and efficient to switch from linear strain triangles to constant strain triangles. This is accomplished by using four- and five-node triangles to maintain compatibility with both the CST and LST elements. For these mixed formulation hybrid elements, the displacements along edges may vary quadratically or linearly, depending on whether an LST or CST triangle is contiguous to the respective sides.

4.3.4.2 Membrane Input Data

The input data for the membrane family of triangular elements are specified with the following input cards:

Group	Keyword	Required	Comments
C	TRIM	Yes	Element identification and connectivity.
H	MAT1 MAT2 MAT3	one required	One of these card types is used to specify material properties.
H	THIK	Yes	Element thickness.
H	MBET	No	Angle in degrees between local x-axis and principal 1-axis for material orthotropy.
I	CONC	No	Concentrated loads at nodes.

4.3.4.3 Membrane Connectivity (TRIM, Group C)

The nodes for the triangular family of elements are specified around the perimeter beginning with a major (vertex) node followed by a minor (midside) node and then major and minor nodes as shown in Fig. 4.3.5. The absence of a minor node must be indicated by a zero or blank field in the proper position. Minor nodes must be midway between the major nodes. However, it is not necessary to manually calculate the coordinates for a midside node, since the program does this calculation automatically and disregards any such coordinates input by the user. In fact, the mid-side nodes can be omitted completely from the Group D (coordinates) input. The local x-axis is defined as the direction of the vector from node i to node k as specified in the Group C input. The local y-axis is the in-plane normal to the x-axis towards node m. The member identification can be any number of up to five digits. These numbers need not be consecutive but must be unique for the model.

Example: Five Node Triangle
```
0.........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890123456789 0
TRIM      27   5   7  10      11  12
```
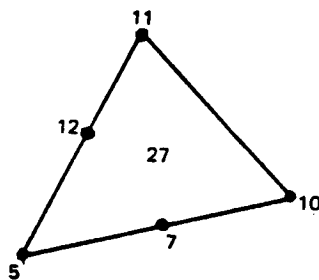
4.3.4.4 Membrane Material Properties (Group H)

All material properties are constant within the element. Three input options are available:

MAT1   -   Material isotropy for elastic and plastic behavior with ideally plastic behavior or linear or nonlinear kinematic hardening.

MAT2   -   Orthotropic, Elastic-ideally plastic behavior.

MAT3   -   Orthotropic, Elastic-linear or nonlinear hardening plasticity.

One of these specifications must be included for each TRIM element. Up to 20 different sets of plane stress material input cards may be used, totalled over MAT1, MAT2, and MAT3 types. Three types of plastic behavior can be considered, as follows:

Input Parameters

PLAS1	PLAS2	Types of Plasticity
0.0	0.0	Perfect plasticity (zero modulus in plastic range)
E_t/E	0.0	Linear strain hardening parameter $= E_t/E$, with E_t equal to the slope of the uniaxial stress versus strain curve in the plastic range
n	$\sigma_{0.7}$	Nonlinear strain hardening with Ramberg-Osgood form of the uniaxial stress-strain curve, see p. 4.19.

These plane stress material property parameters should ideally be taken from curves of engineering stress versus true strain. Engineering stress should be used because the program does not change the thickness due to Poisson's ratio under in-plane strains. Using engineering strain will involve errors whose magnitude is approximately half the strain (e.g., 10% strain gives a 5% error), and in many cases, this error is acceptable. The material stiffness integrations will be affected only for the nonlinear hardening materials, because the tangent moduli in the plastic range are constant for the linear hardening and perfectly plastic materials. Stresses will not be significantly affected by this strain error in the case of perfect plasticity. Therefore, using engineering stress-strain curves should be adequate for most purposes. Tension and compression properties are assumed to be the same, except for failure.

<u>Isotropic Elastic - Plastic Material Properties - MAT1</u>

The keyword for this case is MAT1.

The following material properties are input:

E   Young's modulus

ANU   Poisson's ratio

PLAS1
PLAS2   Described in table, p 4.69.

YLDST   Yield stress (See Cautionary Note p 4.21)

DENSITY   Mass density (mass/unit volume)

EPSFT   Failure strain in tension (+)

EPSFC   Failure strain in compression (-)

Example 1: Modulus = 10×10^6 psi, Poisson's ratio = 0.25, density = 0.1 lb/in^3 (2.588 x 10^{-4} lb sec^2/in^4), ideally plastic, no failure strain, yield = 51,000 psi, elements 1-100.

```
0........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890
MAT1      1.0      E+07      0.25
          2.588    E-04                                  0.0         0.0
          0.0      0.0            51000.
     1 -100
```

Example 2:   Same as example 1, but Nonlinear Hardening, Ramberg-Osgood parameters n = 10, $\sigma_{0.7}$ = 65,000, elements 3, 5, 10, 20 through 35.

```
0.........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890
MAT1      1.0     E+07    0.25
          2.588   E-04
          10.0             65000.          51000.
      3    5    10    20   -35
```

Example 3:   Same as example 1, but Linear Hardening slope = 2 x 10^6 psi, elements 1 through 20.
```
0.........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890
MAT1      1.0     E+07    0.25
          2.588   E-04
          0.2              0.              51000.
      1   20
```


Orthotropic, Elastic - Ideally Plastic - MAT2

The key word for this case is MAT2. This input is used for orthotropic materials with ideally plastic material behavior in each of its principal directions of orthotropy. Hill's yield criterion is used which involves the yield stress in the thickness direction.

The following material properties are input:

EONE, ETWO   Young's moduli along the principal axes of orthotropy, 1 and 2

GONTO   Modulus for shear applied on the 1-2 principal axes of orthotropy.

VONTO   Poisson's ratio defined as $|\varepsilon_{11}/\varepsilon_{22}|$ when ε_{22} is applied uniaxially

DENSITY   Mass density (mass/unit volume)

SIGOX,   Yield stresses along the principal axes of orthotropy, 1, 2, and 3
SIGOY,   (See Cautionary Note p 4.21)
SIGOZ

SIGXY   Yield stress for in-plane shear applied on the 1-2 principal axes of orthotropy

PLAS1
PLAS2   Described in table, p 4.69.

EPSFT1	Failure strain in tension in the 1 direction (+)
EPSFT2	Failure strain in tension in the 2 direction (+)
EPSFC1	Failure strain in compression in the 1 direction (-)
EPSFC2	Failure strain in compression in the 2 direction (-)

Example: Orthotropic material, Density = .1 lb/in^3 (2.588 x 10^{-4} lb sec^2/in^4), E_1 = 10.55 x 10^6 psi, E_2 = 7.61 x 10^6 psi, G_{12} = 4 x 10^6 psi, n_{12} = .33, in-plane normal yield stresses in 1,2 directions = 61,000, 52,000 psi, in-plane shear yield = 31585 psi, yield in thickness direction = 61,000 psi, ideally plastic behavior, no failure, elements 26 through 28 and 74 through 76.

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
MAT2      10.55 E+06    7.616 E+06                    4.0   E+06    0.33
          2.588 E-04                                  0.0           0.0
          0.0           0.0
          61000.        52000.        61000.          31585.
          0.0           0.0
      26  -28   74   -76
```

Orthotropic, Elastic - Strain Hardening Plastic - MAT3

The MAT3 input is used for orthotropic materials with linear and nonlinear hardening behavior. The approach taken to implement this feature is based on the use of Hill's yield criterion and a hardening behavior in the two principal normal directions and in the principal shear direction. Either a Ramberg-Osgood representation for nonlinear hardening behavior or a linear hardening behavior can be used in each principal direction of orthotropy. The first three cards of input for this feature are the same as for MAT2. The fourth and fifth cards contain input for the hardening parameters. None of the components can be specified as ideally plastic although linear and nonlinear plastic behavior can be mixed.

The following material properties are input:

EONE,ETWO Young's moduli along the principal axes of orthotropy

GONTO Modulus for shear applied on the 1-2 principal axes of orthotropy

VONTO Poisson's ratio defined as $|\varepsilon_{11}/\varepsilon_{22}|$ when ε_{22} is applied uniaxially

DENSITY Mass density (mass/unit volume)

SIGOX,
SIGOY, Yield stresses along the principal axes of orthotropy; 1,2,and 3
SIGOZ see cautionary note p 4.21)

SIGXY Yield stress for in-plane shear applied on the 1-2 principal axes of orthotropy (see cautionary note p 4.21)

PLAS1X PLAS1 for 1 axis of orthotropy (see table p 4.69)
PLAS2X PLAS2 for 1 axis of orthotropy (see table p 4.69)

PLAS1Y PLAS1 for 2 axis of orthotropy (see table p 4.69)
PLAS2Y PLAS2 for 2 axis of orthotropy (see table p 4.69)

PLAS1XY PLAS1 for shear on 1-2 axes of orthotropy (see table p 4.69)
PLAS2XY PLAS2 for shear on 1-2 axes of orthotropy (see table p 4.69)

EPSFT1 Failure strain in tension on the 1 axis
EPSFC1 Failure strain in compression on the 1 axis

EPSFT2 Failure strain in tension on the 2 axis
EPSFC2 Failure strain in compression on the 2 axis.

Example: Orthotropic material, Linear hardening in the principal 1 direction, non-linear hardening in the 2 and 1-2 shear directions, elements 1-100.

General: Density = .1 lb/in^3 (2.588 x 10^{-4} lb sec^2/in^4), Poisson's ratio $_{12}$ = 0.3

Normal 1-direction: Modulus = 10.7 x 10^6 psi, yield = 53,500 psi, linear hardening slope = .394 x 10^6 psi, no failure

Normal 2-direction: Modulus = 21.4 x 10^6 psi, yield = 32,000 psi, nonlinear hardening n = 10, $\sigma_{0.7}$ = 46,600 psi, no failure

Normal 3-direction: Yield = 53,500 psi

Shear 1-2-direction: Modulus = 3.923 x 10^6 psi, yield = 22,000 psi, nonlinear hardening n = 9.675, $\sigma_{0.7}$ = 28,800 psi, no failure

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
MAT3     1.07   E+07    2.14   E+07                   3.923  E+06    0.3
         2.588  E-04                                  0.0            0.0
         0.0            0.0
         5.35   E+04    3.20   E+04    5.35   E+04    2.2    E+04
         3.68   E-02    0.0            10.0           4.66   E+04    9.675
         2.88   E+04
         1- 100
```

4.3.4.5 Membrane Element Thickness (THIK, Group H)

A value of the thickness must be input for each plane stress element. A set of two cards can cover a group of elements having the same thickness.

For example, the input for a model with 100 TRIM elements with elements 1 through 50 having a thickness of 0.025 and 51 through 100 having a thickness of 0.05 is as follows:

```
0........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890123456 7890
THIK      0.025
   1  -50
THIK      0.05
  51 -100
```

4.3.4.6 Orientation of Axes of Material Anisotropy (MBET, Group H)

The element stiffness matrices are calculated with respect to an element local axis system referred to as x, y in Fig. 4.3.5. For this purpose orthotropic material properties must be transformed from the material's principal orthotropic axis system, shown as 1,2 in Fig. 4.3.5, to the element local coordinate axis system through the angle β, the angle between the x and 1 axes, according to the transformation;

$$
\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{Bmatrix} = \begin{bmatrix} Q'_{11} & Q'_{12} & Q'_{16} \\ Q'_{12} & Q'_{22} & Q'_{26} \\ Q'_{16} & Q'_{26} & Q'_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_{xy} \end{Bmatrix} \tag{4.3.13}
$$

where

$$
Q'_{11} = Q_{11}\cos^4\beta + 2(Q_{12} + 2Q_{66}) \sin^2\beta \cos^2\beta + Q_{22} \sin^4\beta
$$

$$
Q'_{22} = Q_{11}\sin^4\beta + 2(Q_{12} + 2Q_{66}) \sin^2\beta \cos^2\beta + Q_{22} \sin^4\beta
$$

$$
Q'_{12} = (Q_{11} + Q_{22} - 4Q_{66}) \sin^2\beta \cos^2\beta + Q_{12}(\sin^4\beta + \cos^4\beta)
$$

$$
Q'_{66} = (Q_{11} + Q_{22} - 2Q_{12} - 2Q_{66}) \sin^2\beta \cos^2\beta + Q_{66}(\sin^4\beta + \cos^4\beta) \tag{4.3.14}
$$

$$
Q'_{16} = (Q_{11} - Q_{12} - 2Q_{66}) \sin\beta\cos^3\beta + (Q_{12} - Q_{22} + 2Q_{66}) \sin^3\beta \cos\beta
$$

$$
Q'_{26} = (Q_{11} - Q_{12} - 2Q_{66}) \sin^3\beta\cos\beta + (Q_{12} - Q_{22} + 2Q_{66}) \sin\beta \cos^3\beta
$$

and

$$
\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_{12} \end{Bmatrix} = \begin{bmatrix} Q_{11} & Q_{12} & 0 \\ Q_{12} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_{12} \end{Bmatrix} \tag{4.3.15}
$$

All the stress and strain calculations are calculated in the material system and then transformed into the element local system for the output. It should be noted that the angle β is constant for each element and is not updated due to geometry changes during the course of the analysis.

If the body is isotropic or if the local and material axes coincide, then β is taken as zero by default and this input is not necessary. Note that β is from the 1-axis to the x-axis, with the sign convention determined by a right-hand rotation about the z-axis.

As an example, if the angle between the local x and the material 1 axes is 45 degrees for elements 1, 3, 5 ... 21 then the input is:

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
MBET      45.0
      1   -2   -21
```

4.3.4.7 Membrane Loads (Group I)

The following mechanical loads may be applied to the membrane elements:

CONC - Concentrated forces and moments applied at specified nodes in the global directions in the units of force or force times length.

These are discussed in Section 4.6.

4.3.4.8 Membrane Output

This section shows examples of typical output generated for the triangular membrane element.

Membrane Connectivity

MEMBER	TRIANGULAR MEMBRANE ELEMENTS NODE I	NODE J	NODE K	NODE L	NODE M	NODE N
5	11		75		120	
6	75		74		120	
7	74		73		120	
8	73		86		120	
9	76		23		92	
10	76		92		141	
11	74		75		22	
12	92		22		75	
13	69		70		17	
14	27		17		70	
15	19		18		27	
16	27		70		19	
17	71		19		70	
18	72		20		71	
19	19		71		20	
20	12		76		141	
21	75		11		12	
22	73		74		22	
23	21		73		22	
24	16		32		15	
25	14		15		32	
26	15		90		16	
27	91		16		90	
28	5		91		4	
29	90		4		91	
30	15		14		89	
31	89		90		15	
32	3		4		90	
33	90		89		3	
34	88		2		3	
35	3		89		88	
36	32		88		89	
37	89		14		32	
38	91		5		6	
39	6		87		91	
40	16		91		87	
41	87		72		16	
42	71		16		72	
43	16		71		70	
44	32		16		70	
45	70		69		32	
46	69		84		32	

NOTE: The nodes connecting the element are printed in a counterclockwise order, i.e, first a major (vertex) node followed by a minor (midside) node, if present, in the absence of a midside node the column for the midside node is blank.

Membrane Element Material Properties

```
                    MAT1 MATERIAL PROPERTIES SPECIFICATION TYPE  1
      YOUNG'S MODULUS  POISSON RATIO                    PLAS1           PLAS2
       1.1200000E+07  3.3300000E-01              0.0000000E+00  0.0000000E+00
                                                         FAILURE STRAIN
        YIELD STRESS     DENSITY                    TENSION     COMPRESSION
       5.0130000E+04  2.5880000E-04              0.0000000E+00  0.0000000E+00
APPLICABLE MEMBERS
       1001 -1026   2001 -2026   3001 -3026
```

This output indicates ideally plastic behavior (since both PLAS1 and PLAS2 are zero) for an isotropic material with no failure (infinite strain capability). This printout is generated when specifying material properties with a MAT1 card.

Membrane Element Thickness

```
                                   THICKNESS TYPE   1 IS  2.5000000E-02
                                   THE NUMBER OF LAYERS IS   2
        APPLICABLE MEMBERS
            1001 -1026   2001 -2026   3001 -3026
```

The above example is for the specification of a thickness of .025. The number of layers is ignored for membrane elements (TRIM), and defaults to 2 layers if not specified on the THIK card.

Membrane Solution Output

The output as shown below gives the plasticity indicator, the stresses, total strains, and plastic strains in the local axis system (Fig 4.3.5), the ratio of maximum principal stress divided by the current effective yield stress, and the number of subincrements used in satisfying the constitutive laws. All these data are calculated for the single integration point at the centroid of the triangular element. A plasticity indicator of NO indicates an elastic element, YES indicates a plastic element, and UNL indicates an elastic unloading from a previously plastic state. A failed element is deleted from this print-out.

STRESSES AND STRAINS FOR TRIANGULAR MEMBRANES
AT TIME = 4.0000000E-03   INCREMENT NUMBER = 80

MEMBER	PLASTIC	STRESS X	Y	XY	TOTAL STRAIN X	Y	XY	PLASTIC STRAIN X	Y	XY
5	NO	-1.230E+03	-8.503E+02	-6.186E+01	-3.250E-05	-1.604E-05	-5.361E-06	0.000E+00	0.000E+00	0.000E+00
6	NO	-2.191E+03	-3.031E+03	1.305E+03	-4.273E-05	-7.911E-05	1.131E-04	0.000E+00	0.000E+00	0.000E+00
7	NO	4.040E+02	-2.235E+03	8.015E+02	3.581E-06	-7.853E-05	6.946E-05	0.000E+00	0.000E+00	0.000E+00
8	NO	-3.647E+03	-2.886E+03	7.383E+01	-9.271E-05	-5.972E-05	6.399E-06	0.000E+00	0.000E+00	0.000E+00
9	NO	2.124E+02	-2.506E+03	-3.519E+02	3.214E-05	-8.566E-05	-3.050E-05	0.000E+00	0.000E+00	0.000E+00
10	NO	-2.426E+03	5.607E+02	1.445E+02	-8.647E-05	4.295E-05	1.252E-05	0.000E+00	0.000E+00	0.000E+00
11	NO	-1.410E+03	-4.259E+02	3.273E+03	-4.273E-05	-1.007E-07	2.836E-04	0.000E+00	0.000E+00	0.000E+00
12	NO	-3.139E+03	-8.998E+02	2.144E+03	-9.564E-05	1.398E-06	1.858E-04	0.000E+00	0.000E+00	0.000E+00
13	NO	-6.094E+03	-1.825E+03	-2.058E+03	-1.849E-04	1.183E-07	-1.783E-04	0.000E+00	0.000E+00	0.000E+00
14	NO	-3.303E+03	-9.928E+02	3.316E+02	-1.002E-04	-6.635E-08	2.874E-05	0.000E+00	0.000E+00	0.000E+00
15	NO	1.348E+03	-5.932E+02	-6.417E+03	5.088E-05	-3.326E-05	-5.562E-04	0.000E+00	0.000E+00	0.000E+00
16	NO	-2.761E+03	-9.198E+03	5.994E+02	-6.697E-08	-2.790E-04	5.195E-05	0.000E+00	0.000E+00	0.000E+00
17	NO	4.224E+03	-7.625E+03	7.773E+02	2.171E-04	-2.964E-04	6.737E-05	0.000E+00	0.000E+00	0.000E+00
18	NO	2.479E+02	-6.837E+03	-1.122E+02	7.664E-05	-2.304E-04	-9.722E-06	0.000E+00	0.000E+00	0.000E+00
19	NO	4.241E+03	-7.569E+03	-1.489E+03	2.171E-04	-2.947E-04	-1.290E-04	0.000E+00	0.000E+00	0.000E+00
20	NO	2.000E+01	-3.444E+03	-2.212E+03	3.511E-05	-1.150E-04	-1.917E-04	0.000E+00	0.000E+00	0.000E+00
21	NO	-2.287E+03	-4.374E+03	-1.196E+03	-3.250E-05	-1.229E-04	-1.036E-04	0.000E+00	0.000E+00	0.000E+00
22	NO	1.180E+03	3.506E+02	2.781E+03	3.581E-05	-1.106E-07	2.410E-04	0.000E+00	0.000E+00	0.000E+00
23	NO	-2.102E+03	-4.143E+03	-6.691E+03	-2.863E-05	-1.171E-04	-5.798E-04	0.000E+00	0.000E+00	0.000E+00
24	NO	-2.432E+03	8.461E+02	3.478E+02	-8.954E-05	5.253E-05	3.014E-05	0.000E+00	0.000E+00	0.000E+00
25	NO	-1.020E+03	-4.205E+02	1.112E+03	-2.978E-05	-3.819E-06	9.641E-05	0.000E+00	0.000E+00	0.000E+00
26	NO	2.254E+02	-9.207E+02	2.748E+00	4.672E-05	-3.294E-05	2.382E-07	0.000E+00	0.000E+00	0.000E+00
27	NO	1.872E+03	4.938E+02	1.024E+03	5.746E-05	-2.261E-06	8.878E-05	0.000E+00	0.000E+00	0.000E+00
28	NO	5.567E+03	-8.814E+02	1.726E+03	1.944E-04	-8.505E-05	1.495E-04	0.000E+00	0.000E+00	0.000E+00
29	NO	5.804E+02	1.110E+02	-1.264E+03	1.824E-05	-2.103E-06	-1.096E-04	0.000E+00	0.000E+00	0.000E+00
30	NO	-3.738E+02	1.732E+03	-3.107E+02	-2.978E-05	6.148E-05	-2.692E-05	0.000E+00	0.000E+00	0.000E+00
31	NO	-5.497E+02	3.366E+02	6.484E+02	-2.169E-05	1.672E-05	5.620E-05	0.000E+00	0.000E+00	0.000E+00
32	NO	3.775E+02	6.602E+02	9.663E+02	5.980E-06	1.823E-05	8.374E-05	0.000E+00	0.000E+00	0.000E+00
33	NO	3.047E+02	3.184E+03	-7.795E+02	-2.169E-05	1.031E-04	-6.755E-05	0.000E+00	0.000E+00	0.000E+00
34	NO	8.424E+02	-1.034E+03	-2.970E+03	-2.969E-05	-1.187E-04	-2.573E-04	0.000E+00	0.000E+00	0.000E+00
35	NO	3.043E+03	-1.661E+02	2.970E+03	2.912E-04	-3.597E-05	2.574E-04	0.000E+00	0.000E+00	0.000E+00
36	NO	5.375E+03	5.342E+02	-1.726E+03	1.738E-04	-3.595E-05	-1.496E-04	0.000E+00	0.000E+00	0.000E+00
37	NO	1.989E+03	4.818E+02	1.891E+03	6.148E-05	-3.830E-05	1.639E-04	0.000E+00	0.000E+00	0.000E+00
38	NO	5.212E+03	-2.066E+03	-4.426E+03	1.944E-04	-1.210E-04	-3.836E-04	0.000E+00	0.000E+00	0.000E+00
39	NO	2.725E+03	1.671E+02	-2.539E+03	8.917E-05	-2.168E-05	-2.200E-04	0.000E+00	0.000E+00	0.000E+00
40	NO	2.400E+03	2.253E+03	-8.165E+02	5.746E-05	5.111E-05	-7.077E-05	0.000E+00	0.000E+00	0.000E+00
41	NO	2.098E+03	-4.829E+03	-2.838E+01	1.382E-04	-2.486E-04	-2.459E-06	0.000E+00	0.000E+00	0.000E+00
42	NO	4.447E+03	-6.115E+03	3.086E+02	2.094E-04	-2.483E-04	2.675E-05	0.000E+00	0.000E+00	0.000E+00
43	NO	5.323E+03	-3.194E+03	-4.588E+03	2.094E-04	-1.597E-04	-3.976E-04	0.000E+00	0.000E+00	0.000E+00
44	NO	-4.043E+03	-4.525E+03	2.855E+03	-8.954E-05	-1.104E-04	2.474E-04	0.000E+00	0.000E+00	0.000E+00
45	NO	-3.391E+03	7.186E+03	9.499E+02	-1.849E-04	2.734E-04	8.233E-05	0.000E+00	0.000E+00	0.000E+00
46	NO	1.195E+04	7.237E+02	-3.319E+03	3.909E-04	-9.533E-05	-2.876E-04	0.000E+00	0.000E+00	0.000E+00
47	NO	7.357E+03	-1.161E+03	1.614E+03	2.568E-04	-1.123E-04	1.399E-04	0.000E+00	0.000E+00	0.000E+00
48	NO	5.268E+03	1.781E+02	-2.833E+02	1.738E-04	-4.675E-05	-2.455E-05	0.000E+00	0.000E+00	0.000E+00
49	NO	7.137E+03	5.976E+02	2.431E+03	2.319E-04	-5.145E-05	2.107E-04	0.000E+00	0.000E+00	0.000E+00
50	NO	-1.545E+03	8.270E+03	-3.623E+03	-1.342E-04	2.911E-04	-3.140E-04	0.000E+00	0.000E+00	0.000E+00

4.3.5 Nonlinear Spring Element

4.3.5.1 Introduction

This element acts as a one degree of freedom extensional or rotary nonlinear spring with damping as shown in Fig. 4.3.6. It can be used as follows:

o   to simulate structural components or assemblies for which the load versus deflection (or moment versus rotation) behavior has been obtained either by a crush test or by some other means

o   to simulate an energy absorbing device

o   as a gap element that has zero force in some deflection range and otherwise rigid

o   for combinations of the above.

The formulation of this element is similar to that of an axial force stringer with the spring properties being specified in tabular form, with viscous damping (proportional to velocity). A number of different spring types are available. These are: a general extensional spring between any two nodes; a globally-oriented extensional or rotary spring between any two nodes whose axis remains parallel to one of the global axes; a grounded globally oriented extensional or rotary spring, with one node on the structure, the other node at the origin of global coordinates, whose axis remains parallel to one of the global axes. The globally-oriented springs use displacements or rotations which are projected onto a line of action between the two end nodes that is parallel to one of the global axes. The grounded global spring is equivalent to a spring oriented parallel to one of the global axes, with one end at a structural node, whose other end slides freely along the plane through the global origin that is normal to its axis.

AXIAL SPRING   ROTARY SPRING

$$F_x = C (\delta_j - \delta_i)$$

$$M = C (\theta_j - \theta_i)$$

θ ALWAYS GLOBAL

"NODE" k = 0: STRAIGHT LINE TO NODES   "NODE" k = 4, 5, 6 $(\theta_x, \theta_y, \theta_z)$

"NODE" k = 1, 2, 3: GLOBAL COMPONENT ONLY   + SENSE OF M DEPENDS ON
(u, v, w)   SEQUENCE OF NODES i, j
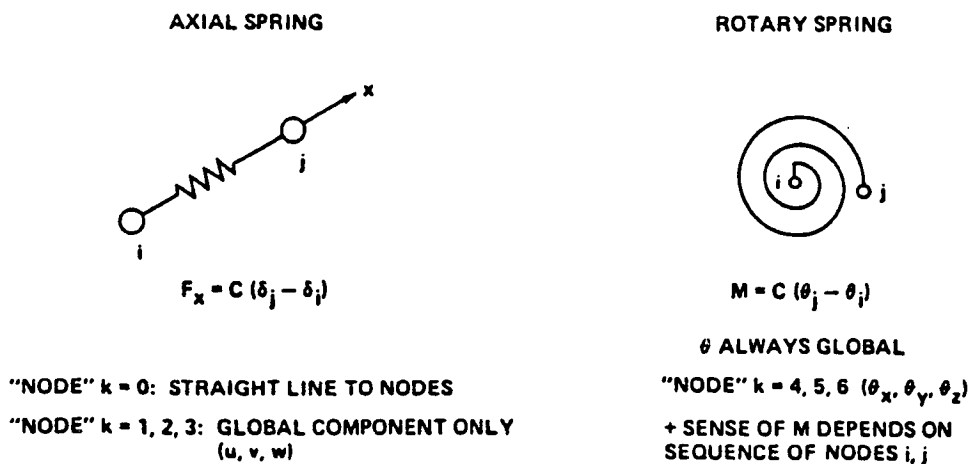
Fig. 4.3.6   Nonlinear Spring Elements

4.3.5.2   Spring Input Data

The data for the nonlinear spring element are specified with the following cards:

Group	Keyword	Required	Comments
C	SPNG	Yes	Element identification and connectivity.
H	PSPR	Yes	Element mechanical properties.
I	CONC	No	Concentrated forces at node.

4.3.5.3 Spring Connectivity (SPNG, Group C)

The nonlinear spring connectivity is specified with a unique member identification, MEM, followed by the two end point nodes, NODEI, NODEJ and an input variable that defines the spring type, NODEK. The member and node identification can be any number of up to five digits. A number of different spring types are available and are determined by the input data on the connectivity card. The various options are described in the table below.

Input Parameters			
NODEI	NODEJ	NODEK	COMMENTS
>0	>0	0	Extensional spring between nodes i and j, that changes orientation as the nodes move.
>0	>0	1 2 3	Global extensional spring between nodes i and j, that remains in the global x, y, or z direction respectively.
>0	>0	4 5 6	Global rotary spring between nodes i and j, that remains oriented to rotate about the global x, y, or z axes, respectively.
>0	0	1 2 3 4 5 6	Grounded global extensional spring oriented parallel to the global x, y, z axes (NODEK = 1,2,3), or grounded global rotary spring rotating about the global x,y,z axes (NODEK = 4,5,6). One end attached to node i, the other to the origin of global coordinates.

For the extensional springs, a positive extension is defined as one which results from an increased distance between its nodes. However, the positive sense of the global rotational spring is under the control of the user. The rotational spring moments are proportional to the relative global rotation, $\theta_j - \theta_i$. Thus, the sense of positive relative rotations and moments depends on the i, j order of the nodes as specified in Group C and the direction of positive rotation for the spring's global axis. Care must be taken to specify moment-rotation curves in Group H that are compatible in the positive/negative sense with the sense of the relative rotation defined by the connectivity input in Group C.

To avoid numerical problems, the extensional springs should have a finite initial length (no matter how small); and should not be allowed to turn inside-out. This latter condition will occur when the two nodes approach and pass through the same point while moving in opposite directions. This event is usually physically unrealistic and can be avoided by use of a gap-type force-displacement curve.

Example: Two translational springs in series
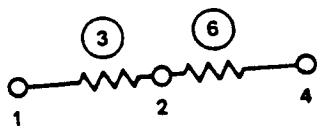

```
0........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890123456789 0
SPNG        3    1    2    0
SPNG        6    2    4    0
```

Example 2:   Global translational spring in global x direction
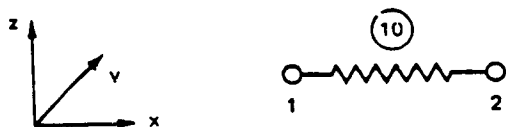

```
0........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890123456789 0
SPNG       10    1    2    1
```

Example 3: Torsional spring about the global x axis

```
0........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890123456789 0

SPNG        5    1    2    4
```


4.84
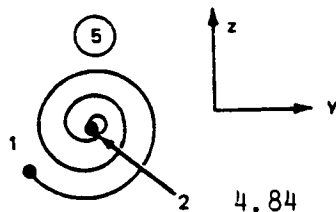
4.3.5.4 Spring Stiffness Properties (PSPR, Group H)

The spring stiffness properties are obtained as the slope of a multilinear force versus elongation or moment versus rotation curves as shown in Fig. 4.3.7.

This table is specified in input Group H as input pairs of force (moment) and elongation (rotation). The necessary input for each table are:

NSPRG   The number of data points in the table; up to 20 points are allowed per table.

USPRG   Slope used for unloading - optional.

FAILT   Failure force in tension (+) - optional.

FAILC   Failure force in compression (-) - optional.

DMASS   Characteristic mass used to compute the damping coefficient-optional.

FORCE   Force (or moment) at the associated ELONG.

ELONG   Relative elongation (or rotation) of the nodes at the associated FORCE, $u_j - u_i$ (or $\theta_j - \theta_i$).

If the elongation exceeds the last tabular input point, the last slope (either 1-2 or 6-7 in Fig. 4.3.7) is continued until failure.
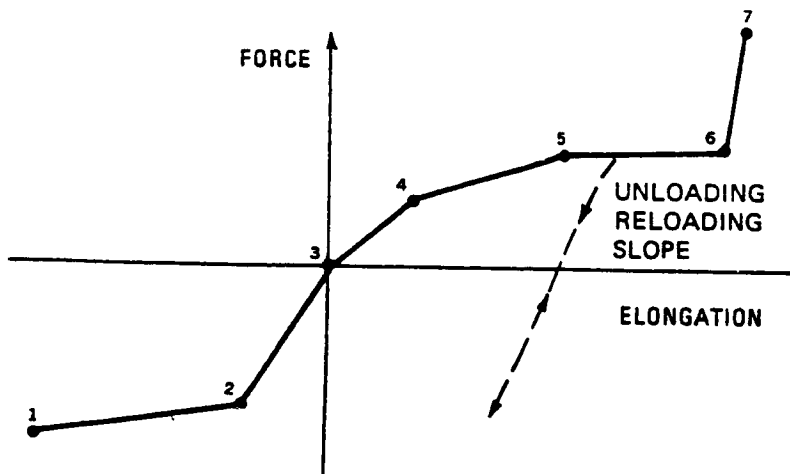

Fig. 4.3.7   Force versus Elongation for Nonlinear Spring Element

The card input is in the following form;

CARD 1 contains the key word PSPR followed by the number of points in the table (NSPRG), the unloading slope, the failure force in tension and compression and the characteristic damping mass. A set of cards (of quantity = NSPRG) follow containing the force, elongation pairs. The specification ends with a list of applicable members.

The input must be specified in the order of monotonic increasing elongation (from negative to positive values, as shown by the integers on Fig. 4.3.7). If it is not, an error message will be printed and the job will be aborted. In addition, one pair must include the zero elongation point (not necessarily at zero force).

Note that the positive sense of the rotational springs are controlled by the user in the connectivity input Group C (Section 4.3.5.3 above). The curves input here should be compatible with the positive/negative sign convention as defined there, for each rotary spring.

Unloading can occur in two ways. If the unloading slope has been input as zero (or blank) then the spring unloads along its original force-displacement path (conservative or elastic unload). If a finite slope has been input then unloading occurs along the linear path such as shown by the dotted line in Fig. 4.3.7. Subsequent reloading returns along the linear unloading slope, then rejoins and follows the nonlinear curve. This element then has nonrecoverable or dissipative energy.

Positive and negative failure elongation (or rotation) can be specified, which causes deletion of element stiffness and force (or moment) as for other elements (Sect. 4.1.7), when these failure values are exceeded.

More complicated behavior can be developed by placing two or more nonlinear springs in parallel or in series. However, take care to avoid massless nodes in dynamic problems, or nodes without lateral restraint in static problems.

The damping force is proportional to, and opposes, the relative velocity of the element's end nodes. The proportionality factor is not constant, but is dependent on the current stiffness of the element, according to

$$C = 2 \sqrt{km} \qquad (4.3.16)$$

where k is the absolute value of the current spring stiffness and m is a characteristic mass (DMASS) specified in the input. Equation (4.3.16) represents the critical damping factor for a one-degree of freedom system having the stiffness k and mass m. Thus a part of the damping factor in a nonlinear spring element is set by the user when specifying the characteristic mass DMASS in Group H, and a part is controlled by the program in accordance with the changing element stiffness. This was done to help control the behavior of gap elements, as discussed next. Damping is not operational with the explicit time integrators ADAM and CDIF.
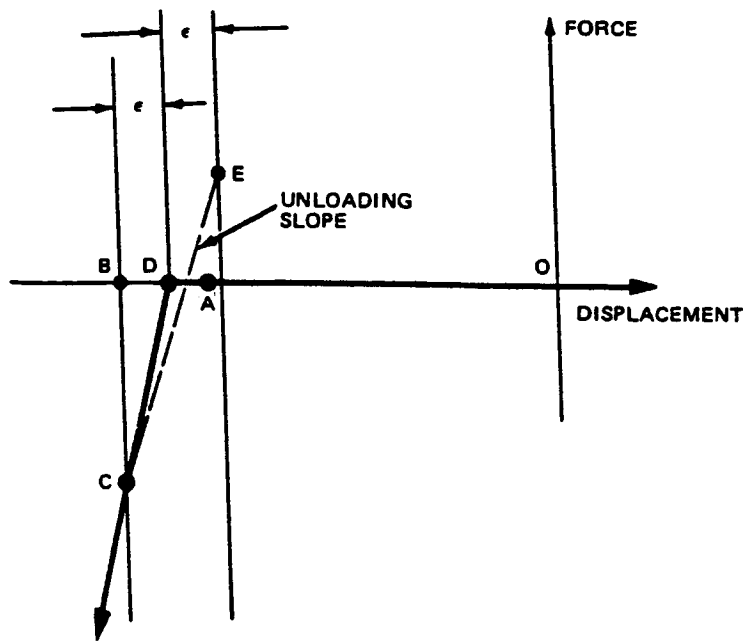
Fig. 4.3.8 Nominal Gap Spring Behavior

The nonlinear spring element can be used as a gap element to simulate the kinematic constraints associated with contact and rebound such as occurs at impact with external barriers or between parts of the structure. As shown in Fig 4.3.8, the spring stiffness would change from zero in the gap region to a large stiffness to simulate a "rigid" contact. This stiffness should be in the order of magnitude of 10^2 or 10^3 times greater than the effective structural stiffness at the contact node, to be sufficiently "rigid" but still not too stiff to cause certain numerical problems. However, using a sudden change in stiffness can still cause dynamic difficulties. In Fig 4.3.8, it can be seen that as the gap is closed, the spring displacement proceeds in one time step from point A to point B. At the end of that step, the spring force is found to be at point C. If the spring stiffness and the displacement increment are large enough, an unrealistically large force could suddenly be imposed on the structure. This arbitrarily large force could cause false failures in the surrounding structure and could result in the rapid repulsion of the spring nodes from each other back into the gap zone. In many problems, the nodes may then be returned toward each other again by external forces or inertias. This can result in a series of rapid repeated contacts and rebounds, that can cause numerical difficulties, and is not realistic.

A procedure that usually works well to prevent this problem, is to increase the stiffness from zero by increments over a small distance, with large damping. This will allow the spring force to increase to a more realistic value without overshoot, and prevent the artificial contact-rebound oscillations, while allowing the spring to rebound back into the gap region under the action of steady external tension forces. This has worked well for simulating the impact and subsequent rebound of structures with rigid barriers and with other moving bodies. The procedure is as follows (see Fig 4.3.9):

4.87

1. Estimate the largest possible displacement increment imposed on the spring by the impact speed and the time step at the instant of gap closure. This can be easily done in most cases, by using the product of the initial impact speed and the maximum allowed time step (for example; 14 m/sec x 50 x 10^{-6} sec = 7 x 10^{-4}m = 0.7mm, therefore use 1 mm).

2. Define the value of the "rigid" barrier stiffness at 10^2 to 10^3 times the stiffness of the surrounding structure (10^6-10^7 lb/in. or 10^8-10^9 Newtons/meter have worked well for "rigid" stiffnesses in some autos and aircraft in past cases).

3. Increase the stiffness from zero to the "rigid" value in 3 or 4 jumps (for example; 0, 10^6, 10^7, 10^8, 10^9 N/m as in Fig. 4.3.9). Use the displacement increment found in step 1 above for each segment to insure that at least one point on each stiffness segment will be used.

4. Specify large damping by using a damping mass (DMASS) that is very large compared to the effective mass at the spring nodes (100 times will usually work well). If one node is fixed, then use the mass of the movable node as the effective mass. If both are movable, use the smaller of the two as the effective mass.

5. Do not use an unloading slope (set USPRG=0). The spring will then be nonlinearly elastic and will not permanently dissipate energy.

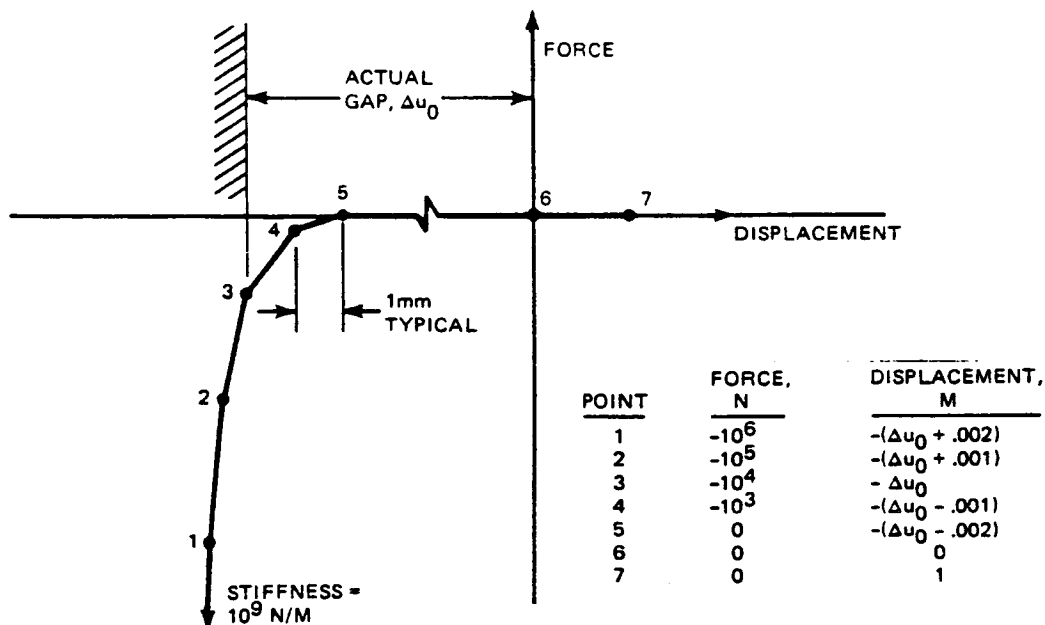| POINT	FORCE, N	DISPLACEMENT, M
1	-10^6	$-(\Delta u_0 + .002)$
2	-10^5	$-(\Delta u_0 + .001)$
3	-10^4	$- \Delta u_0$
4	-10^3	$-(\Delta u_0 - .001)$
5	0	$-(\Delta u_0 - .002)$
6	0	0
7	0	1

Fig 4.3.9 Gap Spring Boundary Layer Example

In this method, the "rigid" barrier will usually be penetrated some acceptable small distance and the spring compression will respond smoothly to the external forces on it by increasing and decreasing in a controlled way. Note that, in Fig 4.3.9, the acutal barrier is in the middle of the boundary layer, which begins 2 mm before the actual gap is closed. This was done to minimize the penetration, but the user can place the actual gap anywhere in the boundary layer with little error, assuming a small boundary layer and a large gap.

The program also has a built-in procedure for controlling the contact/rebound dynamics, involving an automatic "capture" mechanism. This works by creating a "proportional band" around the gap closure point, and is activated simply by specifying a finite unloading slope in step 5 above. The program will then create a proportional band of displacement width 2ε where ε is the initial penetration distance DB in Fig. 4.3.8. The gap spring will unload from point C with a stiffness equal to the unloading slope. Oscillations with damping will proceed between points C and E. When the displacements would rebound past E, the force would drop to zero and the node is released to rebound freely into the gap zone, and it can return to start the contact process once again. The proportional band 2ε is automatic and not under the user's direct control. The unloading slope should be large (10^9 N/m for the example used above), and the boundary layer (steps 1-4 above) should still be used.

The preferred method is to use the boundary layer without the proportional band "capture" because the user has greater control over the contact behavior.

Example: a nonlinear crush spring with permanent deformation (energy dissipation) in compression, and elastic in tension.

Given a table with 5 points for members 1 through 10 and 53

FORCE	ELONGATION
-1000	-.2
-1000	-.1
0.0	.0
1000	.1
2000	.2

Unloading slope is 1×10^4 (equal to the tension slope)
No failure criterion or damping

```
0.........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890123456789 0
PSPR      5.0            1.  E+04    0.0             0.0             0.0
PSPR     -1000.0         -.2
PSPR     -1000.0         -.1
PSPR      0.0            .0
PSPR      1000.0         .1
PSPR      2000.0         .2
     1  -10   53
```

4.89

4.3.5.5 Loads (Group I)

The following mechanical loads may be applied to the spring element:

CONC - Concentrated forces applied at specified nodes in the global directions in the units of force.

In addition, a table of the time distribution of the load, (load magnitude versus time), must be specified in the PTME, PTM2, or PTM3 input in Group I.

These loads are discussed in Section 4.6

4.3.5.6 Description of Output

This section shows examples of typical output generated when using the spring element.

Element Connectivity

MEMBER	SPRING ELEMENTS NODE I	NODE J	DIRECTION
901	133	125	1
902	133	126	1
903	133	105	1
905	133	100	1
906	133	99	1
907	133	97	1
908	133	31	1
909	133	82	1
910	133	96	1
911	133	98	1
912	133	26	1
913	133	13	1
914	133	45	1
915	133	10	1
916	133	95	1
918	133	121	1
919	133	132	1
920	133	122	1
921	8	97	
922	78	132	1
923	83	78	
924	113	114	3
925	115	116	3

The values in the last column indicate the direction of global springs.

Element Stiffness Properties

The example here is an energy-dissipating crush spring with a varying force and a maximum crush of 6.2 inches. The large unloading slope causes permanent deformation with little spring-back. Damping must not be used when there is a negative slope as in this case. A gap spring can be in parallel to also limit the crush.

```
                 SPRING MATERIAL PROPERTY TABLE  4
            UNLOADING                FAILURE FORCES
             MODULUS          TENSION      COMPRESSION    DAMPING MASS
           1.0000000E+06    0.0000000E+00  0.0000000E+00  0.0000000E+00

           FORCE(MOMENT)                  DISPLACEMENT(ROTATION)
           -1.585000E+05                   -6.200000E+00
           -5.850000E+04                   -6.100000E+00
           -8.500000E+03                   -6.000000E+00
           -6.400000E+03                   -4.250000E+00
           -8.503000E+03                   -3.620000E+00
           -2.800000E+03                   -2.500000E+00
           -5.531000E+03                   -1.750000E+00
           -2.200000E+03                   -1.150000E+00
           -4.600000E+03                   -5.000000E-01
           -7.589000E+03                   -2.500000E-01
            0.000000E+00                    0.000000E+00
```

APPLICABLE MEMBERS
    923

Force - Deflection Output

The output shown below is for a series of spring elements. The stiffness force comes from the input curve of force vs displacement (or moment vs rotation), while the damping force is calculated using Eq. 4.18 on p. 4.88. The Total force is the sum of these two forces. The load state will be LOADING whenever the relative velocity of its nodes is zero or is in the same direction as it experienced in the first time increment, and UNLOADING when the velocity is in the opposite direction.

```
                    ELEMENT FORCES AND ELONGATION FOR NONLINEAR SPRING MEMBERS
                AT TIME = 2.0000000E-03              INCREMENT NUMBER = 40

                                    STIFFNESS      DAMPING        TOTAL
MEMBER  NODE I  NODE J  LOAD STATE   FORCE          FORCE          FORCE         ELONGATION       LENGTH
 901     133     125    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0542330E+00   1.0772767E+01
 902     133     126    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0545435E+00   1.0772457E+01
 903     133     105    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0465750E+00   1.0780425E+01
 905     133     100    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0511169E+00   1.0657863E+01
 906     133      99    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0444565E+00   1.0664543E+01
 907     133      97    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0504163E+00   1.0776584E+01
 908     133      31    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0450536E+00   1.0781946E+01
 909     133      82    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0444234E+00   1.0782577E+01
 910     133      96    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0569256E+00   1.0770074E+01
 911     133      98    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0407565E+00   1.0766244E+01
 912     133      26    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0433634E+00   1.0783637E+01
 913     133      13    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0464067E+00   1.0786593E+01
 914     133      45    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0494269E+00   1.0777573E+01
 915     133      10    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0440764E+00   1.0782924E+01
 916     133      95    LOADING    0.0000000E+00  0.0000000E+00  0.0000000E+00  -1.0547331E+00   1.0772267E+01
 918     133     121    LOADING   -1.0075313E+02  -1.4742916E+03  -1.5750448E+03  -1.0075313E-02   5.9899247E+00
 919     133     132    LOADING   -1.3197024E+02  -1.5153446E+03  -1.6473149E+03  -1.3197024E-02   5.9868030E+00
 920     133     122    LOADING   -1.2985876E+02  -1.5126397E+03  -1.6424984E+03  -1.2985876E-02   5.9870141E+00
 921       8      97    LOADING    0.0000000E+00   0.0000000E+00  0.0000000E+00  -2.6074018E-03   2.9052393E+01
 922      78     132    LOADING   -4.9288666E-03   0.0000000E+00  -4.9288666E-03  -1.0195888E-02   4.8074112E+00
 923      83      78    LOADING    4.8921902E-02   0.0000000E+00   4.8921902E-02  -1.0089176E-02   8.2829111E+00
 924     113     114    LOADING   -1.4389397E-02   0.0000000E+00  -1.4389397E-02  -1.2733979E-04   1.9999873E-01
 925     115     116    LOADING    2.9891276E-01   0.0000000E+00   2.9891276E-01   2.6452457E-03   2.0002645E-01
```

4.92

4.3.6 Plate Bending Element - TRP2

4.3.6.1 Introduction

TRP2 is a triangular plate bending element formulated for use in nonlinear dynamics problems. The need for computational economy dominated the formulation of this element in the sense that the "best" simplest element was sought. Numerical evaluation has shown the TRP2 element to possess more than sufficient accuracy for engineering computations, while its relative simplicity leads to less costly computations. Nevertheless, it is still much more costly to use (from 9 to 27 times) than a membrane triangular element. Consequently the user is advised to use membrane elements to model for the essential features of the structural response wherever possible.

In the derivation* of this element the in-plane displacements, u and v are taken to vary linearly. This leads to constant strain membrane behavior. The transverse displacement, w, is cubic. For convenience transverse shear deformation is postulated during the derivation. The rotations of the plate normal are given a quadratic distribution throughout the element, and the transverse displacement is assumed to vary quadratically along the edges. Additionally, the thin plate Kirchhoff hypotheses are imposed at the vertices and at the midsides. These midside points are used only as a convenience in the derivation and are not nodes in the final formulation. What results from these discrete point Kirchhoff assumptions, and an additional assumption on the rotation of the plate normal along the element edges, is an element in which the Kirchhoff hypotheses hold along the edges as well as the vertices. In its final formulation the TRP2 element is the three node triangular plate bending element shown in Fig. (4.3.10). In global coordinates three translational and three rotational degrees of freedom may be imposed at each node.
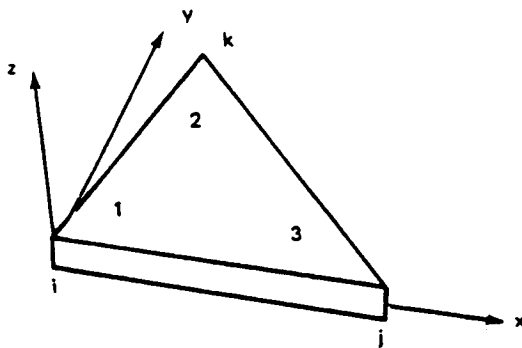
Three Gauss integration points in the plane, from three to nine Simpson (including surfaces) integration points throughout the element thickness at each Gauss point.

Fig. 4.3.10 Triangular Plate Element - TRP2

---

*See: H. Garnet and A.B. Pifko, "An Efficient Triangular Plate Bending Element For Crash Simulation", in Advances and Trends In Structural and Solid Mechanics, Ed. by A.K. Noor and J. Housner, Pergamon Press, New York, 1983, pp. 371-379 (Also Computers& Structures Vol. 16, No. 1-4, pp. 371-379, 1983)

4.3.6.2 Input Data

The input data for the triangular bending elements TRP2 are specified with the following input cards:

Group	Keyword	Required	Comments
C	TRP2	Yes	Element identification and connectivity.
H	MAT1	Yes	This card type is used to specify material properties.
H	THIK	Yes	Element thickness and number of layers through the thickness for Simpson's rule integration.
I	CONC	No	Concentrated loads at nodes.
	SURF	No	Distributed surface loads.

4.3.6.3 Element Connectivity (TRP2, Group C)

The node numbers for the TRP2 element are identified with the vertices of each triangle. The member identification can be any number up to five digits. These numbers need not be consecutive, but must be unique for the job. The local x-axis is defined as the direction of the vector from node i to node j as specified in group C input (Fig. 4.3.10).

Example:
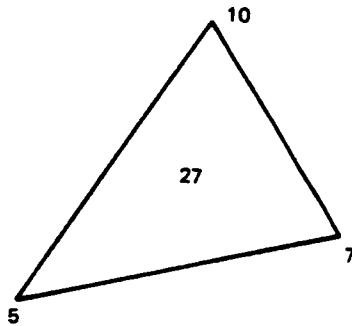

```
0........1.........2.........3.........4.........5.........6.........7.........8
123456789012345678901234567890123456789012345678901234567890123456789012345678 90
TRIM        27    5    7    10
```

4.3.6.4 Plate Material Properties (Group H, MAT1)

All material properties are constant within the element and are assumed to be isotropic. These are input on MAT1 cards:

MAT1 - Material isotropy for elastic and plastic behavior with ideally plastic behavior or linear or nonlinear kinematic hardening.

One of these specifications must be included for each TRP2 element. Three types of plastic behavior can be considered, as follows:

Input Parameters		
PLAS1	PLAS2	Types of Plasticity
0.0	0.0	Perfect plasticity (zero modulus in plastic range)
E_t/E	0.0	Linear strain hardening parameter, E_t/E, with E_t equal to the slope of the uniaxial stress versus strain curve in the plastic range
n	$\sigma_{0.7}$	Nonlinear strain hardening with Ramberg-Osgood form of the of a uniaxial stress-strain curve, see p. 4.19.

The plane stress material property parameters should ideally be taken from curves of engineering stress versus true strain. Engineering stress should be used because the program does not change the thickness due to Poisson's ratio under in-plane strains. Using engineering strain will involve errors whose magnitude is approximately half the strain (e.g., 10% strain gives an error of 5%), and in many cases this error is acceptable. The material stiffness integrations will be affected only for nonlinear hardening, because the tangent moduli in the plastic range are constant for linear hardening and perfectly plastic materials. Stresses will not be significantly affected by this strain error in the case of perfect plasticity. Therefore, using engineering stress-strain curves should be adequate for most purposes. Tension and compression properties are assumed to be the same.

Isotropic Elastic - Plastic Material Properties - MAT1

The keyword for this case is MAT1.

The following material properties are input:

E   Young's modulus

ANU   Poisson's ratio

PLAS1
PLAS2   Described in table, p 4.96

YLDST   Yield stress (See Cautionary Note p 4.21)

DENSITY   Mass density (mass/unit volume)

EPSFT   No failure criterion currently available

EPSFC   No failure criterion currently available

Example 1: Modulus = 10×10^6 psi, Poisson's ratio = 0.25, density = 0.1 lb/in^3 (2.588×10^{-4} lb sec^2/in^4), ideally plastic, no failure strain, yield = 51,000 psi, elements 1-100.

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
MAT1      1.0     E+07     0.25
          2.588   E-04                                         0.0         0.0
          0.0             0.0             51000.
      1 -100
```

Example 2:   Same as example 1, but Nonlinear Hardening, Ramberg-Osgood parameters
n = 10, $\sigma_{0.7}$ = 65,000, elements 3, 5, 10, 20 through 35.


```
0........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890123456789 0
MAT1      1.0    E+07    0.25
          2.588  E-04
          10.0             65000.          51000.
      3    5    10    20   -35
```


Example 3:   Same as example 1, but Linear Hardening slope = 2 x 10^6 psi, elements
1 through 20.


```
0........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890123456789 0
MAT1      1.0    E+07    0.25
          2.588  E-04
          0.2              0.              51000.
      1   -20
```

4.3.6.5 Element Thickness and Number of Integration Layers (THIK, Group H)

A value of the thickness and NLAY, the number of layers through the thickness for Simpson's rule integration must be input for each element. However, one THIK specification can cover a group of elements having the same thickness and NLAY. The TRP2 element uses numerical integration in the plane and through the thickness to evaluate element matrices and stress resultants. Three Gauss integration points in the element planform are used along with Simpson's rule integration through the thickness at each of these points. Up to eight layers defining nine points are allowed at each of the Gaussian integration points, with a minimum of two being required. Thus, from nine to twenty-seven stress recovery points can be used. The choice of the number of points is governed by the desired accuracy in tracing an elastic-plastic boundary through the thickness of the element.

Example: Given a model with 100 TRP2 elements with elements 1 through 50 having a thickness of 0.025 and 51 through 100 having a thickness of 0.05. Six layers through the thickness is used throughout. Note: An even number must be input for NLAY.

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
THIK      0.025             6.0
   1  -50
THIK      0.05              6.0
  51 -100
```

4.3.6.6 Loads (Group I)

The following mechanical loads may be applied to the TRP2 elements:

CONC - Concentrated forces and moments applied at specified nodes in the global directions in the units of force or force times length.

SURF - Distributed surface load on the element lateral surface. A linear variation of the distributed load between nodes is assumed.

These are discussed in Section 4.6.

4.3.6.7 Output

This section shows examples of typical output generated for the TRP2 element.

Element Connectivity

TRIANGULAR PLATE ELEMENTS (TRP2)

MEMBER	NODE I	NODE J	NODE K
209	100	134	131
210	42	94	56
211	56	55	42
212	41	42	55
213	55	54	41
214	40	41	53
215	53	41	54
216	129	53	52
217	52	130	129
218	103	129	130
219	130	104	103
220	93	103	104
221	104	17	93
222	39	40	103
223	53	129	40
224	129	103	40
225	44	131	134
226	134	57	44
227	103	93	39

Element Material Properties

MAT1 MATERIAL PROPERTIES SPECIFICATION TYPE 1

YOUNG'S MODULUS	POISSON RATIO	PLAS1	PLAS2
3.0000000E+07	3.0000000E-01	1.3820000E-03	0.0000000E+00

FAILURE STRAIN

YIELD STRESS	DENSITY	TENSION	COMPRESSION
2.3980000E+04	7.3320000E-04	3.0000000E-01	-1.0000000E+00

APPLICABLE MEMBERS
    1   -86   241   -256   271   -276

4.101

Element Thickness

APPLICABLE MEMBERS
  1   -30

THICKNESS TYPE   1 IS   2.4990000D+00
THE NUMBER OF LAYERS IS   8

Stress and Strain Output

The output below is typical for each TRP2 element for which output is desired. First are the local coordinates of three Gauss points (Fig 4.3.10), followed by the plate stress resultants, N_x, N_y, N_z, M_x, M_y, M_{xy} and membrane strains and curvatures, e_x, e_y, e_{xy}, k_x, k_y, k_{xy} at each Gaussian integration point. These data are followed by a list of each Simpson integration point through the thickness, including the top and bottom surfaces at each Gauss point, with the ratio of the Von Mises stress to the current effective yield stress, and a status indicator. A stress ratio less than one indicates that the point is elastic having never been plastic, with an indicator of NO, or elastically unloaded from a previous plastic state, indicated by UN. If a point is plastic the indicator is YES. The upper and lower surface total strains and stresses follow. If all the points in the element are elastic this completes the output. If any point is or previously was plastic, the stresses and plastic strains for each point follows, from the upper to the lower surface for the three Gauss points in turn. In this particular example there are 8 layers so that there are a total of 27 stress points. (9 Simpson points through the thickness times 3 Gauss points along the plane)

BYCAST EXAMPLE PROBLEM   IMPULSIVELY LOADED PLATE CLAMPED (101981) (011282.PM)   PAGE 127

STRESSES AND STRAINS FOR TRIANGULAR BENDING ELEMENT (TRP2EL)
AT TIME = 3.800000D-04   MEMBER = 1

INTEG. POINT	LOCAL COORDINATES X	Y	MX	MY	MXY	MX	MY	MXY
1	4.242476D+00	5.789247D-01	-4.108825D+01	1.204919D+02	5.946661D+01	7.723697D+00	3.740962D+01	1.503505D-01
2	1.074719D+01	7.206920D+00	-1.250505D+02	-1.191471D+02	-2.376508D+01	6.344673D+01	1.203730D+02	-4.865647D+01
3	1.077585D+01	5.789247D-01	-1.577335D+01	9.378031D+01	3.218580D+01	1.773350D+02	7.019760D+01	-3.806516D+01

STRESS RESULTANTS

INTEG. POINT	MEMBRANE STRAIN EX	EY	EXY	KAP-X	KAP-Y	KAP-XY
1	1.590148D-02	4.281731D-02	-4.270248D-03	1.346657D-03	2.344930D-02	1.295234D-04
2	1.590148D-02	4.281731D-02	-4.270248D-03	3.410456D-04	2.223194D-02	-3.018165D-05
3	1.590148D-02	4.281731D-02	-4.270248D-03	5.099512D-04	2.347966D-02	-3.407079D-05

CURVATURES

INTEG. POINT	THICKNESS COORDINATE	PLASTIC	FRACTION OF YIELD STRESS	INTEG. POINT	PLASTIC	FRACTION OF YIELD STRESS	INTEG. POINT	PLASTIC	FRACTION OF YIELD STRESS
1	1.249500D+00	UN	1.197583D-01	2	UN	7.194587D-01	3	UN	4.261063D-01
	9.371250D-01	UN	6.848804D-01		UN	6.245786D-01		UN	4.704446D-01
	6.247500D-01	UN	2.959823D-01		UN	7.737645D-01		UN	1.437641D-01
	3.123750D-01	UN	3.971142D-01		UN	7.595823D-01		UN	2.379603D-01
	0.0	UN	8.604031D-01		UN	8.604031D-01		UN	8.604031D-01
	-3.123750D-01	UN	7.783900D-01		UN	8.658778D-01		UN	8.861994D-01
	-6.247500D-01	UN	3.557092D-01		UN	8.709420D-01		UN	6.564487D-01
	-9.371250D-01	UN	5.578384D-01		UN	4.581847D-01		UN	8.484665D-01
	-1.249500D+00	UN	5.309439D-01		UN	5.543334D-01		UN	7.903592D-01

STRESS AND TOTAL STRAIN AT UPPER SURFACE

INTEG. POINT	TOT-X	TOT-Y	TOT-XY	SIG-X	SIG-Y	SIG-XY
1	1.758438D-02	7.211721D-02	-4.108408D-03	9.235447D+01	1.281467D+02	1.093819D+01
2	1.632762D-02	7.059612D-02	-4.307960D-03	6.231240D+01	2.547878D+02	-4.265470D+01
3	1.653866D-02	7.215514D-02	-4.312819D-03	1.807632D+02	1.344183D+02	-3.248046D+00

STRESS AND TOTAL STRAIN AT LOWER SURFACE

INTEG. POINT	TOT-X	TOT-Y	TOT-XY	SIG-X	SIG-Y	SIG-XY
1	1.421858D-02	1.351741D-02	-4.432087D-03	-6.171170D+00	1.259995D+02	1.664531D+01
2	1.547534D-02	1.503851D-02	-4.232516D-03	-9.550780D+01	1.473184D+01	2.546598D+01
3	1.524430D-02	1.347948D-02	-4.227676D-03	-1.036570D+02	9.817419D+01	2.188990D+01

STRESS   PLASTIC STRAIN

STRESS POINT	PLASTIC	SIG-X	SIG-Y	SIG-XY	EPS-X	EPS-Y	EPS-XY
1- 1	UN	9.235447D+01	1.281467D+02	1.093819D+01	1.732394D-02	7.163199D-02	-4.245796D-03
1- 2	UN	-4.676973D+01	1.651286D+02	1.220415D+01	1.762891D-02	6.392673D-02	-4.302157D-03
1- 3	UN	1.800134D+00	9.833380D+01	1.118239D+01	1.686750D-02	5.699683D-02	-4.332295D-03
1- 4	UN	1.383858D+01	4.055170D+01	4.597679D+01	1.631412D-02	4.996644D-02	-4.807274D-03
1- 5	UN	-4.047471D+01	-1.292435D+02	5.569786D+01	1.606432D-02	4.335403D-02	-4.969834D-03
1- 6	UN	-4.179797D+01	-1.617030D+02	1.940870D+01	1.547441D-02	3.612598D-02	-4.554488D-03
1- 7	UN	-9.359355D+00	9.025217D+01	1.311391D+01	1.523605D-02	2.771780D-02	-4.515883D-03
1- 8	UN	-1.145100D+01	1.341112D+02	1.819309D+01	1.480890D-02	2.017791D-02	-4.620150D-03
1- 9	UN	-4.171170D+00	1.259995D+02	1.664531D+01	1.431000D-02	1.289978D-02	-4.641159D-03
2- 1	UN	6.231240D+01	2.547878D+02	-4.265470D+01	1.639585D-02	6.945556D-02	-3.772200D-03
2- 2	UN	-1.117627D+01	1.419678D+02	-5.838230D+01	1.648367D-02	6.294853D-02	-3.565228D-03
2- 3	UN	-4.477588D+01	-5.787555D+01	-7.184719D+01	1.624698D-02	5.692141D-02	-3.386675D-03
2- 4	UN	-1.344648D+01	-1.183037D+02	-1.710340D+01	1.590161D-02	5.031401D-02	-4.048651D-03
2- 5	UN	-6.047471D+01	-1.292435D+02	5.569786D+01	1.606320D-02	4.335403D-02	-4.969834D-03
2- 6	UN	-8.876104D+01	-1.624103D+02	2.840688D+01	1.598340D-02	3.657846D-02	-4.597952D-03
2- 7	UN	-8.269764D+01	-1.807414D+02	-6.085376D+00	1.582597D-02	2.968120D-02	-4.174957D-03
2- 8	UN	-8.397455D+01	-3.075090D+01	7.000342D+00	1.594299D-02	2.201006D-02	-4.329891D-03
2- 9	UN	-9.550738D+01	1.473184D+01	2.545980D+01	1.595808D-02	1.482892D-02	-4.552398D-03
3- 1	UN	-1.807632D+02	1.344183D+02	-3.248046D+00	1.586022D-02	7.176775D-02	-4.272023D-03
3- 2	UN	1.393917D+02	1.959840D+02	-4.098630D+00	1.598712D-02	6.408558D-02	-3.787372D-03
3- 3	UN	9.425145D+01	9.459477D+01	-9.262824D+00	1.590184D-02	5.714585D-02	-4.175189D-03
3- 4	UN	3.337010D+01	2.590295D+01	1.501857D+01	1.593711D-02	5.007500D-02	-4.469530D-03
3- 5	UN	-6.047471D+01	-1.292435D+02	5.569786D+01	1.606320D-02	4.335403D-02	-4.969834D-03
3- 6	UN	-7.954779D+01	-1.628410D+02	3.083150D+01	1.588328D-02	3.617822D-02	-4.646881D-03
3- 7	UN	-9.699374D+01	5.964082D+01	2.840112D+01	1.613789D-02	2.771971D-02	-4.685691D-03
3- 8	UN	-1.187542D+02	1.024373D+02	3.033296D+01	1.614574D-02	2.014697D-02	-4.419113D-03
3- 9	UN	-1.036578D+02	9.817419D+01	2.188990D+01	1.590734D-02	1.285494D-02	-4.582622D-03

4.3.7   Contact Element - GRDS

4.3.7.1 Introduction

The contact element is a nonlinear spring element that is used for contact between any node and a prescribed fixed contact plane. It is similar to the "gap" capability available with the nonlinear spring element but has a number of additional features. These are;

o The initial gap distance between the contact node and contact plane is calculated automatically.

o The contact plane can be skewed with respect to the global system without the necessity to use multi-point constraint conditions.

o Sliding friction forces between the contact node and contact plane are included.

o User input contact and rebound tolerances can control the contact and rebound.

4.3.7.2 Contact Element Input Data

The data for the ground contact element are specified with the following cards:

Group	Keyword	Required	Comments
C	GRDS	Yes	Element identification and connectivity.
H	PGRD	Yes	Element properties including contact plane definition.

4.3.7.3. Contact Element Connectivity (GRDS, Group C)

A separate ground contact element is defined from each contact node to a fixed contact plane. The contact element connectivity is specified with a unique member identification (MEM) followed by the contact node number (NODE) and an identifier (NDIR) that defines the global axis which is closest to the normal to the contact plane. The NDIR can be 1, 2, or 3 denoting the global X, Y, or Z directions. The member and node identification can be any number of up to five digits.

The use of the direction identifier NDIR can be demonstrated for the case of contact with the ground which is parallel to the global X, Y plane and whose normal is parallel to the global Z axis. Then NDIR = 3, denoting that the contact plane's normal is closest to the global Z direction. For the case of contact with a global Y, Z plane, NDIR = 1. Note that the contact plane does not have to be exactly normal to the NDIR direction, that more than one contact plane is allowed, and that one node can have more than one GRDS element to contact more then one surface.

Example: Mass impacting two constraint planes
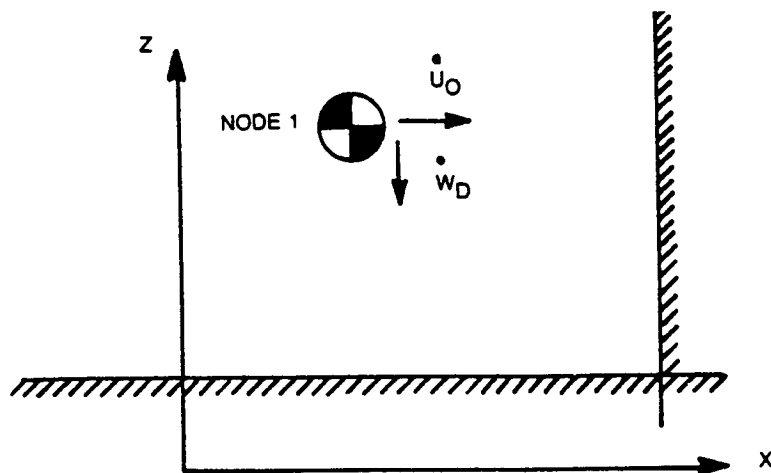

```
0.........1..... ...2..........3..........4..........5..........6..........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890

GRDS           1     1     3
GRDS           2     1     1
```

NOTE: Additional input (GRDS) is required to define the contact plane in Group H.

4.3.7.4 Contact Element Properties (PGRD, Group H)

The properties for the contact element include a multilinear force versus displacement table, three points defining the contact plane, and a number of parameters.

The input parameters on the PGRD cards are summarized below:

NSPRG   The number of force-displacement points

FRICT   Coefficient of sliding friction

EPSC   Contact tolerance

EPSR   Rebound tolerance

DMASS   Characteristic mass used to compute the damping coefficient

COORD   Coordinates of three points on the contact plane

FORCE   Contact force

ELONG   Contact displacement (penetration).

The contact plane is defined by three points denoted by i, j, k in fig. 4.3.11.
The order in which the coordinates (COORD) of these points are given define a right
hand local coordinate system, X', Y', Z' in the following manner: The direction i - j
defines the X' axis, Y' starts from point i and is parallel to the normal vector from
the X' axis to point k, and Z' is perpendicular to the X', Y' plane. At the initial
time t=0, the contact node must be either touching the plane or be above it, i.e. $Z_0 \geq$
0. Since the direction of the Z' axis is determined by the order of the nodes i, j,
k, the user must determine on which side of the contact plane is the structure, and
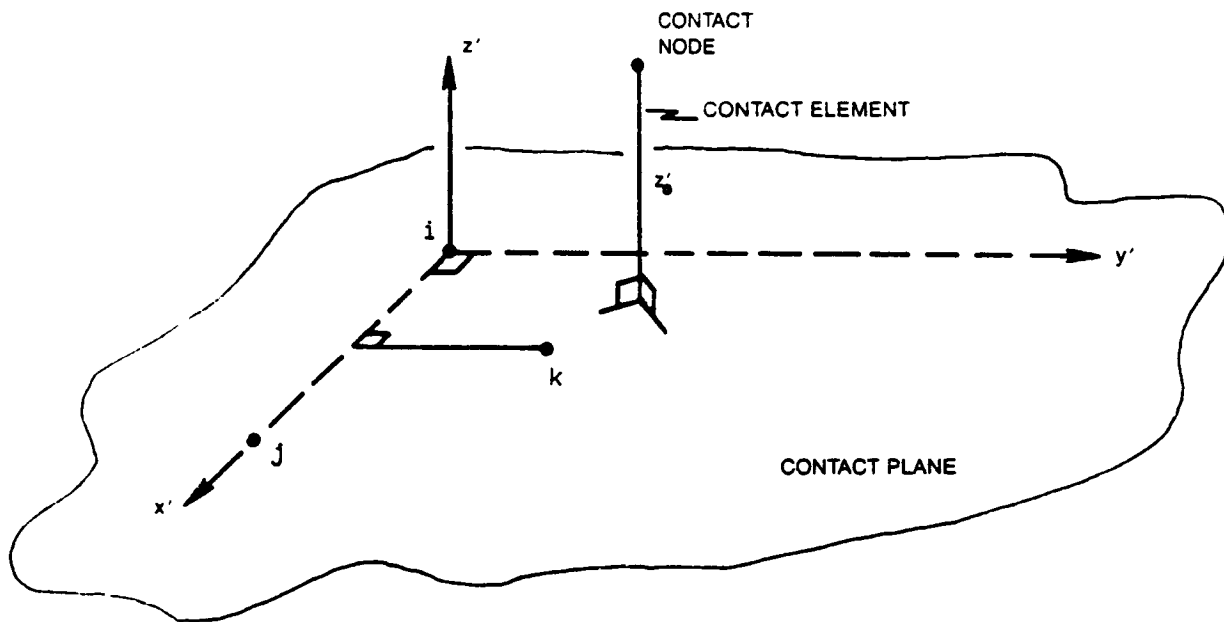specify nodes i, j, k in the proper sequence.

Fig. 4.3.11 Contact Plane Definition

The usual contact rebound sequence is as follows: The gap distance Z' will decrease to zero as contact occurs, then become negative as the contact node penetrates slightly into the surface while the contact force builds up, and eventually the contact node may rebound off the surface. The contact forces will vary as required by the action of the remaining structure on the contact node. The nodal motion normal to the surface is controlled by a user specified curve of contact force vs. penetration distance and by damping which is partly under user control. The nodal motion tangential to the surface produces a friction force with a user-specified friction coeffecient. The node can rebound off the surface, and there is no limit to the number of contact-rebounds.

The contact criterion is defined as:

$Z' > 0$, no contact
$Z' \leq 0$, contact.

Initially as long as $Z' > 0$ the contact spring has zero force and stiffness. When $Z' \leq 0$ a user-specified curve of compression force vs. penetration distance is used. The contact spring stiffness must be in the order of magnitude of 10^2 to 10^3 times greater than the effective structural stiffness at the contact node, to simulate a "rigid" barrier without numerical problems. However, if the large stiffness were imposed suddenly, and there were no special control on the time step, then a large displacement increment together with the large stiffness could place an unrealistically large force on the contact node. This large force could result in the rapid repulsion of the contact node back into the structure causing false failures. In some cases, the contact node can be returned rapidly into the contact surface again by the structure, resulting in a series of rapid repeated contacts and rebounds, which are not real and can cause numerical difficulties.

Several procedures are used to control the contact-rebound behavior. These include: incremental stiffening, damping, displacement increment control, and rebound capture. With these features, the contact force can increase to a realistic value without overshoot, without artificial contact-rebound oscillations, and then the contact node can be pulled away from the barrier surface by the steady rebound motion of the structure. Damping is not operational with the explicit time integrators ADAM and CDIF.

A typical contact spring curve with incremental stiffening is shown in Fig. 4.3.12. The stiffness is increased from zero to its "rigid" barrier value by increments over a small distance (a "boundary layer"). This boundary layer is made small relative to the gross structural deformation (4 mm is used in the example of Fig. 4.3.12).

As the contact node initally approaches the contact plane, the penetration d = -Z' is negative and the contact force is zero. After penetration d > 0, and the force rises smoothly and steadily over a small penetration distance, from points 1 to 2 to 3 etc., until the contact force is in dynamic equilibrium with the structural and inertial forces acting on the contact node. The contact force and penetration will vary as the impact event continues, depending on the behavior of the structure attached to the node, and additionally upon the damping characteristics of the contact spring.

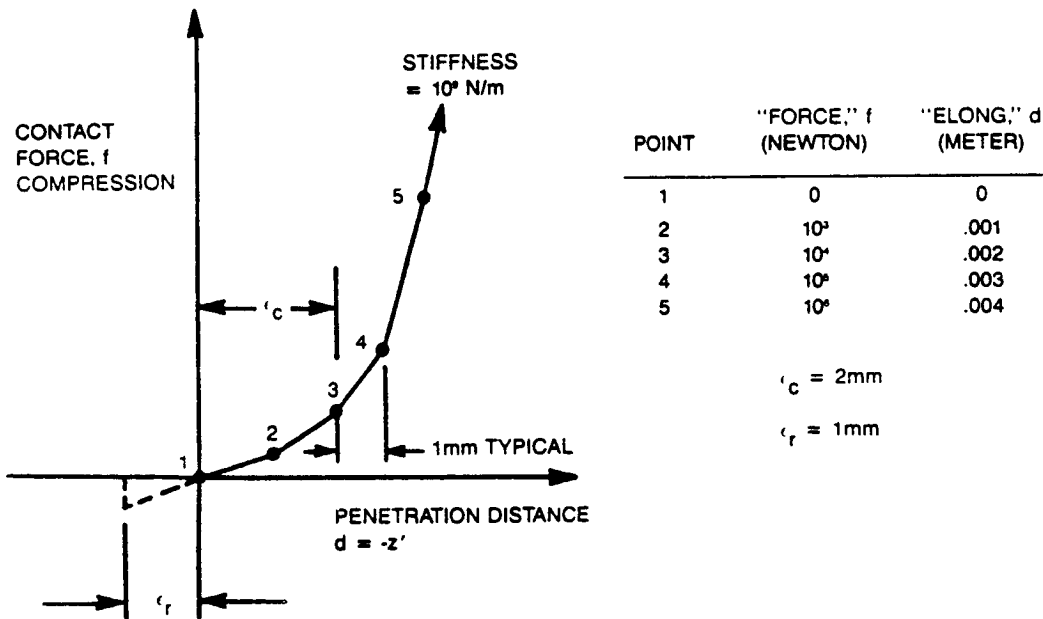| POINT	"FORCE," f (NEWTON)	"ELONG," d (METER)
1	0	0
2	10^3	.001
3	10^4	.002
4	10^5	.003
5	10^6	.004

ϵ_c = 2mm

ϵ_r = 1mm

Fig. 4.3.12 Contact Spring Boundary Layer Example

Additionally shown on fig. 4.3.12 is a user-specified contact tolerance ϵ_c (EPSC). If, during initial contact, the contact node passes through the ground plane such that $d > \epsilon_c$, the incremental solution will be discarded, the time (or load) increment will be reduced to make $d < \epsilon_c$, and the increment solution will be recomputed. This will prevent initial excess penetration with its corresponding excess force, but the computation time will be increased.

If the contact node is withdrawn from the contact plane so that $d < 0$, a special rebound procedure is used. The rebound criterion involves a user-specified rebound tolerance, ϵ_r (EPSR), to control spurious rebound and intermittant contact. As shown in fig. 4.3.12, a tension force is imposed between the contact plane and the node in the rebound tolerance region, $-\epsilon_r \leq d < 0$, after the initial contact has occurred. In this tension region, the stiffness is the same as that used for the first compression increment. If the contact node is pulled out to $d < -\epsilon_r$, then the contact force drops to zero. Subsequent contacts are treated the same as the first one; that is, no force until $d > 0$, then the contact/rebound process can repeat.

The damping force applied to the contact node is

$$f_d = 2\sqrt{km}\ \dot{Z}' \tag{4.3.17}$$

Where k is the absolute value of current stiffness of the contact spring, m is a user-specified characteristic "damping mass" (DMASS), and $\dot{Z}'$ is the normal velocity of the contact node relative to the contact plane. This nonlinear damping force is proportional to, and opposes, the normal velocity of the contact node. Equation 4.3.17 represents the critical damping for a one-degree-of-freedom system having stiffness k and mass m. The damping factor is partly under user control through the DMASS in Group H input, and partly by the program through the changing element stiffness. Note that the damping force is zero until the contact node touches the contact plane. At the precise point of contact $d = 0$, there will be no stiffness force but the damping force will be nonzero. This will usually only be seen when the contact node is touching the contact plane at $t = 0$. Damping is not operational with the explicit time integrators ADAM and CDIF.

The sliding friction force applied to the contact node is tangential to the contact surface in a direction opposite to the current tangential velocity of the node relative to the contact surface. The magnitude of the tangential friction force is

$$f_t = -\mu f_N \tag{4.3.18}$$

where μ is the user-specified coefficient of sliding friction (FRICT), and f_N is the stiffness force of the contact spring. The damping force is not used here because of its artificial purpose. Note that only sliding friction is calculated. If the contact node's tangential velocity is zero, the friction force is also zero, so that no static friction is imposed. In addition, no friction force will be applied in static problems (STAT time integrator) even though sliding can occur.

The recommended procedure for preparing the input for the contact element's properties on the PGRD cards of Group H is as follows:

1. Estimate the largest possible displacement increment imposed on the spring by the impact speed and the time step at the instant of gap closure. This can be easily done in most cases, by using the product of the initial impact speed and the maximum allowed time step (for example; 14 m/sec x 50 x 10^{-6} sec = 7 x 10^{-4}m = 0.7mm, therefore use 1 mm).

2. Define the value of the "rigid" barrier stiffness at 10^2 to 10^3 times the stiffness of the surrounding structure (10^6-10^7 lb/in. or 10^8-10^9 Newtons/meter have worked well for "rigid" stiffness in some autos and aircraft in past cases).

3. Increase the stiffness from zero to the "rigid" value in 3 or 4 jumps (for example; 0, 10^6, 10^7, 10^8, 10^9 N/m as in Fig. 4.3.12). Use the displacement increment found in step 1 above for each segment to insure that at least one point on each stiffness segment will be used. The FORCE-ELONG input pairs of Fig. 4.3.12 were chosen for convenience rather than exactness.

4. Choose a contact tolerance EPSC that is larger than the displacement increment calculated for step 1 above. This should allow the "boundary layer" stiffness forces to increase properly without reducing the time step. The contact tolerance then becomes an upper limit control, which will protect against unexpectedly large increments, but at the price of temporarily reducing the time step. (For the example, 2mm would be used).

5. The rebound tolerance EPSR might be in the range between zero and the contact tolerance value (EPSC). Try using a value equal to the displacement increment calculated in step 1, until greater experience is gained.

6. Specify large damping by using a damping mass (DMASS) that is very large compared to the effective mass at the contact nodes (from 10 to 100 times will usually work well).

With these values, the "rigid" barrier plane will usually be penetrated some acceptably small distance into its "boundary layer", and the contact force will respond by increasing and decreasing in a smooth, controlled manner between reasonable limits, until the structure rebounds. Note that the damping is not operational with the explicit time integrator ADAM and CDIF.

It will be noted that this PGRD input for the ground contact spring element GRDS is similar to that of the PSPR input for the general purpose nonlinear spring SPNG when used as a gap spring. The ground contact spring is a more recent development which provides an enhanced contact capability, yet reduces the amount of input data. The user does not specify the initial gap distance from the node to the contact surface because the program calculates it. Once a contact surface's properties have been defined in one PGRD card set, it can be assigned to any number of contact nodes having varying gap distances. Therefore, a large number of gap spring property card sets are not needed with the ground contact springs. Usually, only one such card set is required. In contrast, the general purpose gap spring requires a separate PSPR card set for each different gap distance.

The card input is in the following form; CARD 1 contains the key word PCON followed by the number of force-displacement points (NSPRG), the friction coefficient (FRICT), the contact tolerance (EPSC), rebound tolerance (EPSR), and the damping mass (DMASS). The next three cards each contain the global coordinates of point on the contact plane. The order of their input determines the direction of the outward pointing normal to the contact node. This normal should always be on the same side of the plane as the contact node. The succeeding cards are the force (FORCE) versus displacement (ELONG) pairs. These must start at 0,0 and must be always positive. The ELONG values must increase monotonically, but the FORCE values can vary arbitrarily. The PGRD card set ends with a list of the identification numbers for the applicable GRDS members.

Example:

Contact on a plane whose perpendicular is in the global Z direction.

The following parameters are specified: Coefficient of friction $\mu = .3$, Contact/rebound tolerances $\varepsilon_c = .10$, $\varepsilon_r = .05$, damping-mass $m = 10$.

The three points defining the contact plane are:

Point	X	Y	Z
i	0.0	0.0	1.0
j	0.0	10.0	1.0
k	-10.0	0.0	1.0
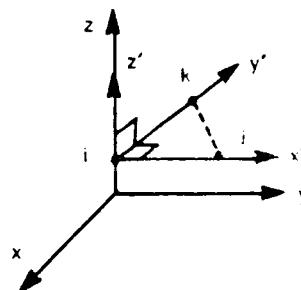
Five points are used to defined the contact force versus penetration displacement curve.

Force	Displacement
0.0	0.0
$.5 \times 10^2$	.05
$.5 \times 10^3$	.10
$.5 \times 10^4$	.15
$.5 \times 10^5$	.20

(These are the pound and inch approximations of the boundary layer example in Fig. 4.3.12, p. 4.111. It ends in a "rigid" stiffness of 10^6 lb/in.)

The data is applicable for GRDS members 101 and 102.

```
0....... 1..........2..........3..........4..........5..........6..........7..........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890

PGRD       5.0              .3          0.10              0.05          10.
           0.0              0.0          1.0
           0.0             10.0          1.0
         -10.0              0.0          1.0
           0.0              0.0
           0.5      E+02    0.05
           0.5      E+03    0.10
           0.5      E+04    0.15
           0.5      E+05    0.20
     101  102
```

4.3.7.5 Contact Element Loads (Group I)

CONC - Concentrated forces applied at specified contact nodes in the global directions in units of force.

In addition, a table of the variation of the load magnitude versus time must be specified in the input of Group I. CONC is discussed in Section 4.6.

4.3.7.6 Contact Element Output

This section shows examples of typical output generated when using the contact spring element.

Contact Element Connectivity

```
                 GROUND CONTACT ELEMENTS
        MEMBER     NODE    DIRECTION
         101       701        3
         102       702        3
```

Contact Element Stiffness Properties

```
            GROUND CONTACT ELEMENT PROPERTY TABLE  1
      FRICTION            DAMPING
      COEFFICIENT         MASS
      3.0000000E-01   1.0000000E+01
      COORDINATES OF GROUND CONTACT PLANE
          X              Y               Z
      0.0000000E+00   0.0000000E+00   1.0000000E+00
      0.0000000E+00   1.0000000E+01   1.0000000E+00
     -1.0000000E+01   0.0000000E+00   1.0000000E+00
          FORCE                    ELONGATION
      0.000000E+00             0.000000E+00
      5.000000E+01             5.000000E-02
      5.000000E+02             1.000000E-01
      5.000000E+03             1.500000E-01
      5.000000E+04             2.000000E-01
```

```
        APPLICABLE MEMBERS
            101   102
```

Contact Element Output

The output shown below are for two different cases, one before contact and one after contact. Contact is indicated in the appropriate column with a YES or NO. The "Closest global axis" indicates that the normal to the plane is specified to be closest to one of the global XYZ directions indicated by a 1, 2, or 3 respectively.

The friction force components are tangent to the contact plane and in the contact plane's local coordinate system, defined earlier in Section 4.3.7.4 and Fig. 4.3.11.

The "distance from plane" is the value of Z'. A positive value is the gap distance from the node to the plane before contact, and a negative value is the penetration after contact. Note that the input penetration distance d has the opposite sign.

The stiffness and damping forces are added to produce the total normal force. Normal forces are positive for tension (pulling the node into deeper penetration) and negative for compression (pushing the node out). The signs of the stiffness force and distance from plane should always agree.

The total friction force (vector sum) is equal to the friction coefficient times the normal stiffness force.

FORCES AND CONTACT CONDITION FOR GROUND CONTACT ELEMENTS AT TIME = 0.500000E-01

MEMBER	NODE	CLOSEST GLOBAL AXIS	CONTACT	FRICTION LOCAL X	FORCES LOCAL Y	NORMAL FORCES			DISTANCE FROM PLANE
						STIFFNESS	DAMPING	TOTAL	
101	701	3	NO	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	5.000E-02
102	702	3	NO	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	5.000E-02

FORCES AND CONTACT CONDITION FOR GROUND CONTACT ELEMENTS AT TIME = 0.150000E+00

MEMBER	NODE	CLOSEST GLOBAL AXIS	CONTACT	FRICTION LOCAL X	FORCES LOCAL Y	NORMAL FORCES			DISTANCE FROM PLANE
						STIFFNESS	DAMPING	TOTAL	
101	701	3	YES	-1.295E+00	1.295E+00	-6.106E+00	3.152E+00	-2.953E+00	-6.106E-03
102	702	3	YES	-1.304E+00	1.304E+00	-6.146E+00	1.941E+00	-4.204E+00	-6.146E-03

4.116

4.4   NODAL COORDINATES (GROUP D, GRDX, GRDY, GRDZ)

Nodal coordinates are specified in a global XYZ coordinate system. Coordinates are input using three key words, GRDX, GRDY, GRDZ for the coordinates in the X,Y,Z directions respectively. Each input card of this type contains one of the above key words to specify the component, the coordinate location, and the applicable nodes. All the short form notations can be used to specify applicable nodes, i.e., 1 through 100 and 2, 4, 6 ... 100 becomes 1-100 and 2-2-100, respectively.

Example: An X coordinate of 20 is specified for nodes 1 through 10, 14, 20, 23, 26, 49, 54, 57, 37, 39, 40 and 101, 103 ... 121.

```
0........1.........2.........3.........4.........5.........6.........7.........8
1234567890123456789012345678901234567890123456789012345678901234567890123456789 0
GRDX      20.0            1  -10   14   20   23   26   49   54   57   37   39   40
GRDX               101    -2 -121
```

The coordinate 20.0 need not be input on the continuation card since these fields are ignored. Any number of continuation cards can be used with one blank I5 field ending the input for the coordinate. If the applicable nodes fill up the card, i.e., end on the last I5 field, then an additional continuation card must be input with the first I5 field blank.

4.4.1 Vehicle and Global Coordinate Systems (XSHF, YSHF, ZSHF, YAW, PITC, ROLL

While it is usual to use a single fixed global coordinate system for specifying the initial nodal locations, it can sometimes be better to specify the initial coordinates in a more convenient second system and then specify the position of the second system relative to the global system. This second coordinate system will be called the vehicle coordinate system, and will be useful in cases where the initial coordinates of the vehicle or structure are already known in the vehicle system which is displaced and rotated with respect to the global system fixed to the impact surface. This is even more useful when the vehicle will be given several different initial positions relative to the impact surface. In each case, the same nodal coordinates can be used in the vehicle coordinate system, while different initial positions are specified for the vehicle system relative to the global system. DYCAST will internally transform the initial vehicle coordinates into the initial global coordinates, which DYCAST will use for all subsequent calculations.

Therefore, use of any of the parameters XSHF, YSHF, ZSHF, YAW, PITC, and ROLL in input group D, will cause the initial coordinate transformation to be implemented, and the nodal coordinates input using GRDX, GRDY, GRDZ, must then be in the vehicle coordinate system. Values of XSHF, YSHF, and ZSHF will shift the origin of the vehicle coordinate system by the specified distances from the global system origin in the global X, Y, and Z directions. The vehicle coordinate system will be rotated relative to the global coordinate system according to the angles given by the parameters YAW, PITC, and ROLL, corresponding to rotations about the vehicle's Z, Y, and X axis, in that order. The order is important to note, since each rotation is done about the new position of the vehicle's axes, and a different order produces a different final orientation. First, the rotation is done about the vehicle Z axis, then about the new position of the Y axis, and finally about the new position of the X axis. In conventional aircraft terminology these correspond to the yaw (or vertical) axis, the pitch (or lateral) axis, and the roll (or longitudinal) axis. If the vehicle is then re-positioned relative to the global system, only the six parameters XSHF, YSHF, ZSHF, YAW, PITC, and ROLL need to be changed.

If all of the parameters XSHF, YSHF, ZSHF, YAW, PITC, and ROLL are omitted (or zero), then the vehicle and global coordinate systems coincide, and the nodal coordinates are in the global system, as before.

Note that the boundary conditions (group E) and initial conditions (group F) are not transformed internally during this operation, so the user must specify these input parameters in the global coordinate system, Thus, the group E and F input data may have to be changed whenever the vehicle coordinate system is rotated relative to the global system using YAW, PITC, and ROLL inputs. On the other hand, groups E and F data will not have to be changed if the vehicle system is only translated (not rotated), since the vehicle system will remain parallel to the global system.

DYCAST can be made to perform the necessary rotational transformation, on the boundary conditions, using the Non-Global Constraints Feature described on p. 4.128. In this context, the "new coordinate system" mentioned in p. 4.128, is the vehicle coordinate system. However, this input requires a separate card for each DOF, and therefore has limited utility.

Example 1: Global coordinates are specified for nodes 101 through 106.

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
GRDX    56.855         101 -106
GRDY    12.328         102  104  106
GRDY   -12.328         101  103  105
GRDZ    0.0            103  104
GRDZ    10.101         101  102
GRDZ   -10.101         105  106
SEND
```

Example 2: Same as above, but the structural model is translated 20 length units along the global X axis and rotated 30 degrees about the vehicle Z axis (which is initially parallel to the global Z axis).

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
GRDX    56.855         101 -106
GRDY    12.328         102  104  106
GRDY   -12.328         101  103  105
GRDZ    0.0            103  104
GRDZ    10.101         101  102
GRDZ   -10.101         105  106
XSHF    20.
YAW     30.
SEND
```

4.5   SINGLE AND MULTIPOINT CONSTRAINTS - GROUP E

Single and multipoint constraints are specified with three key words:

SPC   - Single point constraints.

MPC   - Multipoint constraints.

APPL   - Applied generalized displacements.

ACEL   - Applied generalized accelerations.

Each node can have up to 6 degrees of freedom, specified in the order u, v, w, θ_x, θ_y, θ_z

where

DOF 1 = u, translation in global x-direction
DOF 2 = v, translation in global y-direction
DOF 3 = w, translation in global z-direction
DOF 4 = θ_x, rotation about global x-axis
DOF 5 = θ_y, rotation about global y-axis
DOF 6 = θ_z, rotation about global z-axis

4.5.1 Single Point Constraints (SPC)

Input for the single point constraints appear on cards containing the key word SPC, followed by the NBND parameter in a six-digit word describing the boundary condition for each of the six DOF for a node in the order of $u, v, w, \theta_x, \theta_y, \theta_z$. The NBND parameter uses the following system:

0 - Denotes a completely fixed degree of freedom

1 - Denotes a free degree of freedom

2 - Denotes that a prescribed value of displacement or acceleration is to be specified for the degree of freedom

The nodes that are applicable to the specified boundary condition follow on the input card, right justified in fields of five. All the shorthand notation described in Section 3.1 can be used. That is, nodes 1 through 100 are written 1-100 and nodes 2, 4, 6 ... 100 are written 2-2-100. Any number of continuation cards may be used for a given specification. However, the columns specifying the boundary conditions (i.e., columns 6 through 11) are ignored on the continuation card. A zero or blank field of five (i.e., I5 field) terminates the card scan for a given boundary condition specification. If the last field of a card (columns 76-80) is the last specification, an additional blank card (continuation card) must follow.

If a node's boundary conditions are not set on an SPC card, then all the degrees of freedom are assumed to be free. In an idealization that contains beam or plate elements this default is equivalent to 6 free degrees of freedom (NBND = 111111). It should be noted in that case, that rotational degrees of freedom should be fixed for all nodes that do not connect beam or plate elements; that is, nodes that have no rotational stiffness or inertia assigned. To avoid the singular coefficient matrix caused by the missing stiffness and inertia, DYCAST will detect a zero on a main diagonal of the coefficient matrix, replace it with the value 1.0, and print a warning message. If there are no externally applied moments for the rotational DOF, the solution for rotation will still be zero, since there will be no internal (stiffness) moment. Therefore, the rotational boundary conditions do not have to be fixed by the user, but this fixity is recommended to reduce the total number of DOF required for the solution.

If a problem has no beam or plate elements (no rotational degrees of freedom) the default condition is internally set to 111000. Any of these default conditions can be changed by setting the appropriate default condition on the first SPC card with all nodes used in the problem specified. The last specification for any node will be the one used. There is no limit to the number of nodes in the SPC list, but a maximum of 200 different combinations of 0, 1, and 2 can be used in the six digit NBND word.

4.5.2 Applied Displacements (APPL)

Applied displacements can be specified only for static solutions (STAT in Group A). If prescribed values are to be specified for any of the degrees of freedom, the value is input on additional cards, that appear after the SPC's and MPC's. Note that the complete applied displacement at any node is the product of the displacement factor APPL for the node and the time function PTME of Group I.

The applied displacement cards (APPL) and multipoint constraint cards (MPC) can be mixed in any order following the SPC cards, but they must be preceeded by a card containing either of the key words APPL or MPC alone on the card. The following cards contain the key word APPL, the node number, degree of freedom number, and the prescribed value. The degree of freedom number is input as a number from 1 to 6 according to the component whose displacement is being specified, as in p. 4.120.

Example 1: Beam clamped at both ends, uniform transverse load through shear center (no torsion) along an axis of symmetry of cross section (planar deformation)
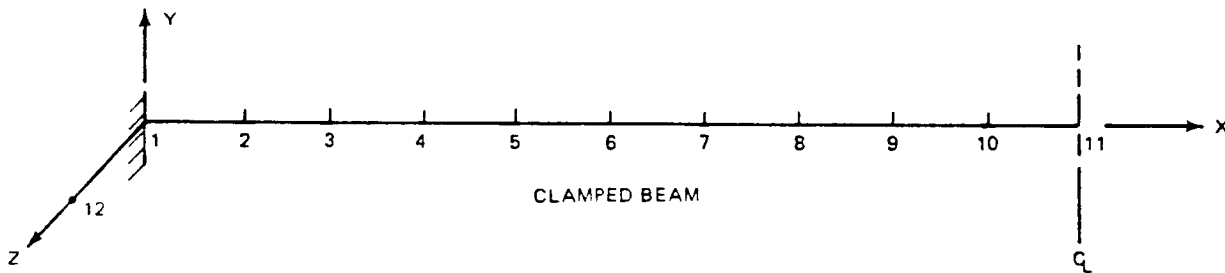
CLAMPED BEAM

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
SPC   000000        12    1
SPC   010000        11
SPC   110001         2   -10
```

NOTE: Node 12 defines the initial orientation of the beam cross sectional axes and since it is not part of the structural model it must be fixed. The SPC card for nodes 2-10 need not have been specified if three-dimensional beam elements are used but was included as shown to reduce the number of DOF to the minimum.

Example 2: Rectangular membrane panel under doubly symmetric in-plane loads with unrestrained edges.
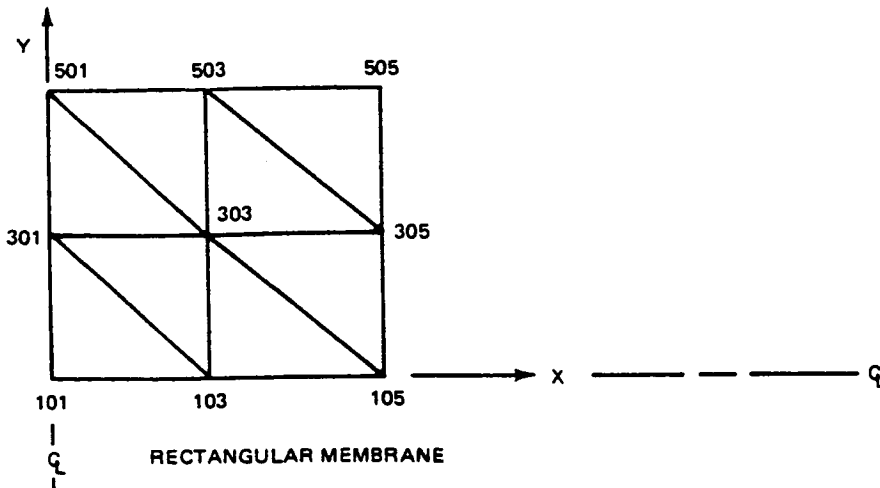
RECTANGULAR MEMBRANE

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
SPC  000              101
SPC  100              103  105
SPC  010              301  501
SPC  110              303  305  503  505
```

NOTE: The default conditions for membrane elements alone is 111000, and the rotational DOF need not be specified in this problem, since no beam or plate elements exist.

Example 3: Same as Example 2 with displacement factor of 0.04 applied in the y-direction at nodes 501, 503, 505.

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
SPC  000              101
SPC  100              103  105
SPC  010              301
SPC  020              501
SPC  110              303  305
SPC  120              503  505
APPL
APPL  501   2                     0.04
APPL  503   2                     0.04
APPL  505   2                     0.04
```

NOTE: The first APPL card must be without data, if it is the first card following the SPC set.

4.123

4.5.3 Multipoint Constraints (MPC)

The program implements a multipoint constraint capability of the form

$$\delta_i^d = \sum_{j=1}^n a_{ij} \delta_j \qquad (4.5.1)$$

where δ_i is the ith dependent degree of freedom, a_{ij} are prescribed coefficients and δ_j are a set of independent degrees of freedom.

These constraints are specified on input cards that appear after the SPC input. The applied displacement cards (APPL) and multipoint constraint cards (MPC) can be mixed in any order following the SPC cards, but they must be preceded by a delimiter card containing either of the key words APPL or MPC alone on the card.

This delimiter card is followed by cards containing the dependent node number and degree of freedom (left hand side of Eq. 4.5.1) along with the independent node number, degree of freedom, and a_{ij} (right hand side of Eq. 4.5.1). Each card can accommodate up to two independents of the equation. Any number of continuation cards can be used and are indicated by leaving the dependent node specification blank on succeeding cards.

When a degree of freedom (DOF) is dependent it must also be deleted (set to zero as if fixed) on an SPC input card and it may not appear as an independent in any constraint equations, because it is eliminated from the set of unknown DOF.

If a node has been assigned any added inertia in Group G, each of its DOF can have at most only one term in its dependency equations, i.e., can be dependent on only one other DOF.

A current restriction exists in the program that allows applied displacements and dependent degrees of freedom to be written at a maximum of 200 nodes and with a maximum of 500 coefficients a_{ij} in total.

4.124

Example 1:

The displacement u at node 100 is related to independent degrees of freedom according to the following equation:

$$u_{100} = 0.5\ u_{10} + 0.5\ u_{11} + 0.25\ u_{12}$$

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
MPC
MPC    100    1        10   1   0.5            11   1   0.5
MPC                    12   1   0.25
```

NOTE: The <u>first</u> MPC card, without data, is only necessary if this is the first card following the SPC set.

Example 2: Rectangular panel, fixed on one edge, uniform shear displacement on the opposite edge.
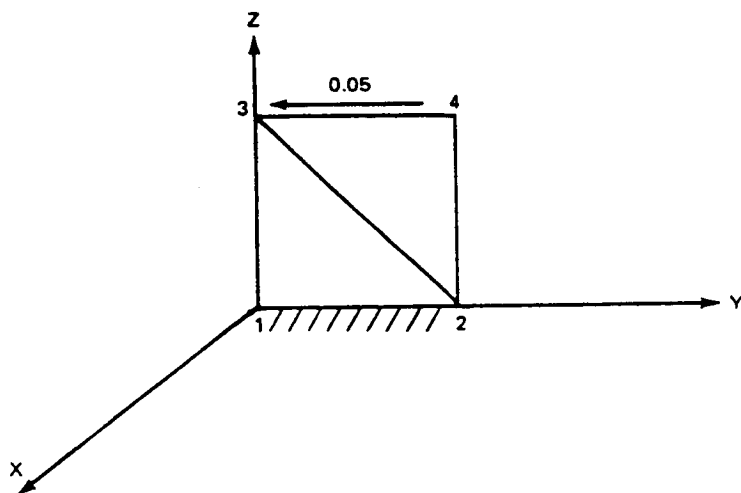
Nodes 1 and 2 are fixed. There is an applied displacement at node 3 in the negative y direction. The distance between nodes 3 and 4 remains constant. Line 3-4 remains parallel to line 1-2

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
SPC  000            1   2   4
SPC  021            3
APPL
APPL    3   2                  -0.050
MPC     4   2        3   2   1.0
MPC     4   3        3   3   1.0
```

NOTE: The SPC cards must specify zero for node 4 degrees of freedom 2 and 3 because they are effectively eliminated from the solution (they are dependent degrees of freedom). (An alternative procedure can be to specify the same SPC and APPL for node 4 as for node 3, and delete the dependencies. This is a slightly different condition, in that edge 3-4 is then allowed to rotate.)

4.5.4 Non-Global Constraints (MPC)

The multipoint constraints can be used to transform displacements at a node to another coordinate system. Thus, the MPC's can be written as

$$\delta_i = \alpha_1 \bar{\delta}_i + \alpha_2 \bar{\delta}_j + \alpha_3 \bar{\delta}_k$$
$$\delta_j = \beta_1 \bar{\delta}_i + \beta_2 \bar{\delta}_j + \beta_3 \bar{\delta}_k \qquad\qquad (4.5.2)$$
$$\delta_k = \gamma_1 \bar{\delta}_i + \gamma_2 \bar{\delta}_j + \gamma_3 \bar{\delta}_k$$

where the α, β, γ's are the direction cosines of rotation, δ_i, δ_j, δ_k are the displacements with respect to the original global directions, and $\bar{\delta}_i$, $\bar{\delta}_j$, $\bar{\delta}_k$, are the components of displacements at the node with respect to the new coordinate system.

The new coordinate DOF will replace the original global cartesian DOF, with the appropriate transformations of the local stiffness, mass, loads, etc. Thus, the SPC input must refer to the new coordinate DOF. This feature can be used if a node is constrained in a non-global direction. It will be particularly useful for structures that conform to cylindrical or spherical coordinate systems. The output displacements, velocities, and accelerations are transformed back to the global system.

The input for this feature is the same as for the standard MPC case with the exception that the SPC's are not fixed to account for this type of MPC. However, they may be fixed to account for constraints in the new coordinates.

Example:
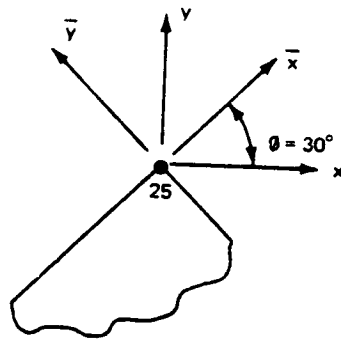
For this problem the displacement $\bar{u}$ is set equal to zero and $\bar{v}$ (along $\bar{y}$) is free. Assume there are no rotations or z displacements. The global coordinates of the problem are x, y. Thus

$$u_{25} = \bar{u}_{25} \cos \phi - \bar{v}_{25} \sin \phi = -\bar{v}_{25} \sin \phi$$
$$v^{25} = \bar{u}_{25} \sin \phi + \bar{v}_{25} \cos \phi = \bar{v}_{25} \cos \phi$$

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
SPC   010000        25
MPC
MPC      25    1       25    1   0.8660254        25    2    -0.5
MPC      25    2       25    1   0.5              25    2    0.8660254
```

NOTE: On the SPC card, node 25 degree of freedom 1 is specified to be 0 but node 25 degree of freedom 2 is specified 1 or free since it remains an independent degree of freedom (although rotated). If the normal displacement $\bar{u}$ were not fixed it would have a 1 boundary condition specified.

For this particular example, the terms involving $\bar{u}_{25}$ could have been omitted, since its value is set to zero by the SPC.

The applied load is separated into the product of a spatial distribution function and a time function

$$P(x, y, z, t) = P_1(x, y, z) F(t) \tag{4.6.1}$$

The spatial distribution can be specified as concentrated loads at nodes in the global cartesian coordinate system or as a distributed loading on an element, which loading is consistently assigned to the element node points.

The time function $F(t)$ can be specified in three different ways as explained later. A simple single-phase (one time function for all load components) or more complex multi-phase (each component having its own time function) loading can be used.

At each time step, the consistent nodal load components from the element-oriented loads (BMLO and SURF) are calculated, transformed into globally-oriented components, and added to the globally-oriented concentrated loads (CONC). Each global load component at each node is then multiplied by the current value of the time function assigned to it.

4.6.1 Concentrated Loads (CONC)

Concentrated loads are applied at nodes in the global coordinate system, and keep their orientation while following the node through its motions. These loads are input as Group I data in a sequence of two input cards. The key word specification is CONC. The first card in the sequence contains the key word, CONC, the node number and the forces in the global x, y, z directions. The second card contains the key word and the three moments with respect to the global x, y, z directions.

A CONC input card can be specified for any node point in the finite element model. Example input are shown below:

Example 1: Load factor 100 Newtons in the positive global x direction and moment factor 55 Newton meters in the global y direction at node 105.

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
CONC    105    100.0          0.0            0.0
CONC           0.0            55.0           0.0
```

Example 2: Applied load factors 25, 33, 15 lb at node 15 in positive x, y, z global directions, no moments

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
CONC    15     25.0           33.0           15.0
CONC           0.0            0.0            0.0
```

4.6.2 Line Load on a Beam Element (BMLO)

Line loads are specified in the element local coordinate system, and follow the element orientation. The consistent load vector for a distributed line load on a beam element is obtained from

$$\{f\} = \int_0^1 [N]^t [Q] dx \qquad \begin{matrix} q_i \\ \\ q_j \end{matrix} \qquad (4.6.2)$$

where $[N]$ is the shape function for the lateral displacement, $[Q]$ is the distribution of the load from node i to node j. In DYCAST the distributed line load can vary linearly between each node. The quantities q_i and q_j are the magnitude of the loading at each node in the local y and z directions. These quantities represent input and are in units of force per length or moment per length. The local y direction is towards the third node point that specifies the beam coordinate system on the connectivity input (Section 4.3.3) and local z is perpendicular to this direction. Thus the consistent load vector f is defined in the local x-y and x-z planes. This specification defines a following load which is always perpendicular to the current beam configuration.

Example 1: Uniform load factor of 100 lb/in in the local y direction for elements 1, 2 and 3

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
BMLO     100.0         100.0          0.0           0.0
    1     2     3
```

The applicable members are specified using usual notation as described in Section 3.

Example 2:

Linearly varying line load in the z-direction whose magnitude factor is equal to 150 Newtons/meter at node i and zero at node j for element 95.


```
0.........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
BMLO     0.0            0.0           150.0          0.0
  95
```

4.6.3 Distributed Surface Loading on Triangular Bending Elements (SURF)

Surface loads are specified in the element local coordinate system, and follow the element orientation. The three components of the surface loads are assumed to vary linearly from node to node in the plane of the element. They are applied in the element local coordinate system in the x, y, z directions. Local x is in the direction of the first two nodes specified in the connectivity input and local y is in the plane of the element perpendicular to the x direction. The three components of the surface load per unit area P_x, P_y, P_z are specified at the three nodes of the element. Applicable members with these values of loads at the respective nodes are then specified. Specific consistent load vectors are calculated in the element routines based on this input and the element shape functions. This load specification represents a following load and is always perpendicular to the current element configuration. Examples of the required input follow:

Example 1:

Uniform surface load factor 100 psi in the negative z direction for elements 5 through 10.

```
0........1.........2.........3.........4.........5.........6.........7........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
SURF          0.0            0.0            -100.0
SURF          0.0            0.0            -100.0
SURF          0.0            0.0            -100.0
     5   -10
```
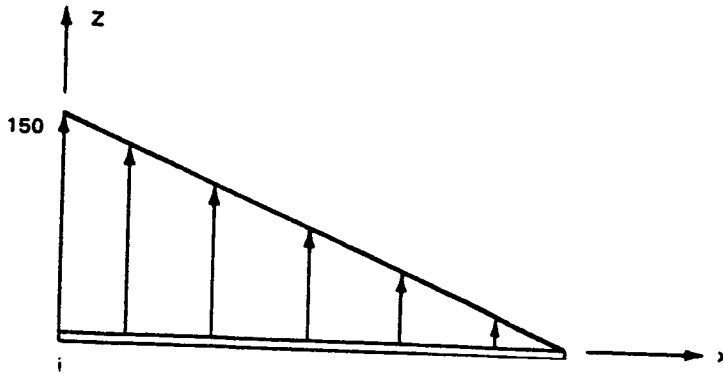
The applicable members are specified using the usual notation as described in Section 3.

Example 2:

Linearly varying surface load factors 150, 100, 50 Newtons/m^2 at nodes i, j, k in the local z direction for element 50.

```
0........1.........2.........3.........4.........5.........6.........7........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890

SURF          0.0            0.0            150.0
SURF          0.0            0.0            100.0
SURF          0.0            0.0            50.0
     50
```

4.6.4 Load Factor Time Functions (PTME, PTM2, PTM3)

The time variation functions for the applied loads are specified in tabular form of load factor versus time. Values of the load factor, P_1, and load factor increment, ΔP_1, are obtained by interpolation within these tables. If the maximum time in a table is exceeded, the load factor is held constant at the last value, with zero slope. Up to 50 points can be input per table. The tables must be in order of increasing time, starting from $t = 0$.

This input is required whenever a spatial distribution of applied load (CONC, BMLO, or SURF) or displacement APPL is specified. If it is not input an error message is printed and the job is aborted.

There are three types of time functions available. The first (PTME) involves only one common time function for all the applied loads (single-phase loads). The second (PTM2) allows different time functions to be specified at different nodes, but all the six components at any node will have one time function. The third (PTM3) allows a different time function for every load component. One of these three time functions must be used whenever applied loads are specified.

The applicability of the various time functions is given in the following table:

| Time Function | Applied loads | | | Applied Displacement | Applied Acceleration |
	CONC	BLMO	SURF	APPL	ACEL
PTME	Yes	Yes	Yes	Yes	No
PTM2	Yes	Yes	Yes	No	No
PTM3	Yes	Limited*	Limited*	No	No
ACELT	No	No	No	No	Yes

*Small rotations

These time functions are all applied to the globally-oriented nodal components. The BMLO and SURF are element-oriented loads that follow the elements as they move, and lead to globally-oriented nodal force components whose magnitudes will vary with the rotation of the elements. Thus, the time functions are applied to the resulting global load components, not to the element-oriented load components. There will be no difference in the results except for the cases where different PTM3 time functions are assigned to more than one force component or more than one moment component at the same node for the element-oriented BMLO and SURF load inputs. In these cases, the loads applied to the nodes can not generally be predicted in advance, and will not be those desired, except when the element rotations are small. Therefore, the use of PTM3 with BMLO or SURF is limited to small rotations in these cases. As a result, the PTM3 is not generally recommended for use with the BMLO or SURF loads.

In cases where most of the load components share the same time function, and a small number of components have different time functions, it will be useful to use the PTME input for the most common time function and then use either the PTM2 or PTM3 time functions for the exceptions.

4.133

If the time step used in the solution of a problem is comparable to or large than the period of an oscillating load function, or comparable to or larger then the time interval between abrupt changes in a nonlinear load function, then some portion of the input load function will be ignored. This is similar to the errors incurred in using too large a digital sampling rate for rapidly varying data. Therefore, if all the details of the input load function are important, than the user should insure that the time steps are small enough to follow the input function with sufficient accuracy. This can be done by using a maximum time step (DTMAX) on the NEWM, STAT, or WLSN cards. However, small time steps can be costly, so the input load function should use as large a time interval or be as smooth as possible, to avoid unnecessary reductions in the solution time step.

The load factor versus time tables must also be input for a static analysis in order to define the load factors and load increments Here the time is used as a parameter defining these quantities. If a table contains one straight line segment (constant rate) then the load factors are proportional to time, and the load rate is a constant. A variable load step procedure in a static analysis can be defined by inputting a number of linear segments. This will be demonstrated in the following example.

Example:

Variable load step static analysis (i.e., decreasing slope)

```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
PTME        0.0            0.0
PTME        1.0            1.0
PTME        1.5            2.0
PTME        1.75           3.0
```

Note that the time step is used to obtain the load and the load step from the table. Assuming that the time step will be constant, the load steps may not be constant. If sufficient nonlinearities are involved, then the convergence criteria may force the time step and therefore the load step to vary further throughout the problem.

For the special case of applied displacements, the global spatial displacement distribution (APPL) is multiplied only by the common time function (PTME), and is thus restricted to single-phase behavior. The APPL input is described earlier in Group E, p. 3.47.

The applied acceleration time function ACELT is used only for the applied accelerations ACEL, but is included in the above table of time functions for completeness. It is discussed in the next section.

4.6.5 Gravity Load or Weight (GRAV)

The main purpose of imposing gravity has been to compute the gross (rigid body) motions of a vehicle as it interacts with and rebounds from an impact surface. It is not intended here to compute accurately the stresses and deformations due to gravity, although that may result in some cases.

Distributed gravity forces (weights) can be imposed at the nodes parallel to any one of the global axes, using the GRAV input in Group I. These nodal weight vectors are added to the set of external load vectors (P in Eq 1.3.9 on p. 1.13), with the following conditions:
- o Each element's weight is distributed among its nodes in such a way that the resultant gravity force acts at its center of mass. The offset beam element is an exception as discussed below.
- o The weight of any user-specified lumped mass is also added to each node.
- o The resulting total weight at each node is equivalent to the total mass at the node multiplied by g, the acceleration due to gravity.
- o The time variation for dynamic problems is a step function starting at t=0 and constant for all subsequent time.

In lumping the weight vectors at the nodes, two simplifications have been made. First, the bending moments at the nodes of BEAM and TRP2 elements, caused by gravity acting on the distributed mass of an element, have been neglected. This means that the bending within an element induced directly by the weight is ignored, but the bending developed in the structure due to the weights placed at the nodes is included. Thus a model composed of only one beam or plate element will not deform correctly under gravity, because all the weights are placed at the nodes which are on the boundary. The finer the model the more accurate the gravity solutions become. This limitation should lead to negligible errors in typical vehicle models where the size of individual elements are small compared to the total size of the model.

Second, the beam elements have their weight placed at their end nodes, and not on their centroidal axes. In actuality, a beam that has its end nodes offset from the longitudinal centroidal axis will develop a torque about the nodal axis due to the gravity forces. This gravity torque is neglected in the beam element, but might be significant when the gravity forces and accelerations (1g) are dominant or are a significant part of the total forces and accelerations in the problem, and the beam elements have large offsets between their nodal and centroidal axes. Therefore, the problem solution could be significantly in error when gravity is the only load and large beam offsets are present. However, in the majority of vehicle crash simulations, the gravity loads and 1g acceleration are small compared to those imposed by the impact, and this error will have a negligible effect on the structural response. Note that this error does not diminish as the model is refined by merely increasing the number of beam elements. However, an accurate portrayal of this gravity torque can be made by linking the offset nodes to a set of nodes on the beam's centroidal axis, using short "rigid" beams.

It should be noted that, in dynamic problems, the step function time variation of the gravity loads is well-suited for initial conditions in which a vehicle is in a free-fall condition prior to impact. That is; there are no other external loads besides the weight, and the vehicle is then accelerating at 1g. In this case, there will be no initial structural deformations due to gravity, since each node is accelerated at exactly 1g. However, when any part of the weight is reacted by an external force, such as a ground reaction or aerodynamic lift, the program will solve the dynamic response of the structure to the distributed gravity forces applied as step functions in time. To-avoid this transient response, the nodes must be given an initial displacement which places the entire structure in static equilibrium with the applied loads. These static displacements are obtainable a from one-step static solution of the gravity load problem, and can then be introduced as inital displacements (IDSP) input in the dynamic problem.

4.6.6 Applied Acceleration (ACEL, ACELT)

Acceleration versus time functions can be specified for any motion component at any node, using the ACEL and ACELT inputs. The resulting velocities and displacements are calculated using the acceleration versus time function and the initial values of velocity and displacement specifed in the ACELT input card set. Since the applied acceleration leads to an applied displacement, which is a type of constraint, the ACEL card is used in Group E input - Constraints. Also in Group E, the constraint code NBND = 2 must be specified for the appropriate node and DOF on an SPC card, as in the case of applied displacement.

The variation of the acceleration function within the time step is ignored when calculating the accelerations; only the input acceleration value at the end of a time step is used directly. However, the numerical integrations to calculate the resulting velocity and displacement use the entire input acceleration function within a time step. Therefore, in a linear problem (small deformation elastic) there will be no significant loss of accuracy in using the input acceleration versus time function due to large time steps, since the applied accelerations and the resulting velocities and displacements will be correct at the end of each step. A nonlinear problem, on the other hand, is path dependent because conditions of plasticity, failure, and large deformations may occur within a large time step interval that will change the subsequent behavior and therefore these intermediate states cannot always be ignored. If the input acceleration function is changing rapidly, the time step used in the solution must be small enough to sample accurately this varying function. This is particularly important if there are peak acceleration amplitudes within a time step. The time step can be limited by use of the maximum time step input (DTMAX) on the NEWM, STAT, and WLSN input cards. However, small time steps can be costly, so the time intervals used in the input acceleration functions should be as large as possible, and the acceleration functions as smooth as possible, to avoid unnecessary reductions in the solution time step.

This page intentionally blank

5.0   DATA PREPARATION FOR PREPROCESSING AND POSTPROCESSING

This section describes the input for the SATELLITE and GRAFIX programs used for pre- and postprocessing of input and output.

SATELLITE has the capability of reading and checking input data and plotting the undeformed and deformed structural idealization. Additionally, an option exists to calculate the ordering of the external node numbers that lead to the minimum semibandwidth.

GRAFIX was written to postprocess data from a DYCAST restart file. The program currently reads data from a restart file and produces plots of the histories of displacement, velocity, and acceleration.

5.1 INPUT PREPARATION FOR SATELLITE

The input data cards must be in the sequence given here.

/ Card : Key Word DYCA Columns 1-4/

This required card uses four-character words as follows:

DYCA   The SATELLITE program can be used with a number of nonlinear programs. The key word for DYCAST is DYCA in the first four columns.

A number of options are available. These are specified by additional words on Card 1 in fields of four. These can be consecutive on the card or any multiple of four spaces can be skipped on the input card, between these key words. The available options and key words are:

BAND   Specifies that the optimum nodal sequence that minimizes the semibandwidth is to be determined. In this case the optimum order of the external node numbers for DYCAST Group B input data will be printed and output to a card image file. The original Group B nodal list should be replaced by this optimized Group B list. If this key word is not specified the bandwidth will not be optimized.

NOTE: Connectivities provided by multipoint constraints (MPC) will be ignored here! Thus two substructures connected only by MPC will be seen as two separate bodies, and only one will be optimized. This can be corrected by temporarily connecting the MPC'd nodes by dummy stringer elements for bandwidth optimization only.

SCAN   Process and check input data but do not produce any plots.

PRIN   Print transformed coordinates to be plotted.

REST   Read an existing DYCAST restart file and produce deformed plots.

SATELLITE can be used as a preprocessor to check input by plotting the undeformed finite element model or it can be used as a postprocessor to plot the deformed finite element model.

Format	Columns	Symbol	Comments
A4	1-4	DYCA	key word for DYCAST processing.
19A4	5-80		Any of the optional words listed above. Multiples of four blank columns can be inserted between key words for spacing appearance.

/ Model Data/

Preprocessing - Plot Undeformed Finite Element Model

The DYCAST input data deck (defining the finite element model) starting with the title card, follows the DYCA Card when producing undeformed plots. The minimum required input is from the title card through the nodal coordinates Group D, followed by a STOP or END card. Additional parts of the input deck beyond Group D will be ignored until a STOP or END card is reached. Alternatively, the entire deck may be input, starting from the title card. The title in the DYCAST deck will be used on the plots.

Postprocessing - Plot Deformed Finite Element Model

When deformed views are desired, specify the REST word on DYCA card above. This indicates that input will be from a restart file previously generated by DYCAST. Consequently, the input data deck should not be included. All that is necessary here is a title card that is used to label the plots produced by SATELLITE. The file identification and loading instructions are part of the system JCL or procedural commands that preceed the DYCAST input.

This required card determines the type of plots desired. Five different types of labeled pictures can be obtained. These are specified with the fixed point numbers right justified in any of the fields of five as follows:

1 = Numbered unconnected nodes

2 = Numbered nodes, unnumbered members

3 = Numbered nodes and members

4 = Unnumbered nodes, numbered members

5 = Unnumbered nodes and members

Format	Columns	Symbol	Comments
I5	1-25		As many of the number codes above as desired, in every fifth column

NOTES:

1. A complete set of plots is repeated for each of the types requested here. Thus, if 10 plots are requested by the following set of cards, and codes 3 and 5 are specified here, then 20 plots will be made, 10 for code 3 (numbered plots) and 10 for code 5 (unnumbered plots).

2. Plotting parameters can appear in groups, starting with a "plot type" card and ending with a REPT card. Several such groups can be used, one after the other, in the same input.

/ Initial Plane/

This required card defines the initial viewing plane as one of the model's three global coordinate planes, and orients it either horizontally or vertically on the image plane. Three characters are specified, corresponding to any sequence of the model global axes, x, y, z. The first designates the model global axis to be in the positive longitudinal (usually rightward horizontal) direction in the image surface; the second in the positive transverse direction (usually upward vertical); and third in the positive normal direction (toward the viewer). For example, yzx defines a view of the global yz plane with y to the right, z upward, and x toward the viewer.

The angles ALPHA, BETA, GAMMA on the next card set called "Viewing Angle and Time Increments" are applied to this initial coordinate view to produce the first view to be plotted.

Format	Columns	Symbol	Comments
3A1	1-3		Any permutation of xyz
3A1	4-6		Input the three characters +++ here (required)

NOTES:

1. The characters +++ in columns 4-6 are required, but are currently inactive.

/ Viewing Angle and Time Increments/

This required set of one or two cards alters the initial viewing angles and increments the angles for a sequence of rotated plots. These cards also contain controls on the time steps to be used in plotting a deformed structure from a DYCAST restart tape.

ALPHA, BETA, and GAMMA are initial angles added to the coordinate view specified in the "Initial Plane" card that preceeds this set. ALPHA is the rotation applied about the positive longitudinal axis (usually rightward horizontal) in the image surface, BETA about the positive transverse (usually upward vertical) axis, and GAMMA about the outward axis toward the viewer. They are applied in the sequence given, and follow the right- hand rule for positive rotations. Thus a positive ALPHA rotates the top of the image towards the viewer. Zero values for all three angles will leave the orientation as specified on the "Initial Plane" card.

The first plot will have the orientation determined by the "Initial Plane" card plus the initial angles ALPHA, BETA, GAMMA. The image will then be rotated by adding one increment each of INCA, INCB, INCG in sequence, and then the next plot will be made. Another set of angle increments will be added, and the next plot made, and so on.

Card 1:

Format	Columns	Symbol	Comments
I5	1-5	INCA	Increment of ALPHA (degrees).
E10.0	6-15	ALPHA	First value of rotation angle about the longitudinal axis in the image surface (degrees).
I5	16-20	INCB	Increment of BETA (degrees).
E10.0	21-30	BETA	First value of rotation angle about the transverse axis in the image surface (degrees).
I5	31-35	INCG	Increment of GAMMA (degrees).
E10.0	36-45	GAMMA	First value of rotation angle about the normal axis, out of the image surface (degrees).
I5	46-50	MAXROT	Total number of rotational increments to be plotted in a series, including the first (Default = 1 for only one view).
E10.0	51-60	X	Max. dimension of plot in the transverse or vertical direction in the image surface. Depends on the plotting device used.

(the following fields are only for deformed plots from a restart tape)

Format	Columns	Symbol	Comments
I5	61-65	IFACT	Scale factor to magnify deformations (default = 1).
I5	66-70	LST	Time step number for first plot, from DYCAST run that generated restart file. Use only for automatic time step incrementing.
I5	71-75	INC	Increment of time step number. Use only for automatic incrementing.
I5	76-80	NTLPM	Final time step number from DYCAST run. Use only for automatic time step incrementing.

Deformed plots can be requested at individual time steps. This is done with the following optional card input. This input is initiated by setting LST, INC, and NTLPM equal to zero or blank (blank field columns 66-80) on Card 1 above.

Card(s) 2: optional

Format	Columns	Symbol	Comments
16I5	1-80		Specified individual time steps for which the model is to be plotted from generating run of DYCAST (see Sect. 4.1.4), in increasing numerical order. Up to 50 can be specified. As many cards as needed can be used. A blank I5 field ends the data list. If list ends in last I5 field on a card, a blank card <u>must</u> follow.

NOTES:

1. The DYCAST output file will not contain data for every time step if KPRINT>1 was used on the REST card (p. 3.17) in the DYCAST analysis. In that case, the deformed model cannot be plotted at the "lost" time steps.

/ Deletions/

This required card allows the user to delete entire element types from the plot.

BEAM   —   will omit all beam elements

TRIM   —   will omit all triangle membrane elements

STRG   —   will omit all stringer elements

SPNG   —   will omit all spring elements

TRP2   —   will omit all triangular plate elements

GRDS   —   will omit all contact elements

If there are no element types to be omitted, a blank card must be specified. For example, to plot only BEAM elements, specify TRIM, STRG, SPNG, and TRP2 on this card.

Format	Columns	Symbol	Comments
4(A4,1X)	1-20		Any of the four character words listed above, with one blank space between them
	21-80		Blank

NOTES:

1. A blank card must be used if all types of elements are to be plotted.

2. To delete all elements, plotting only nodes, input the number 1 on the "Plot Type" card discussed above.

/ Nodes/

This required set of at least two cards specifies nodes to be included in scaling the plot. The plot will be scaled to get all nodes specified here onto the image surface. Therefore, the nodes specified here should at least include all the nodes attached to elements to be plotted. However, the size of the plot can be reduced by including additional nodes that are not attached to elements to be plotted. This is a useful method for controlling the size of partial views of the model. For example, if the part is be to drawn to the same scale as the total structure all the nodes in the structure should be input. If a part of the structure is to be plotted to the maximum size, these cards should include only the nodes that are in that part.

A blank card or card with only zero entries ends this card set.

Card(s) 1:

Format	Columns	Symbol	Comments
16I5	1-80	NODE	Node list as described above. All the shorthand notations are allowed. Any number of continuation cards are allowed.

Card 2:

A blank card is <u>required</u> here.

NOTES:

1.   If <u>all</u> nodes are to be included, it is sufficient to specify only one negative number whose absolute magnitude is greater than the largest node number in the model.

2.   An inclusive range of numbers can also be specified even though not all the numbers within the range are used for nodes. In this input, node numbers specified but not used in the model are ignored, as a convenience. For example, this input can specify nodes 1000-1999, although 1200-1299 do not exist in the model.

/ Elements/

This required set of at least two cards specifies elements to be included in the plot. Input format is the same as for "Nodes" above. All nodes associated with the element to be plotted must be specified on "Nodes" card set. A blank card or card with only zero entries ends this card set.

Card(s) 1:

Format	Columns	Symbol	Comments
16I5	1-80		Member numbers as described above. All shorthand notations and any number of continuation cards are allowed.

Card 2:

A blank card is _required_ here.

NOTES:

1. If all the elements are to be plotted it is sufficient to specify one negative number in the first I5 field whose magnitude is greater than the largest element number in the finite element model.

2. An inclusive range of numbers can also be specified even though not all numbers within the range are used. For example, specification can be made for elements 1000-1999, although 1200-1299 do not exist for that job.

/ End of Data/

This required card contains one of the control words STOP or REPT in columns 1-4 to end the job or to redefine a different set of plotting parameters, as follows:

Undeformed Plots - preprocessing from a DYCAST input deck.
If the control word is:

o   STOP, and a STOP card ended the DYCAST input deck, this is the last card of the SATELLITE input.

o   STOP, and the DYCAST input deck ended with an END card, to indicate that another DYCAST input deck follows, then the next input deck will be processed, subject to the same SATELLITE plotting commands.

o   REPT, then another set of SATELLITE input cards will follow, starting with a "plot type" card (p.5.4), to define new plotting parameters for the same model.

Deformed Plots - post processing from an existing DYCAST restart file. If the control word is:

o   STOP, this is the last SATELLITE input card.

o   REPT, then another set of SATELLITE input cards will follow, starting with a "plot type" card (p. 5.4), to define new plotting parameters for the same analysis results.

Format	Columns	Symbol	Comments
A4	1-4		Either STOP, or REPT, as discussed above.

NOTES:

1.   A special default case is available, for which the input consists only of the initial DYCA card, the model data file, and a STOP card. In that case, only one plot will be produced, of the undeformed model, viewing the XY plane along the Z axis, with all nodes and elements included and unnumbered. Either the DYCAST input deck or the DYCAST restart file can be used for the data file.

Example of SATELLITE Inputs

Example 1: Pre-processing

An undeformed plot of a structure is desired with 86 nodes, 50 members.
Triangular membrane members are omitted. One plot is desired with labeled nodes and
members in the yz plane with z in the paper transverse (vertical) direction. The
maximum plot size is to be 25 inches out of the 30 inch paper height. Optimization of
the nodal bandwidth is desired.

```
0........1.........2.........3.........4.........5.........6.........7.........8
123456789012345678901234567890123456789012345678901234567890123456789012345678901234567890
DYCA    BAND
(DYCAST input data deck from title to STOP)
    3
YZX+++
    0    0.       0    0.      0    0.        1     25.0
TRIM
  -86
(blank card)
  -50
(blank card)
STOP
```

Example 2: Post-processing

A deformed plot of a rectangular frame having 86 nodes and 50 elements is desired
at time 0.776 msec (corresponding to time step 100), with deformations magnified by
factor of nine, unlabeled members and nodes, omit triangular membrane elements.
Viewing angle is from the front left upper quadrant, where model global axes x,y,z are
forward, left, and upward in the structure. That is, starting from a head-on view of
front end (plane yz), rotate the model nose down 45 degrees and right 23 degrees as
seen by viewer. Plot size is to be 25 inches.

```
0........1.........2.........3.........4.........5.........6.........7.........8
123456789012345678901234567890123456789012345678901234567890123456789012345678901234567890
DYCA    REST
    RECT FRAME -  RESTART - DEFORMED PLOTS
    5
YZX+++
        45.0          -23.0           0.0           25.0          9
    100
TRIM
    1  -86
(blank card)
    1  -50
(blank card)
STOP
```

The plot generated is shown on the next page.
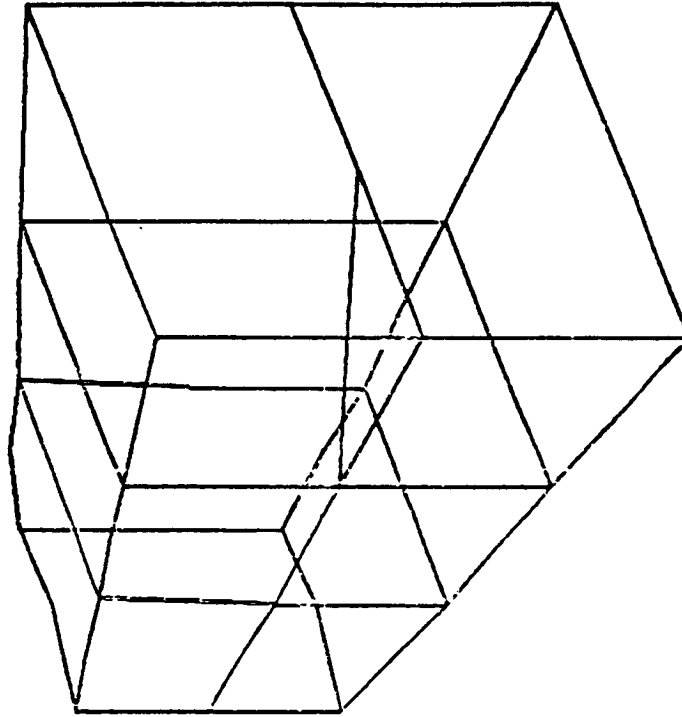
ALPHA = 45. BETA = −23. GAMMA = 0.

Y Z X + + +

RECT FRAME − RESTART − DEFORMED PLOTS

SCALE FACTOR 9.0   INCREMENT NO. 100   TIME 0.000775987

0199-012(T)

5.2   INPUT PREPARATION FOR GRAFIX

The GRAFIX post-processing program can read a DYCAST restart file and print and plot displacement, velocity, and acceleration versus time for any degree-of-freedom. The input is as follows:

<u>/ Any 80 Character Title/</u>

This required title card serves as a heading for all pages of output and for all plots.

This optional set of cards is used when a printout is desired of displacement, velocity, and acceleration for each node at specified time increments from a DYCAST restart file.

Format	Columns	Symbol	Comments
A4	1-4	PRIN	Print data.
14I5	11-80	INC	Time step numbers from DYCAST run, corresponding to the times at which data are to be printed. Each time has an associated unique increment number. If data are not printed in the original generating run, it can be retrieved and printed with this option. The short form input described in Section 3.1 may be used. That is, 1 through 100 is written as 1 - 100 and 2, 4, 6 ... 100 is written as 2-2-100. Any number of continuation cards can be used of the same format as the first card. If the short form notation is used, its specification must be complete on one card.

NOTES:

1. Either the PRIN or the PLOT set or both may be used.

2. The DYCAST output file will not contain data for every time step if KPRINT>1 was used on the REST card (p. 3.17) in the DYCAST analysis. In that case, the data cannot be printed for the "lost" time steps.

This card begins the parameters for a group of plots of data versus time.

Format	Columns	Symbol	Comments
A4	1-4	PLOT	Plot specified data
	5-10		Blank
I5	11-15	INC1	Initial increment number to be plotted
I5	16-20	INC2	Final increment number to be plotted
I5	21-25	GSCL	Acceleration is always plotted in g's. (1g = standard gravitational acceleration at earth surface).

If GSCL = 1 or blank; units of the DYCAST run to be plotted are in meters per sec^2, $g = 9.800m/sec^2$

= 2; units are in inches per sec^2; $g = 386.088 in/sec^2$

= 3; units are in feet per sec^2; $g = 32.1740 ft/sec^2$

NOTES:

1. If this PLOT option is specified the following cards determine the variables to be plotted and necessary scale factors. Many of these cards have default values that allow the card to be omitted.

2. Plotting parameters appear in groups starting with a PLOT card and ending with a SEND card. Within a group, the XLEN, XSCALE, YSCALE, and Plot Type card sets define the parameters for a group of plots. These card sets can appear in any order within the group. As many groups as desired may be used, one after the other, in an input deck.

3. The DYCAST output file will not contain data for every time step if KPRINT>1 was used on the REST card (p. 3.17) in the DYCAST analysis. In that case, the data cannot be plotted for the "lost" time step.

This optional card specifies the physical dimension of the time scale for all the plots in a group (between PLOT and SEND cards).

Format	Columns	Symbol	Comments
A4	1-4	XLEN	
	5-10		Blank
I5	11-15	LENGTH	Length of abscissa in inches, with scale markings every inch
	16-80		Blank

NOTES:

1. <u>Default</u>: If this card is not specified the length of the time scale is taken as 6 inches with scale markings every inch.

2. A combination of XSCALE and XLEN values should be chosen to produce a useful set of time scale markings.

/ Key Word XSCALE/

This optional set of two cards specifies the maximum time scale for the plots in a group (between PLOT and SEND cards).

Card 1:

Format	Columns	Symbol	Comments
A6	1-6	XSCALE	Indicates that the scale of the (horizontal) time axis is specified on the following card

Card 2:

Format	Columns	Symbol	Comments
E15.0	1-15		Maximum time for the plot

NOTES:

1.   Default: If these two cards are omitted, time will be automatically scaled to the maximum time on the restart tape.

2.   A combination of XLEN and XSCALE values should be chosen to produce a useful set of time scale markings. For example, if XSCALE = .050 secs and XLEN = 5, then the time scale will be marked every .010 secs.

This optional set of two cards specifies the maximum displacement, velocity, and acceleration scales for the plots in a group (between PLOT and SEND cards).

Card 1:

Format	Columns	Symbol	Comments
A6	1-6	YSCALE	Indicates that the (vertical) scales of displacement, velocity, and acceleration are specified on the following card

Card 2:

Format	Columns	Symbol	Comments
E15.0	1-15	MAXDISP	Vertical axis for displacements will be centered about zero, with limits of plus and minus MAXDISP.
E15.0	16-30	MAXVEL	Same as above for velocity
E15.0	31-45	MAXACEL	Same as above for acceleration
	46-80		Blank ·

NOTES:

1. Default: If these two cards are omitted, the vertical axes will be automatically scaled to accommodate the maximum positive and negative values in the group (between PLOT and SEND cards).

2. The vertical scale is 2 inches high and symmetric about zero.

<u>/ Plot Type/</u>

   This required card set specifies which types of plots are desired and the degrees of freedom whose data are to be plotted.

   Each card has a key word followed by pairs of node number and degree of freedom. The key word options are:

   DISP   -   Plot displacements only for each specified node and degree of freedom

   VEL   -   Plot velocities only for each specified node and degree of freedom

   ACEL   -   Plot accelerations only for each specified node and degree of freedom

   ALL   -   Plot displacement, velocity, and acceleration for each specified node and degree of freedom.

Format	Columns	Symbol	Comments
A4	1-4		One of the above key words (start in column 1)
	5-10		Blank
14I5	11-80		Pairs of: node number followed by its degree of freedom to be plotted, right justified in fields of 5 columns each. The degree of freedom is specified by a number from 1 through 6 where

   1   denotes global x translation
   2   denotes global y translation
   3   denotes global z translation
   4   denotes global x-rotation
   5   denotes global y-rotation
   6   denotes global z-rotation

NOTES:

1.   Any number of these cards can be specified.

This card is required to end a group of plots having the same plotting parameters.

Format	Columns	Symbol	Comments
A4	1-4	SEND	Key word
	5-80		Blank

NOTES:

1.   Plotting parameters appear in groups starting with a PLOT card and ending with a SEND card. Within a group, the XLEN, XSCALE, YSCALE, and Plot Type card sets define the parameters for a group of plots. These card sets can appear in any order within the group. As many groups as desired may be used, one after the other, in an input deck.

/Key Word STOP/

The last card in the input deck must be an alphanumeric STOP left justified in columns 1 through 4.

Format	Columns	Symbol	Comments
A4	1-4	STOP	Key Word
	5-80		Blank

Example of GRAFIX Inputs:

    Plot displacement, velocity, and acceleration histories in the global z direction
for node 2000. Use automatic scaling. Restart tape goes from 0-50 msec, with last
time step = 253, and acceleration units are in/sec^2.


```
0........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890

GRAPHICS TEST PROBLEM - DYNAMIC ANALYSIS OF        FRAME
PLOT           1 253    2
ALL         2000    3
SEND
STOP
```


A sample plot is shown below.

6.0 EXAMPLE INPUT

This section contains listings of input data for a number of sample problems. The principal intent in providing these sample inputs is to give the user an overall view of some complete representative but simple input decks. Hopefully, these can be used as a supplement to the discussions in Section 3 and 4. Also provided with each problem is a brief discussion of results and representative plots of output data. These can serve as benchmark results for program validations.

A detailed picture of the sample problem idealizations showing node and element labeling has not been provided. It is suggested that the input plotting program SATELLITE be used to generate computer plots of the input data, if desired.

6.1 EXAMPLE PROBLEM NO. 1 - FINITE LENGTH BAR FIXED AT ONE END AND SUBJECTED TO A UNIFORM STEP PULSE AT THE OTHER END

The problem is defined in Fig. 6.1.1 and involves a rod fixed at one end and subjected at the other end to a step pulse of 10,000 stress units. Twenty equal size stringer elements were used with 21 nodes numbered consecutively from 1 to 21. Node 21 is at the fixed end and node 1 is at the point of application of the step pulse. In order to demonstrate DYCAST input four input listings are provided. These are inputs for the basic problem with the following variations:

o Explicit modified Adams integrator used with lumped mass, elastic material.

o Integrator changed to central difference

o Integrator changed to Newmark-β and consistent mass. Material now elastic, perfectly plastic (σ_{yield} = 9000)

o Beam elements used instead of stringer elements for case 1.

Figure 6.1.2 shows the input list for case 1. Note the keyword ADAM in Group A data indicating that the Modified Adams integrator is used. Also shown in Group A is the keyword LUMP. When used with an explicit integrator this indicates a diagonal mass matrix. As a result of this, lumped masses must be input with CNM1 cards in Group G data. The analysis is elastic so that the yield stress on the MSTG card (line 69) in Group H data is set as an arbitrarily high number, 9×10^9. The input deck for case 2 is shown in Fig. 6.1.3. Here the central difference integrator is used so the only change is to replace the ADAM card with CDIF. Figure 6.1.4 shows the input listing using the Newmark-β integrator and a consistent mass matrix formulation. Card 5 now specifies the keyword NEWM with the following blanks indicating all the default parameters. The LUMP card has been deleted although had it been kept it would have been overridden since a consistent mass formulation is always used with the implicit integrators. This problem assumes an elastic, perfectly plastic material, indicated by the blank/or zero in the third and fourth field of the MSTG card (card 62) and a yield stress of 9000.

The last example, case 4, is shown in Fig. 6.1.5. This case is similar to case 1 differing only in its use of beam elements instead of stringer elements. Thus all the Group C data specify BEAM elements. Because the beam element requires an additional node to set the orientation of the cross section, each member card specifies node 22. This node is offset in the global z direction from node 1. The use of this additional node requires its specification in Group B data, its coordinates in Group D and constraints in Group E. The node is completely fixed since it serves only to set the initial orientation of the element. The only other changes are in Group H data where material properties are specified for the beam with the keyword MBM, and the section properties are specified with a solid rectangular section, SREC. Any section with appropriate properties could have been used, however, since the analysis is elastic. Results are shown in Figs. 6.1.6 to 6.1.13. These show the bar stress at one-half and one full transit time. Results are shown for case 1 and also for this case with an elastic perfectly plastic material. In this case the yield stress was set to 9000 on card 69. Figures 6.1.10

and 6.1.11 show results for case 3 using the implicit Newmark-β integrator. Figures 6.1.12 and 6.1.13 show results for a 40 element model with linear strain hardening. Here an elastic and plastic wave front is seen.
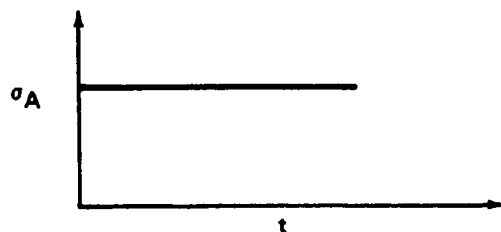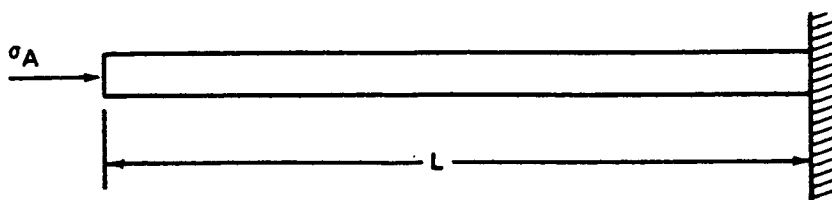
L = 40   20 EQUAL SIZE ELEMENTS

E = 1.0×10^7 ONE TRANSIT IS 20.35×10^{-5} s

$\rho = 2.588 \times 10^{-4}$

WAVE SPEED IS: $C = \sqrt{\dfrac{E}{\rho}}$

Fig. 6.1.1   Finite Length Bar Fixed at One End and Subjected to a Uniform Step Pulse at the Other End

```
CARD         ........1.........2.........3.........4.........5.........6.........7.........8
COUNT        12345678901234567890123456789012345678901234567890123456789012345678901234567890


 1-     DYCAST CHECK PROBLEM  1 UNIFORM BAR STEP PULSE LUMPED MASS ELASTIC              ,
 2-                    MODIFIED ADAMS...
 3-     DYNA 0000    5 100
 4-     DYNA      1.0  .          20.5  E-05  5.00    E-06      0.0
 5-     ADAM      .05               .01            .0001
 6-     LUMP
 7-     TIME
 8-     NPRIN
 9-        1  -20
10-     NPRIN
11-        1  -21
12-     SEND
13-        1  -21
14-     STRG         1    1    2
15-     STRG         2    2    3
16-     STRG         3    3    4
17-     STRG         4    4    5
18-     STRG         5    5    6
19-     STRG         6    6    7
20-     STRG         7    7    8
21-     STRG         8    8    9
22-     STRG         9    9   10
23-     STRG        10   10   11
24-     STRG        11   11   12
25-     STRG        12   12   13
26-     STRG        13   13   14
27-     STRG        14   14   15
28-     STRG        15   15   16
29-     STRG        16   16   17
30-     STRG        17   17   18
31-     STRG        18   18   19
32-     STRG        19   19   20
33-     STRG        20   20   21
34-     SEND
35-     GRDX      0.0           1
36-     GRDX      2.0           2
37-     GRDX      4.0           3
38-     GRDX      6.0           4
39-     GRDX      8.0           5
40-     GRDX     10.0           6
41-     GRDX     12.0           7
42-     GRDX     14.0           8
43-     GRDX     16.0           9
44-     GRDX     18.0          10
45-     GRDX     20.0          11
46-     GRDX     22.0          12


        ........1.........2.........3.........4.........5.........6.........7.........8
        12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

Fig. 6.1.2   Example Problem No. 1 - Input Listing Case 1 (Sheet 1 of 2)

PROGRAM LISTING OF INPUT DATA CARDS

```
CARD       .........1.........2.........3.........4.........5.........6.........7.........8
COUNT      12345678901234567890123456789012345678901234567890123456789012345678901234567890

  47-   GRDX    24.0        13
  48-   GRDX    26.0        14
  49-   GRDX    28.0        15
  50-   GRDX    30.0        16
  51-   GRDX    32.0        17
  52-   GRDX    34.0        18
  53-   GRDX    36.0        19
  54-   GRDX    38.0        20
  55-   GRDX    40.0        21
  56-   GRDY     0.0         1  -21
  57-   GRDZ     0.0         1  -21
  58-   SEND
  59-   SPC  000000         21
  60-   SPC  100000          1  -20
  61-   SEND
  62-   CNM1  2.5879918 E-04
  63-   CNM1
  64-        1   21
  65-   CNM1  5.1759836E-04
  66-   CNM1
  67-        2  -20
  68-   SEND
  69-   MSTG          1.E 7          1.                         0.    9.  E09
  70-        0.0
  71-        1  -20
  72-   SEND
  73-   CONC    1    1.0   E 04
  74-   CONC
  75-   PTME         1.0          0.0
  76-   PTME         1.0        5.0   E-03
  77-   SEND
  78-   STOP
```

```
           .........1.........2.........3.........4.........5.........6.........7.........8
           12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

Fig. 6.1.2 Example Problem No. 1 - Input Listing Case 1 (Sheet 2 of 2)

6.5

```
CARD      .........1.........2.........3.........4.........5.........6.........7.........8
COUNT     12345678901234567890123456789012345678901234567890123456789012345678901234567890


  1-      DYCAST CHECK PROBLEM  2 UNIFORM BAR STEP PULSE LUMPED MASS ELASTIC            ,
  2-                   CENTRAL DIFFERENCE INTEGRATOR...
  3-      DYNA  0000     5  100
  4-      DYNA      1.0            20.5  E-05  5.00   E-06    0.0
  5-      CDIF
  6-      LUMP
  7-      TIME
  8-      NPRIN
  9-         1  -20
 10-      NPRIN
 11-         1  -21
 12-      SEND
 13-         1  -21
 14-      STRG       1    1    2
 15-      STRG       2    2    3
 16-      STRG       3    3    4
 17-      STRG       4    4    5
 18-      STRG       5    5    6
 19-      STRG       6    6    7
 20-      STRG       7    7    8
 21-      STRG       8    8    9
 22-      STRG       9    9   10
 23-      STRG      10   10   11
 24-      STRG      11   11   12
 25-      STRG      12   12   13
 26-      STRG      13   13   14
 27-      STRG      14   14   15
 28-      STRG      15   15   16
 29-      STRG      16   16   17
 30-      STRG      17   17   18
 31-      STRG      18   18   19
 32-      STRG      19   19   20
 33-      STRG      20   20   21
 34-      SEND
 35-      GRDX     0.0         1
 36-      GRDX     2.0         2
 37-      GRDX     4.0         3
 38-      GRDX     6.0         4
 39-      GRDX     8.0         5
 40-      GRDX    10.0         6
 41-      GRDX    12.0         7
 42-      GRDX    14.0         8
 43-      GRDX    16.0         9
 44-      GRDX    18.0        10
 45-      GRDX    20.0        11
 46-      GRDX    22.0        12
```

```
          .........1.........2.........3.........4.........5.........6.........7.........8
          12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

Fig. 6.1.3 Example Problem No. 1 - Partial Listing of Input - Case 2

PROGRAM LISTING OF INPUT DATA CARDS

```
CARD    ..........1.........2.........3.........4.........5.........6.........7.........8
COUNT   12345678901234567890123456789012345678901234567890123456789012345678901234567890

   1-    DYCAST CHECK PROBLEM 10 UNIFORM BAR STEP PULSE PLASTIC                  ,
   2-                         NEWMARK BETA INTEGRATOR...
   3-    DYNA  0000     5    1
   4-    DYNA       1.0          20.5   E-05   5.00   E-06    0.0
   5-    NEWM
   6-    TIME
   7-    MPRIN
   8-        1  -20
   9-    NFRIN
  10-        1  -21
  11-    SEND
  12-        1  -21
  13-    STRG         1    1    2
  14-    STRG         2    2    3
  15-    STRG         3    3    4
  16-    STRG         4    4    5
  17-    STRG         5    5    6
  18-    STRG         6    6    7
  19-    STRG         7    7    8
  20-    STRG         8    8    9
  21-    STRG         9    9   10
  22-    STRG        10   10   11
  23-    STRG        11   11   12
  24-    STRG        12   12   13
  25-    STRG        13   13   14
  26-    STRG        14   14   15
  27-    STRG        15   15   16
  28-    STRG        16   16   17
  29-    STRG        17   17   18
  30-    STRG        18   18   19
  31-    STRG        19   19   20
  32-    STRG        20   20   21
  33-    SEND
  34-    GRDX      0.0            1
  35-    GRDX      2.0            2
  36-    GRDX      4.0            3
  37-    GRDX      6.0            4
  38-    GRDX      8.0            5
  39-    GRDX     10.0            6
  40-    GRDX     12.0            7
  41-    GRDX     14.0            8
  42-    GRDX     16.0            9
  43-    GRDX     18.0           10
  44-    GRDX     20.0           11
  45-    GRDX     22.0           12
  46-    GRDX     24.0           13

        ..........1.........2.........3.........4.........5.........6.........7.........8
        12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

Fig. 6.1.4   Example Problem No. 1 - Listing of Input - Case 3 (Sheet 1 of 2)

6.7

```
47-     GRDX      26.0           14
48-     GRDX      28.0           15
49-     GRDX      30.0           16
50-     GRDX      32.0           17
51-     GRDX      34.0           18
52-     GRDX      36.0           19
53-     GRDX      38.0           20
54-     GRDX      40.0           21
55-     GRDY       0.0            1   -21
56-     GRDZ       0.0            1   -21
57-     SEND
58-     SPC    000000           21
59-     SPC    100000            1   -20
60-     SEND
61-     SEND
62-     MSTG            1.E 7              1.              0.      9.  E03
63-          2.588E-04
64-        1  -20
65-     SEND
66-     CONC      1    1.0  E 04
67-     CONC
68-     FTME            1.0            0.0
69-     PTME            1.0            5.0    E-03
70-     SEND
71-     STOP
```

Fig. 6.1.4   Example Problem No. 1 - Listing of Input - Case 3 (Sheet 2 of 2)

PROGRAM LISTING OF INPUT DATA CARDS

```
CARD    .........1.........2.........3.........4.........5.........6.........7.........8
COUNT   12345678901234567890123456789012345678901234567890123456789012345678901234567890

 1-     DYCAST CHECK PROBLEM 20 UNIFORM BAR STEP PULSE LUMPED MASS ELASTIC
 2-                             MODIFIED ADAMS INTEGRATOR BEAM ELEMENT...           ,
 3-     DYNA          1  100
 4-     DYNA      1.0            20.5  E-05    5.0    E-06     0.0
 5-     ADAM      .05            .01           .0001
 6-     LUMP
 7-     TIME
 8-     NPRIN
 9-         1  -21
10-     MPRIN
11-         1  -20
12-     SEND
13-         1  -22
14-     BEAM           1    1    2   22
15-     BEAM           2    2    3   22
16-     BEAM           3    3    4   22
17-     BEAM           4    4    5   22
18-     BEAM           5    5    6   22
19-     BEAM           6    6    7   22
20-     BEAM           7    7    8   22
21-     BEAM           8    8    9   22
22-     BEAM           9    9   10   22
23-     BEAM          10   10   11   22
24-     BEAM          11   11   12   22
25-     BEAM          12   12   13   22
26-     BEAM          13   13   14   22
27-     BEAM          14   14   15   22
28-     BEAM          15   15   16   22
29-     BEAM          16   16   17   22
30-     BEAM          17   17   18   22
31-     BEAM          18   18   19   22
32-     BEAM          19   19   20   22
33-     BEAM          20   20   21   22
34-     SEND
35-     GRDX      0.0            1   22
36-     GRDX      2.0            2
37-     GRDX      4.0            3
38-     GRDX      6.0            4
39-     GRDX      8.0            5
40-     GRDX     10.0            6
41-     GRDX     12.0            7
42-     GRDX     14.0            8
43-     GRDX     16.0            9
44-     GRDX     18.0           10
45-     GRDX     20.0           11
46-     GRDX     22.0           12

        .........1.........2.........3.........4.........5.........6.........7.........8
        12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

Fig. 6.1.5   Example Problem No. 1 - Listing of Input - Case 4 (Sheet 1 of 2)

```
CARD     .........1.........2.........3.........4.........5.........6.........7.........8
COUNT    12345678901234567890123456789012345678901234567890123456789012345678901234567890

 47-     GRDX     24.0          13
 48-     GRDX     26.0          14
 49-     GRDX     28.0          15
 50-     GRDX     30.0          16
 51-     GRDX     32.0          17
 52-     GRDX     34.0          18
 53-     GRDX     36.0          19
 54-     GRDX     38.0          20
 55-     GRDX     40.0          21
 56-     GRDY     0.0            1   -22
 57-     GRDZ     0.0            1   -21
 58-     GRDZ     10.0          22
 59-     SEND
 60-     SFC   000000            1   -22
 61-     SFC   100000            1   -20
 62-     SEND
 63-     CNM1  2.5879918 E-04
 64-     CNM1
 65-          1    21
 66-     CNM1  5.1759836 E-04
 67-     CNM1
 68-          2   -20
 69-     SEND
 70-     MBM     1.0     E 07                   0.0           0.0           9.0    E 03
 71-          2.586 E-04
 72-          1   -20
 73-     SREC
 74-          0.0           0.0           0.0
 75-          1.0           1.0
 76-          1   -20
 77-     SEND
 78-     CONC     1    1.0  E 04
 79-     CONC
 80-     PTME           1.0           0.0
 81-     PTME           1.0           5.0   E-03
 82-     SEND
 83-     STOP
```

```
.........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

Fig. 6.1.5 Example Problem No. 1 – Listing of Input – Case 4 (Sheet 2 of 2)
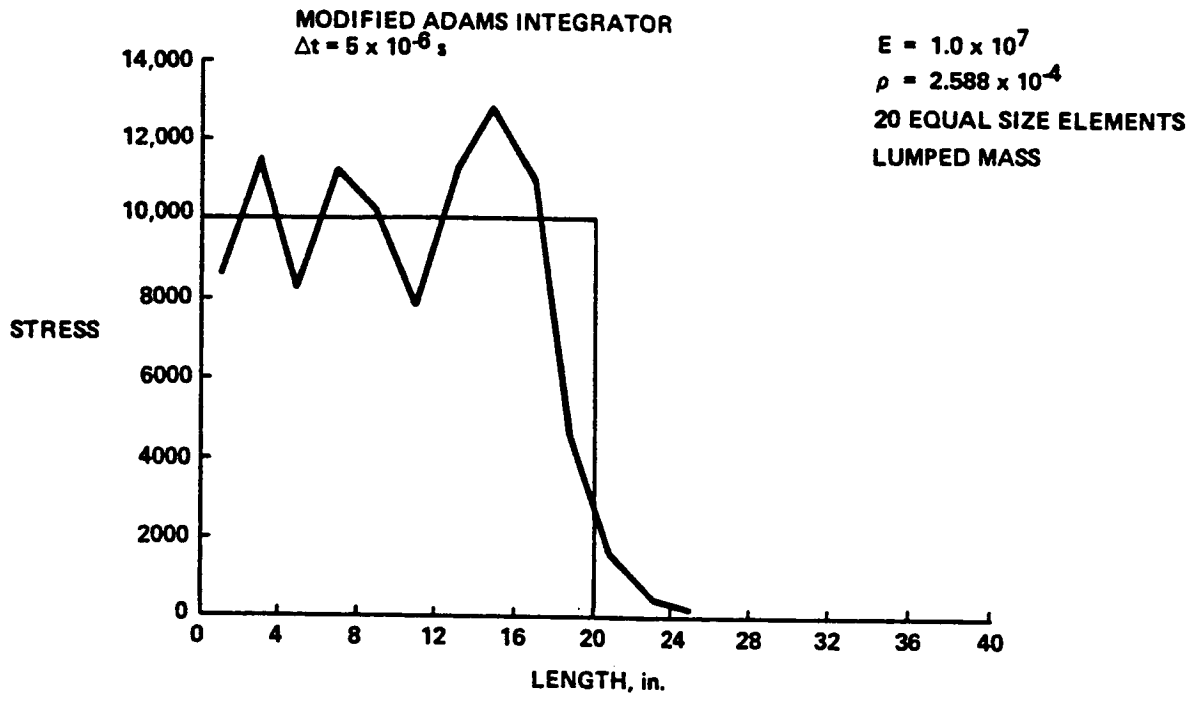
Fig. 6.1.6   Finite Length Bar Fixed at One End Subjected to a Step Pulse.   Time
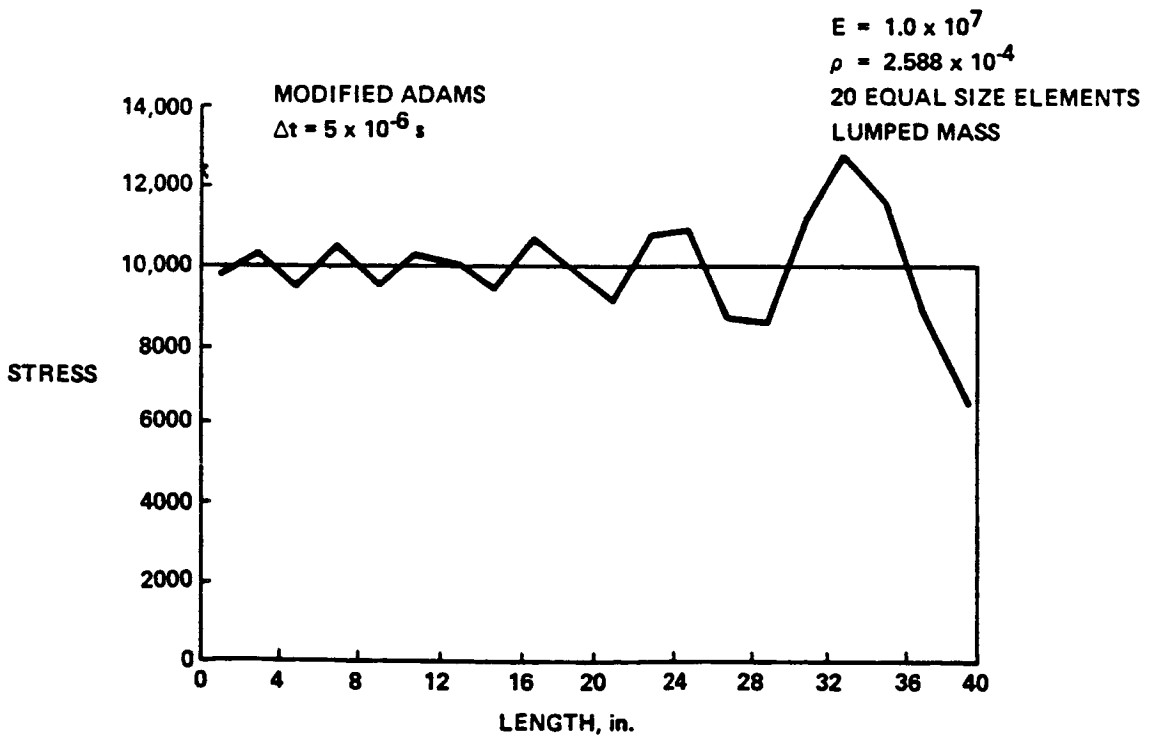Equals One Half Transit.   Case 1 Data Used.

Fig. 6.1.7   Finite Length Bar Fixed at One End Subjected to a Step Pulse.   Time
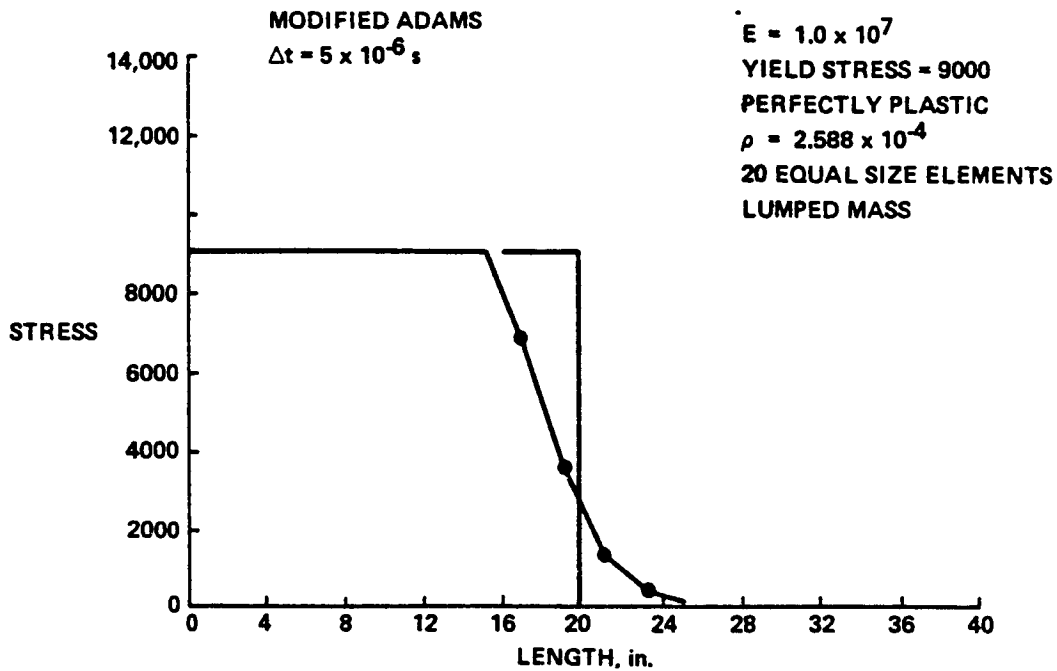Equals One Wave Transit.   Case 1 Data Used.

6.11

Fig. 6.1.8 Finite Length Bar Fixed at One End Subjected to a Step Pulse. Time Equals One-Half Transit. Case 1 Data Used for an Elastic Perfectly Plastic Material.
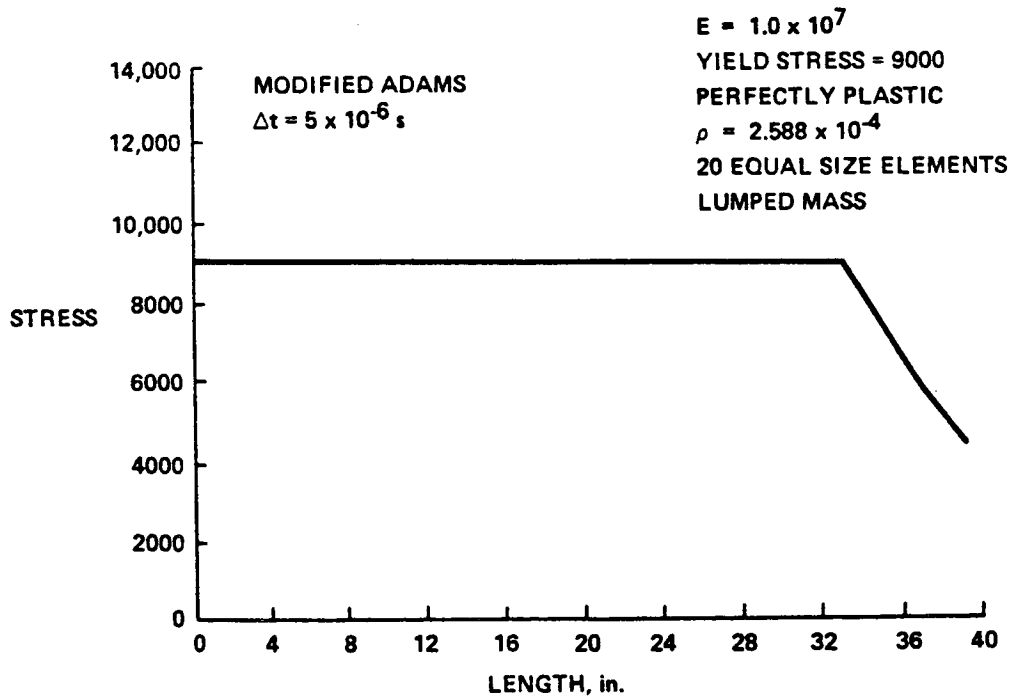

Fig. 6.1.9 Finite Length Bar Fixed at One End Subjected to a Step Pulse. Time Equals One Wave Transit. Case 1 Data Used with an Elastic Perfectly Plastic Material.
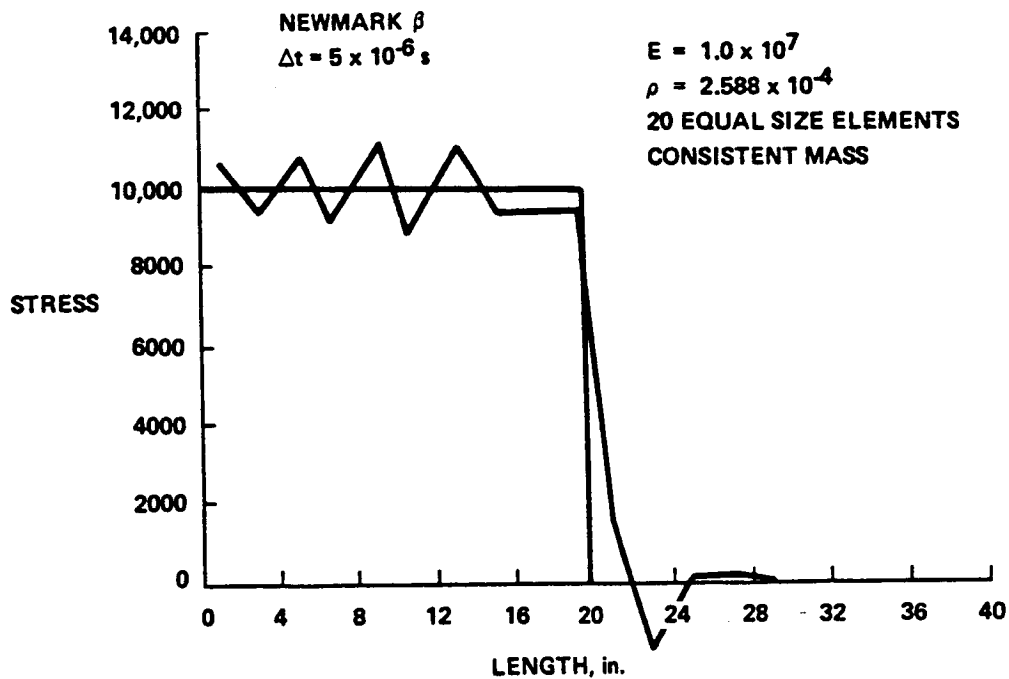
6.12

Fig. 6.1.10   Finite Length Bar Fixed at One End Subjected to a Step Pulse.   Time
Equals One-Half Transit.   Case 3 Data (Elastic).
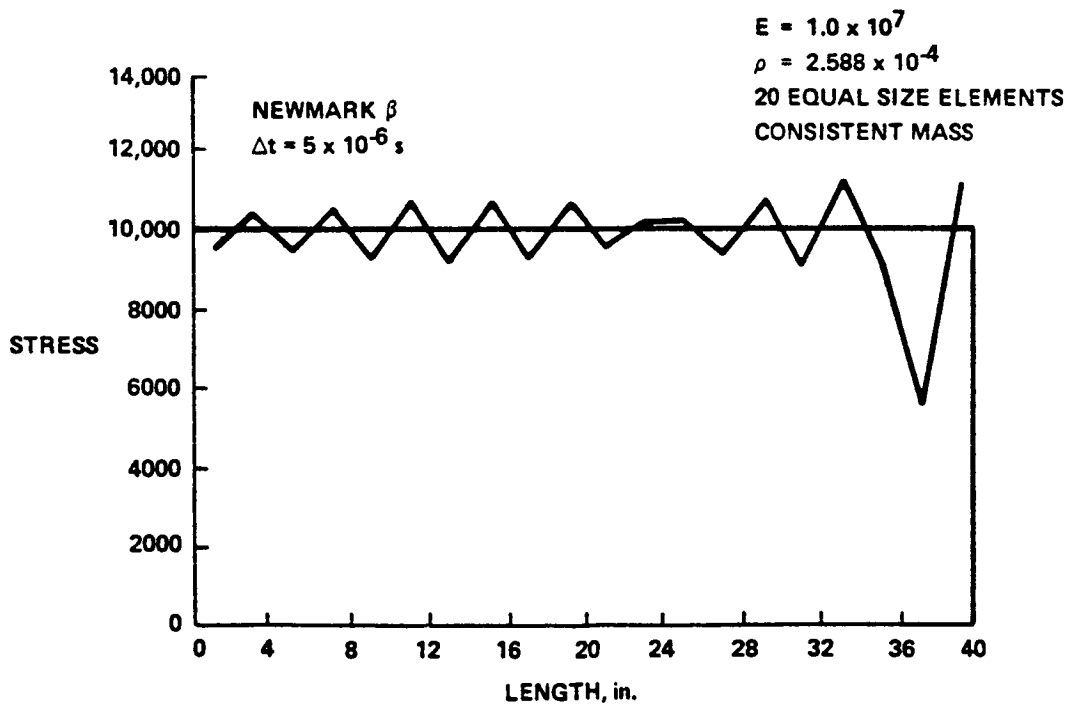
Fig. 6.1.11   Finite Length Bar Fixed at One End Subjected to a Step Pulse.   Time
Equals One Wave Transit.   Case 3 Data Used (Elastic).

6.13

Fig. 6.1.12   Finite Length Bar Fixed at One End Subjected to a Step Pulse.   Time Equals One-Half Transit.   Forty Equal Sized Elements Used and Linear Strain Hardening.
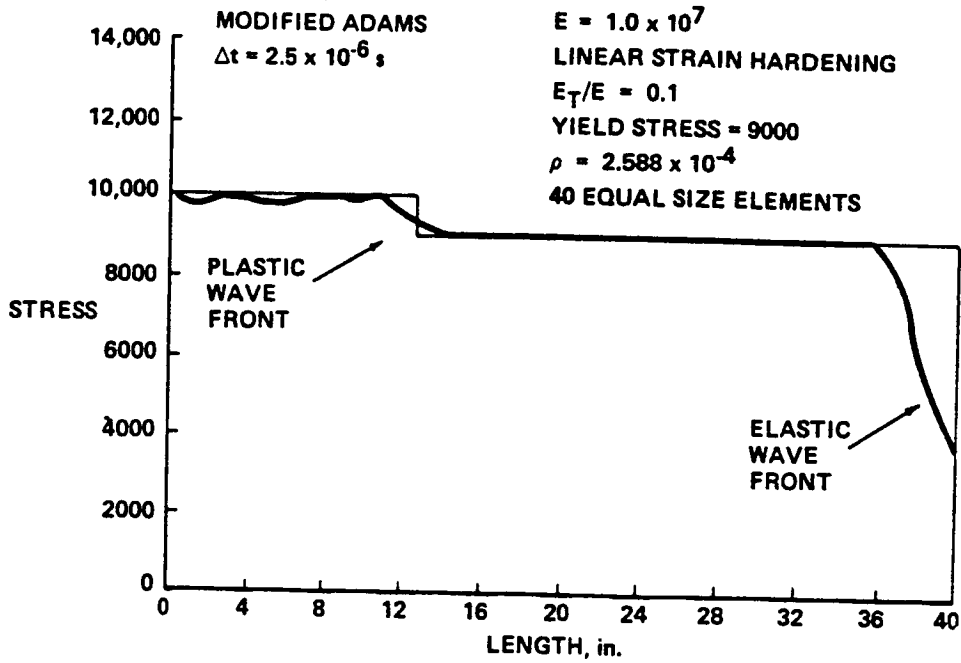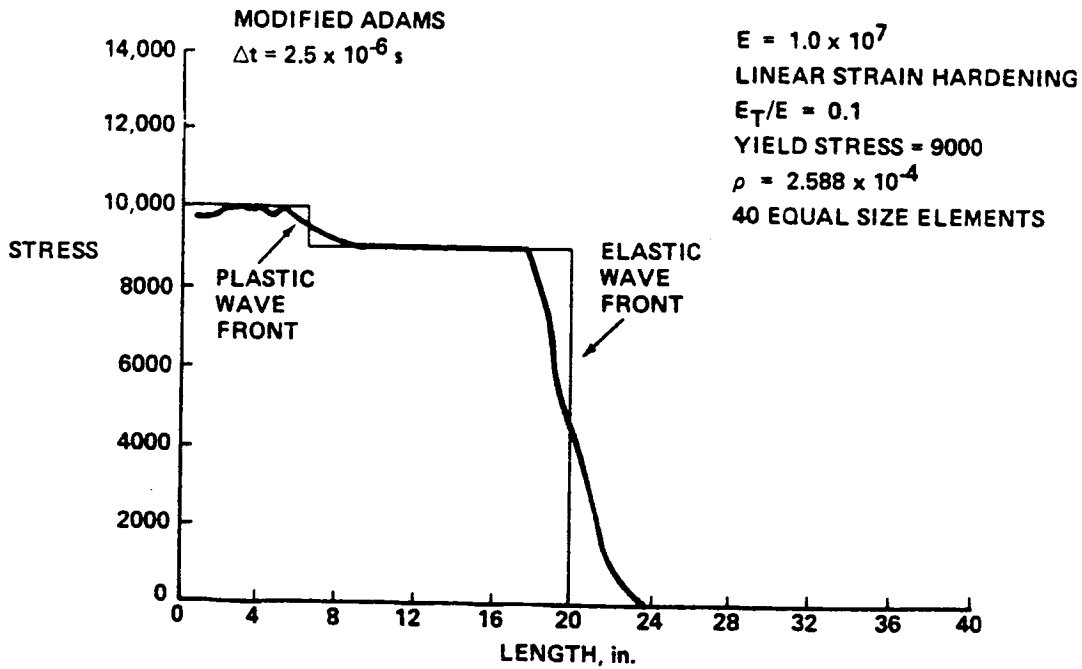

Fig. 6.1.13   Finite Length Bar Fixed at One End Subjected to a Step Pulse.   Time Equals One Wave Transit.   Forty Equal Sized Elements Used and Linear Strain Hardening

6.2 EXAMPLE PROBLEM NO. 2 - SINGLE MASS AND ENERGY ABSORBING SPRING IMPACTING A RIGID BARRIER

The problem is defined in Fig. 6.2.1 and involves a mass with an energy absorbing spring contacting a rigid barrier. Contact is simulated by a gap element. The mass is initially allowed to translate as a rigid body until contact is made. Once this occurs the capture mechanism discussed in Section 4.3.5.4 is initiated.

The input is listed in Fig. 6.6.2 and specifies the Newmark-β integrator. Two spring elements are used, one for the energy absorber and one for the gap. The masses are specified by the keyword CNM1 on Group G input.

Results are shown in Fig. 6.2.3 and 6.2.4. Comparison with an exact solution is quite good. The error is due to an initial overshoot that allows the node impacting the barrier to overshoot the contact point. This causes a lag in the displacement and velocity response. It should be noted that if the capture mechanism was not initiated the initially large capture force, due to the large slope of the gap element, would have caused node 2 to be repelled from the barrier at a very large velocity, rendering the subsequent results meaningless.
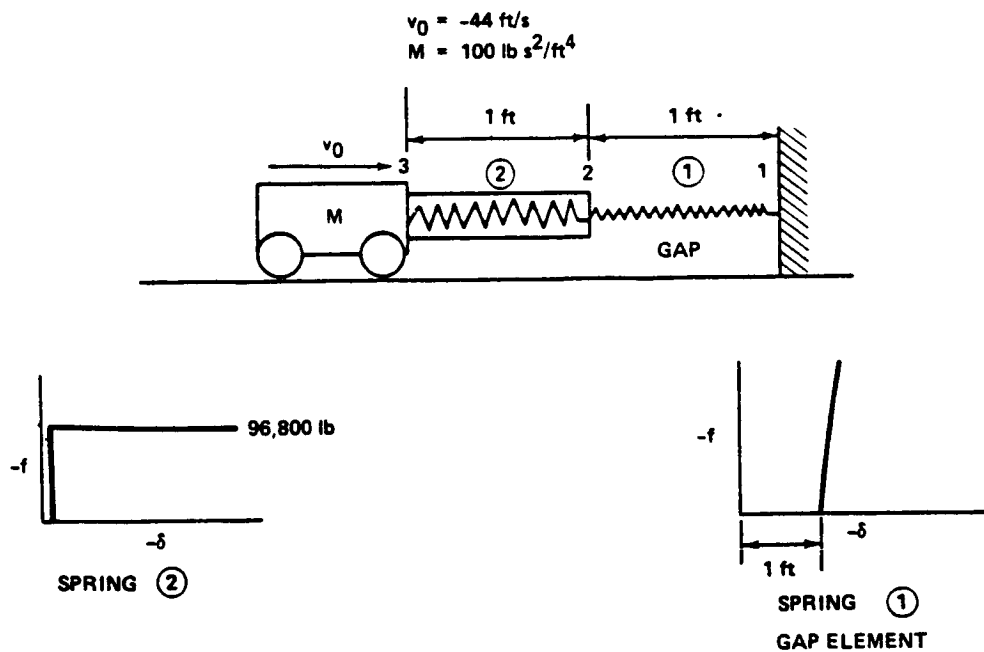

Fig. 6.2.1 Mass with an Energy Absorbing Spring Impacting a Rigid Barrier. Contact Simulated with a Gap Element.

```
CARD        .........1.........2.........3.........4.........5.........6.........7.........8
COUNT       12345678901234567890123456789012345678901234567890123456789012345678901234567890


 1-    DYCAST CHECK PROBLEM 43 TWO MASS IMPACT MODEL DAMPED GAP ELEMENT
 2-    DYNA     0    1    1
 3-    DYNA     1.0           .06827          .002275         0.0
 4-    NEWM
 5-    SEND
 6-          1    2    3
 7-    SPNG              1    1    2
 8-    SPNG              2    2    3
 9-    SEND
10-    GRDX     0.0           1
11-    GRDX     2.0           2
12-    GRDX     4.0           3
13-    GRDY     0.0           1    2    3
14-    GRDZ     0.0           1    2    3
15-    SEND
16-    SFC  100000           2    3
17-    SFC  000000           1
18-    SEND
19-    IVEL     -44.0
20-    IVEL
21-          2    3
22-    SEND
23-    CNM1     100.0
24-    CNM1
25-          3
26-    CNM1     1.0
27-    CNM1
28-          2
29-    SEND
30-    PSPR     5.0           4.2229729 E+08                                   100.0
31-    PSPR     -1.875  E+07  -1.0444
32-    PSPR     -9.375E+06    -1.0222
33-    PSPR     0.0           -1.0
34-    PSPR     0.0           0.0
35-    PSPR     0.0           1.0
36-          1
37-    PSPR     6.0
38-    PSPR     -96800.0      -1.0
39-    PSPR     -96800.0      -0.0001
40-    PSPR     0.0           -0.00005
41-    PSPR     0.0           0.0
42-    PSPR     0.0           .001
43-    PSPR     0.0           1.0
44-          2
45-    SEND
46-    SEND
```

```
.........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

Fig. 6.2.2   Example Problem No. 2 - Input Listing (Sheet 1 of 2)

PROGRAM LISTING OF INPUT DATA CARDS

```
CARD     .........1.........2.........3.........4.........5.........6.........7.........8
COUNT    12345678901234567890123456789012345678901234567890123456789012345678901234567890

  47-    STOP
```

```
.........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

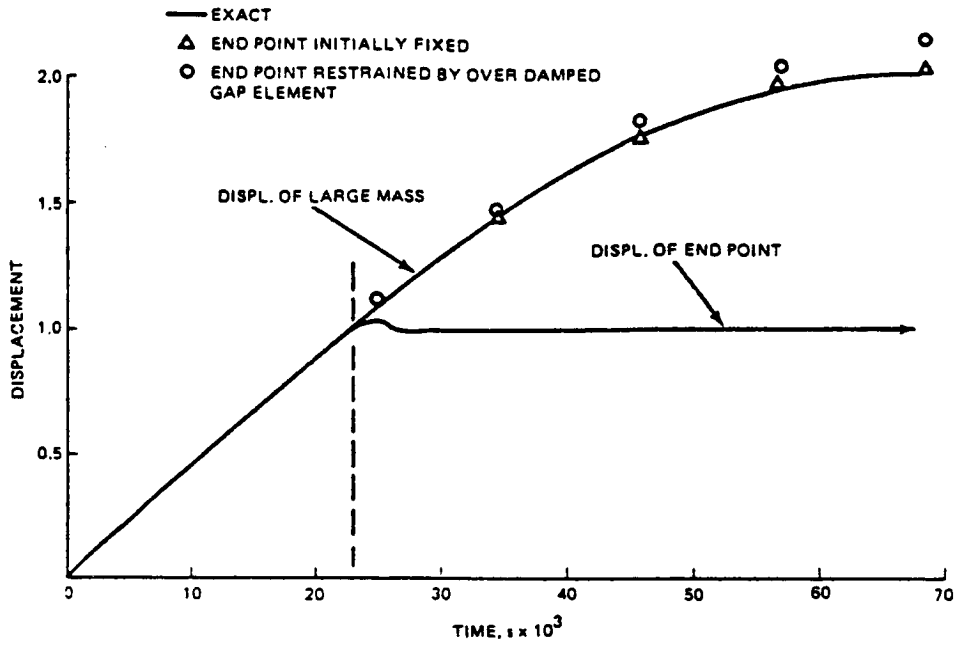Fig. 6.2.2 Example Problem No. 2 - Input Listing (Sheet 2 of 2)

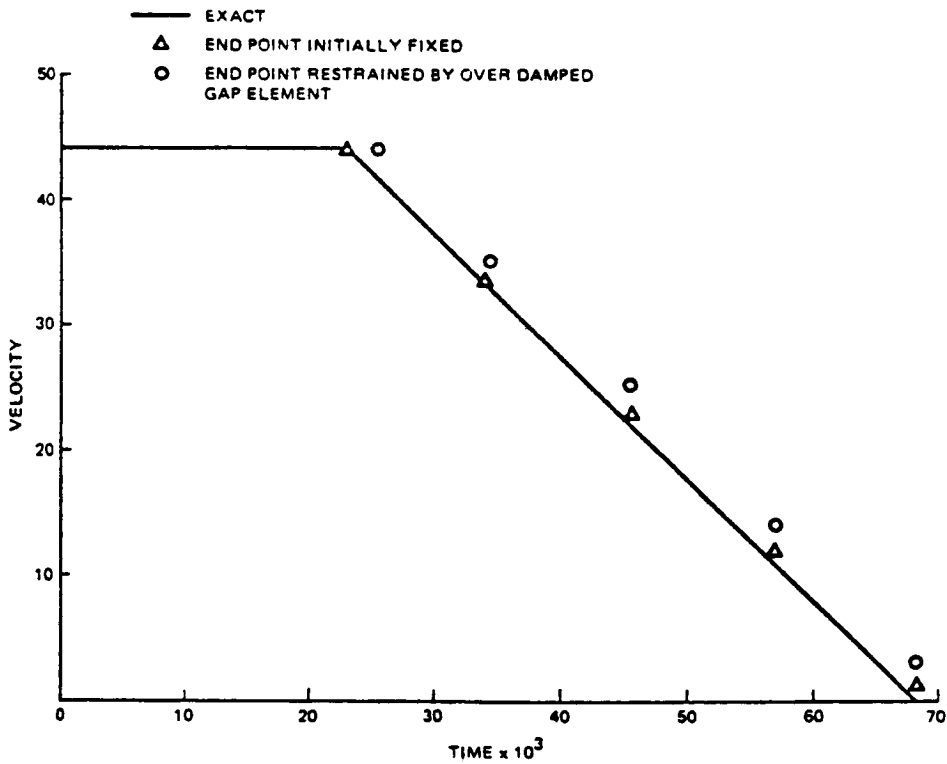Fig. 6.2.3 Displacement Versus Time for Example Problem No. 2

Fig. 6.2.4 Velocity Versus Time for Example Problem No. 2

6.3 EXAMPLE PROBLEM NO. 3 - UNIFORMLY LOADED RESTRAINED BEAM

The problem is defined in Fig. 6.3.1 and involves a restrained beam, clamped at its edges and subjected to a uniform lateral load. Ten beam elements are used for half the beam. The input listing is shown in Fig. 6.3.2. Note that since it is a static analysis the keyword STAT is used in Group A data. The applied load is input using the beam load keyword, BMLO in Group I data. However, static loads still require that the table of load factor versus time be specified. For static analyses the time can be interpreted as a parameter. In the input shown, the maximum load factor is 100 with a time step of 16. This leads to a load step of 960 and a total load of 6000 since the unit values as prescribed on the BMLO input is 60. Results are shown in Fig. 6.3.3 and 6.3.4. These figures show the central deflection and axial load versus applied load. Results are plotted for two loads steps, $\Delta P = 960$ and $\Delta P = 240$. In both cases, they compare identically with an exact solution. Results are also shown (Fig. 6.3.5 and 6.3.6) for the case in which the edges are simply supported. The only change in input for this case is card 35 (Fig. 6.3.2) defining the SPC for node 1. Results are shown for $\Delta P = 60$ and $\Delta P = 240$. There is, however, a fundamental difference in the solution. In the clamped case the slope of the load versus deflection curve, as the beam deforms into the nonlinear range, changes gradually from the initial linear slope. The analysis consequently requires a number of iterations through this region until the flat portion is reached. However, as seen in the simply supported case (Fig. 6.3.5), the change in slope in this region is more pronounced. Consequently, no matter how many iterations are allowed, the solution does not converge until the step size has been reduced to $\Delta P = 60$. This step is required through the highly nonlinear region after which it increases to the prescribed maximum value. Because of this, an iteration limit was taken as 10 so that unnecessary calculations would not be performed in the highly nonlinear region. Again, results compare identically with an exact solution.
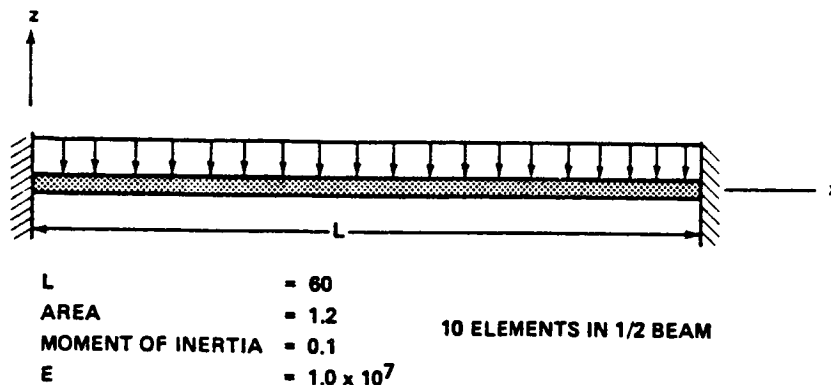

L	= 60
AREA	= 1.2
MOMENT OF INERTIA	= 0.1
E	= 1.0×10^7

10 ELEMENTS IN 1/2 BEAM

Fig. 6.3.1   Uniformly Loaded Restrained Clamped Beam - Static Analysis

```
CARD       .........1.........2.........3.........4.........5.........6.........7.........8
COUNT      12345678901234567890123456789012345678901234567890123456789012345678901234567890

  1-   DYCAST SAMPLE PROBLEM 26B UNIFORMLY LOADED CLAMPED RESTRAINED BEAM STATIC ELASTIC
  2-   DYNA  0000    1   1
  3-   DYNA                        100.0           16.0          0.0
  4-   STAT                                         .01          25.0
  5-   TIME
  6-   SEND
  7-         1  -12
  8-   BEAM          1    1    2   12
  9-   BEAM          2    2    3   12
 10-   BEAM          3    3    4   12
 11-   BEAM          4    4    5   12
 12-   BEAM          5    5    6   12
 13-   BEAM          6    6    7   12
 14-   BEAM          7    7    8   12
 15-   BEAM          8    8    9   12
 16-   BEAM          9    9   10   12
 17-   BEAM         10   10   11   12
 18-   SEND
 19-   GRDX        0.0   E 00    1   12
 20-   GRDX        3.0   E 00    2
 21-   GRDX        6.0   E 00    3
 22-   GRDX        9.0   E 00    4
 23-   GRDX       12.0   E 00    5
 24-   GRDX       15.0   E 00    6
 25-   GRDX       18.0   E 00    7
 26-   GRDX       21.0   E 00    8
 27-   GRDX       24.0   E 00    9
 28-   GRDX       27.0   E 00   10
 29-   GRDX       30.0   E 00   11
 30-   GRDZ        0.0   E 00    1  -12
 31-   GRDY        0.0   E 00    1  -11
 32-   GRDY       10.0   E 00   12
 33-   SEND
 34-   SPC   000000          12
 35-   SPC   000000           1
 36-   SPC   101010           2  -10
 37-   SPC   001000          11
 38-   SEND
 39-   ABM         1.0   E 07    .25                                          3.0   E 09
 40-         0.0             0.0
 41-         1  -10
 42-   TUB
 43-         0.0             0.0           0.0
 44-         1.2             1.0           8.0
 45-         1  -10
 46-   SEND

           .........1.........2.........3.........4.........5.........6.........7.........8
           12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

Fig. 6.3.2   Example Problem No. 3 - Input Listing (Sheet 1 of 2)

PROGRAM LISTING OF INPUT DATA CARDS

```
          .........1.........2.........3.........4.........5.........6.........7.........8
          12345678901234567890123456789012345678901234567890123456789012345678901234567890

  47-   BHLO        0.0         0.0         -1.0        -1.0
  48-       1  -10
  49-   PTHE        0.0         0.0
  50-   PTHE       100.0       100.0
  51-   SEND
  52-   STOP
```

```
          .........1.........2.........3.........4.........5.........6.........7.........8
          12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

Fig. 6.3.2   Example Problem No. 3 - Input Listing (Sheet 2 of 2)
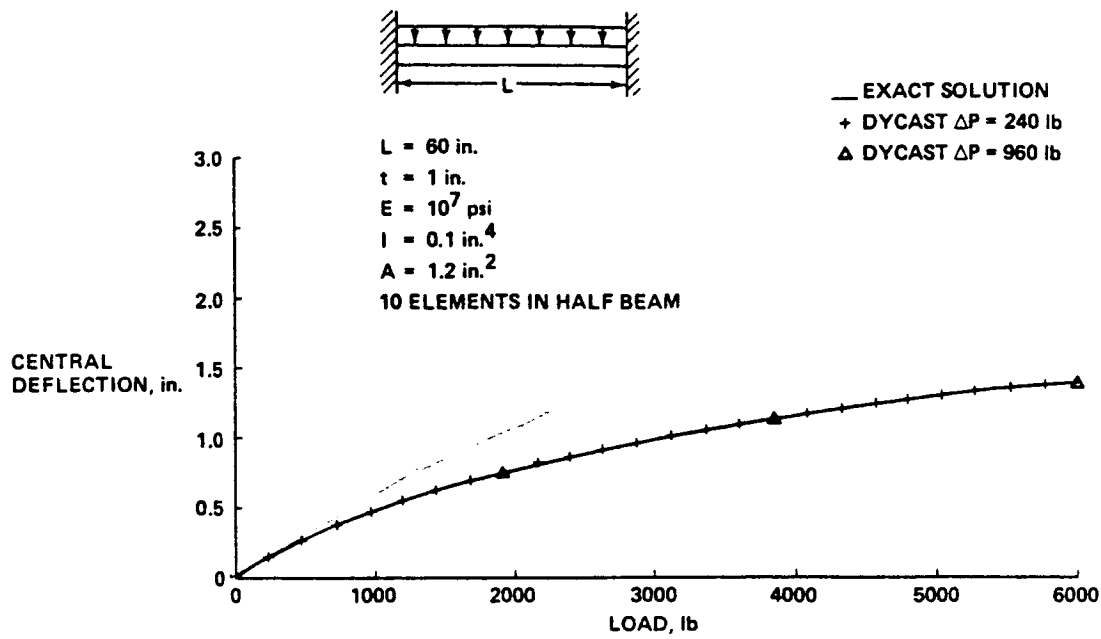
Fig. 6.3.3   Uniformly Loaded Restrained Clamped Beam – Central Deflection
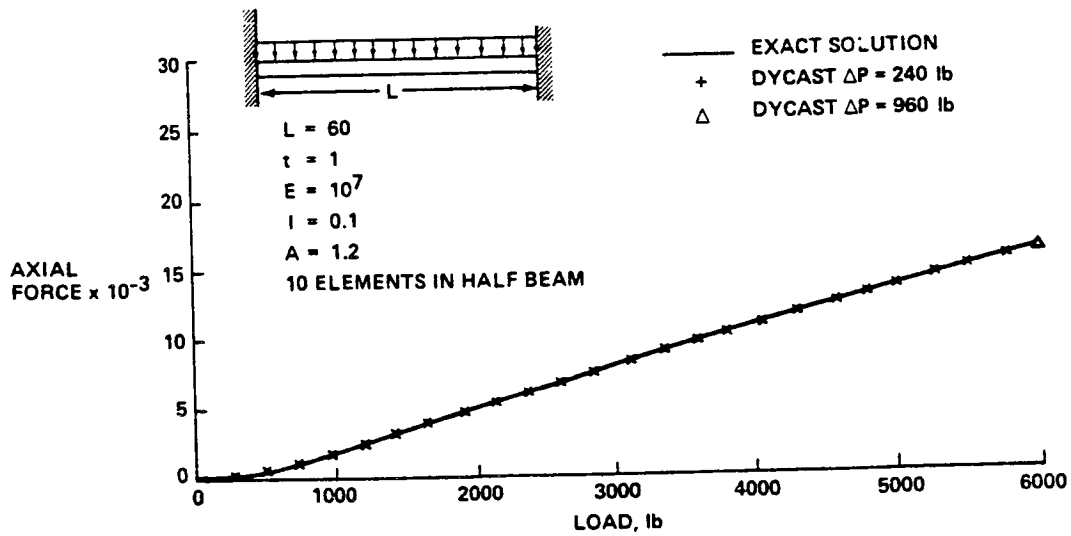   Versus Load

Fig. 6.3.4   Uniformly Loaded Restrained Clamped Beam – Axial Force Versus Load
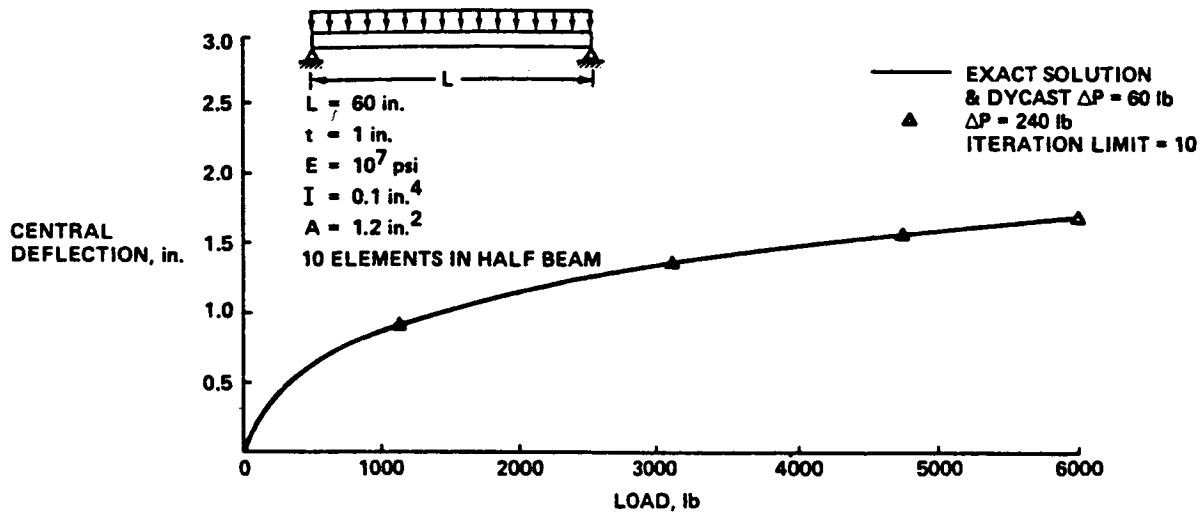
Fig. 6.3.5 Uniformly Loaded Restrained Simply Supported Beam - Central Deflection Versus Load.
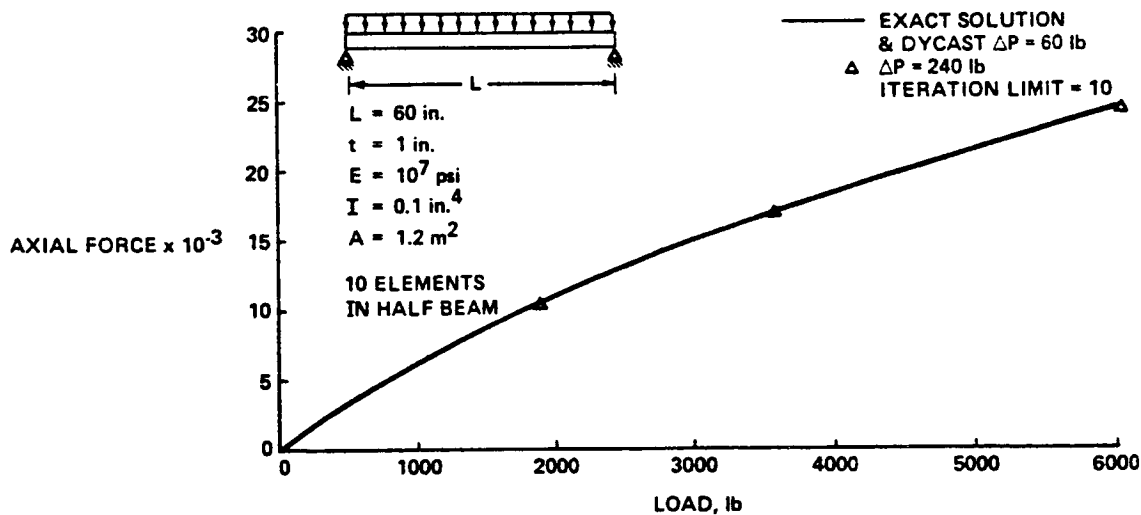

Fig. 6.3.6 Uniformly Loaded Restrained Simply Supported Beam - Axial Force Versus Load

6.23

6.4 EXAMPLE PROBLEM NO. 4 - IMPULSIVELY LOADED CLAMPED BEAM

The problem is defined in Fig. 6.4.1 and involves a rectangular section beam, clamped at both ends, with an impulsively applied load on the center section simulated by an applied initial velocity. The listing of the input is in Fig. 6.4.2. Note the following: A maximum time is set at 1.2 msec with a time step of 10.0×10^{-6} sec. The analysis uses the Newmark-β integrator. Member outputs are to be printed for only 1, 2, and 10. Initial velocities are specified in Group F data with the keyword IVEL. The material is elastic, perfectly plastic with a yield stress of 4.14×10^4 psi.

Figure 6.4.3 shows the results of central deflection versus time. For $\Delta t = 10 \times 10^{-6}$ and $\Delta t = 2 \times 10^{-6}$ s. Figure 6.4.3 indicates that the results are in essential agreement with the maximum differences at later times. The results with the smaller of the two time steps compares favorably with results presented in "Finite Element Analysis of Large Transient Elastic-Plastic Deformations of Simple Structures with Applications to the Engine Rotor Fragment Containment/Deflection Problem," MIT Report ASRL TR 154-4 (1972). It should be noted that results for the two different time steps agree at early times because the initial nonlinearities as seen in Fig. 6.4.3 caused the time step to be reduced to 2.5×10^{-6} s in that region.
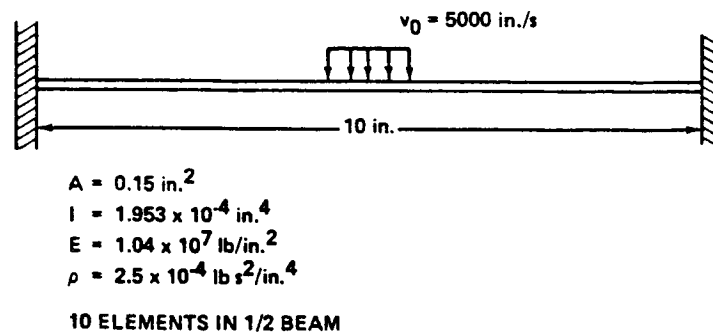

$v_0 = 5000$ in./s

10 in.

A = 0.15 in.^{2}
I = 1.953 x 10^{-4} in.^{4}
E = 1.04 x 10^7 lb/in.^{2}
ρ = 2.5 x 10^{-4} lb s^2/in.^{4}

10 ELEMENTS IN 1/2 BEAM

Fig. 6.4.1 Impulsively Loaded Clamped Beam

PROGRAM LISTING OF INPUT DATA CARDS

```
CARD      .........1.........2.........3.........4.........5.........6.........7.........8
COUNT     12345678901234567890123456789012345678901234567890123456789012345678901234567890

 1-     DYCAST EXAMPLE PROBLEM 37A IMPULSIVELY LOADED BEAM (WU - WITMER)    - PLASTIC   ,
 2-                                   NEWMARK INTEGRATOR...
 3-     DYNA  0000   40 1000
 4-     DYNA      1.0            1.2  E-03   10.00 E-06      0.0
 5-     NEWM       .25            .5          .01           3.0
 6-     CONM
 7-     TIME
 8-     MPRIN
 9-         1    2   10
10-     NPRIN
11-         1  -12
12-     SEND
13-         1  -12
14-     BEAM           1    1    2   12
15-     BEAM           2    2    3   12
16-     BEAM           3    3    4   12
17-     BEAM           4    4    5   12
18-     BEAM           5    5    6   12
19-     BEAM           6    6    7   12
20-     BEAM           7    7    8   12
21-     BEAM           8    8    9   12
22-     BEAM           9    9   10   12
23-     BEAM          10   10   11   12
24-     SEND
25-     GRDX      0.0            1   12
26-     GRDX       .5            2
27-     GRDX      1.0            3
28-     GRDX      1.5            4
29-     GRDX      2.0            5
30-     GRDX      2.5            6
31-     GRDX      3.0            7
32-     GRDX      3.5            8
33-     GRDX      4.0            9
34-     GRDX      4.5           10
35-     GRDX      5.0           11
36-     GRDY      0.0            1  -11
37-     GRDY      1.0           12
38-     GRDZ      0.0            1  -12
39-     SEND
40-     SPC   001000            1
41-     SPC   101010            2  -10
42-     SPC   000000           11   12
43-     SEND
44-     IVEL      0.0           0.0        -.5  E+04
45-     IVEL
46-         1    2

          .........1.........2.........3.........4.........5.........6.........7.........8
          12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

Fig. 6.4.2   Example Problem No. 4 - Input Listing (Sheet 1 of 2)

PROGRAM LISTING OF INPUT DATA CARDS

```
CARD       ..........1.........2.........3.........4.........5.........6.........7.........8
COUNT      12345678901234567890123456789012345678901234567890123456789012345678901234567890


 47-    IVEL    0.0             0.0                 -.25  E+04
 48-    IVEL
 49-          3
 50-    SEND
 51-    SEND
 52-    MBM         1.04  E+07     .3              0.0             0.0             4.14  E+04
 53-          .25     E-03
 54-          1   -10
 55-    TWD
 56-          0.0
 57-          1.2             0.125           6.0
 58-          1   -10
 59-    SEND
 60-    SEND
 61-
 62-
 63-
 64-    STOP
```

```
..........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

Fig. 6.4.2   Example Problem No. 4 - Input Listing (Sheet 2 of 2)
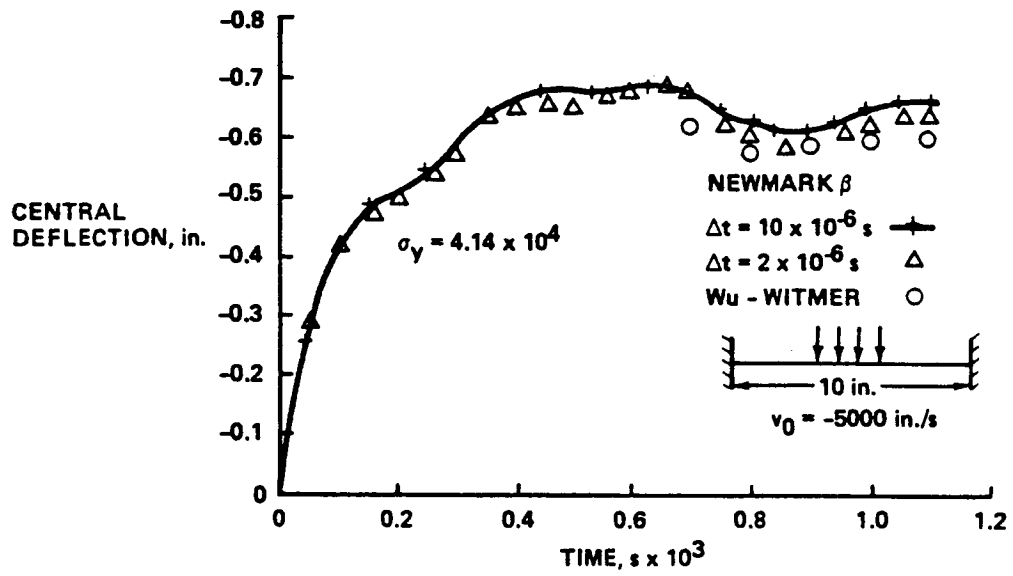
6.26

Fig. 6.4.3 Central Deflection Versus Time for Example Problem No. 4

6.5   EXAMPLE PROBLEM NO. 5 - IMPULSIVELY LOADED ELASTO-PLASTIC RECTANGULAR PLATE

The problem is defined in Fig. 6.5.1 and involves a rectangular plate subjected to an impulsively applied lateral velocity. The listing of the input is in Fig. 6.5.2. Thirty plate elements (TRP2) are used in one quarter of the plate. A constant step central difference integrator is used with a time step of 0.25×10^{-6} s. A material with linear strain hardening is used with $E_T/E = 0.005$ and a yield stress of 233.0 N/mm^2. Eight integration layers were prescribed through the plate thickness. Results are shown in Fig. 6.5.3 and compare favorably with results given in "Transient Dynamic Large Deflection Analysis of Elastic Viscoplastic Plates by the Finite Element Method," Int. J. Mech. Sci., Vol 22, pp 151-166 (1980).
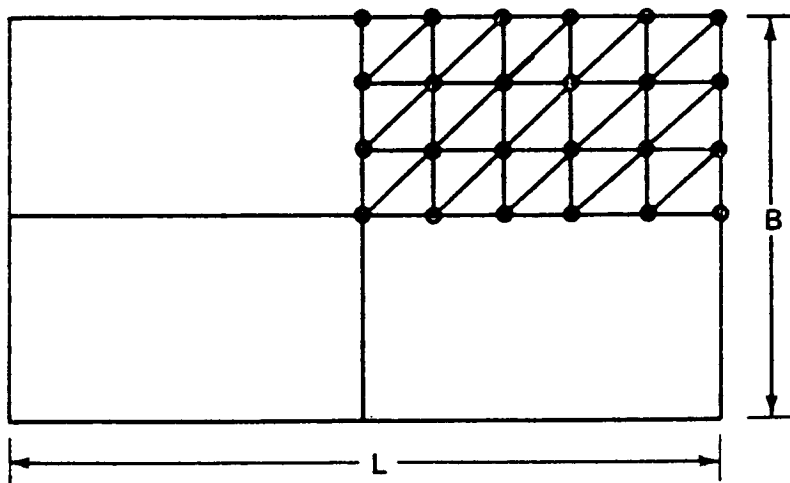
L = 128.59 mm

B = 76.2 mm

THICKNESS = 2.499 mm

$\rho = 7.72 \times 10^{-9}$ N s^2/mm^4

YIELD STRESS = 2.33×10^2 N/mm^2

$E = 2.07 \times 10^5$ N/mm^2

$\nu = 0.3$

Fig. 6.5.1   Impulsively Loaded Elasto-Plastic Rectangular Plate

PROGRAM LISTING OF INPUT DATA CARDS

```
CARD      .........1.........2.........3.........4.........5.........6.........7.........8
COUNT     12345678901234567890123456789012345678901234567890123456789012345678901234567890

  1-     DYCAST EXAMPLE PROBLEM    IMPULSIVELY LOADED  CLAMPED PLATE
  2-     DYNA  0000   80 1000
  3-     DYNA                 0.400      E-03 0.250      E-06
  4-     CDIF
  5-     TIME
  6-     MPRIN
  7-         1    2
  8-     NPRIN
  9-         1   -24
 10-     SEND
 11-         1   -24
 12-     TRP2         1    1    5    6
 13-     TRP2         2    6    2    1
 14-     TRP2         3    2    6    7
 15-     TRP2         4    7    3    2
 16-     TRP2         5    3    7    8
 17-     TRP2         6    8    4    3
 18-     TRP2         7    5    9   10
 19-     TRP2         8   10    6    5
 20-     TRP2         9    6   10   11
 21-     TRP2        10   11    7    6
 22-     TRP2        11    7   11   12
 23-     TRP2        12   12    8    7
 24-     TRP2        13    9   13   14
 25-     TRP2        14   14   10    9
 26-     TRP2        15   10   14   15
 27-     TRP2        16   15   11   10
 28-     TRP2        17   11   15   16
 29-     TRP2        18   16   12   11
 30-     TRP2        19   13   17   18
 31-     TRP2        20   18   14   13
 32-     TRP2        21   14   18   19
 33-     TRP2        22   19   15   14
 34-     TRP2        23   15   19   20
 35-     TRP2        24   20   16   15
 36-     TRP2        25   17   21   22
 37-     TRP2        26   22   18   17
 38-     TRP2        27   18   22   23
 39-     TRP2        28   23   19   18
 40-     TRP2        29   19   23   24
 41-     TRP2        30   24   20   19
 42-     SEND
 43-     GRDX  0.0       E+00    1   -4
 44-     GRDX  1.2859    E+01    5   -8
 45-     GRDX  2.5718    E+01    9   -12
 46-     GRDX  3.8577    E+01   13   -16

          .........1.........2.........3.........4.........5.........6.........7.........8
          12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

Fig. 6.5.2 Example Problem No. 5 - Input Listing (Sheet 1 of 2)

6.29

PROGRAM LISTING OF INPUT DATA CARDS

```
CARD    .........1.........2.........3.........4.........5.........6.........7.........8
COUNT   12345678901234567890123456789012345678901234567890123456789012345678901234567890

47-     GRDX  5.1436    E+01   17  -20
48-     GRDX  6.4295    E+01   21  -24
49-     GRDY  0.0       E+00    1   -4  -21
50-     GRDY  1.27      E+01    2   -4  -22
51-     GRDY  2.54      E+01    3   -4  -23
52-     GRDY  3.81      E+01    4   -4  -24
53-     GRDZ  0.0       E+00    1  -24
54-     SEND
55-     SPC   001000            1
56-     SPC   011100            2    3
57-     SPC   111110            6   -4  -18
58-     SPC   111110            7   -4  -19
59-     SPC   000000            4   -4  -20
60-     SPC   101010            5   -4  -17
61-     SPC   000000           21  -24
62-     SEND
63-     IVEL  0.0       E+00 0.0     E+00 7.045    E+04
64-     IVEL  0.0       E+00 0.0     E+00 0.0      E+00
          1  -24
66-     SEND
67-     MATL  2.07      E+05  .3
68-     7.33      E-09
69-     0.005          0.0         2.33     E+02
70-        1  -30
71-     THIN  2.499          8.0
72-        1  -30
73-     SEND
74-     SEND
75-
76-
77-     STOP
```

```
.........1.........2.........3.........4.........5.........6.........7.........8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

Fig. 6.5.2 Example Problem No. 5 - Input Listing (Sheet 2 of 2)

6.30

$v_0 = 7.045 \times 10^4$ mm/s

□ TRP2 CENTRAL DIFFERENCE INTEGRATOR

$\Delta t = 0.5 \times 10^{-6}$ s

w_c (mm)

REFERENCE F.E. SOLUTION
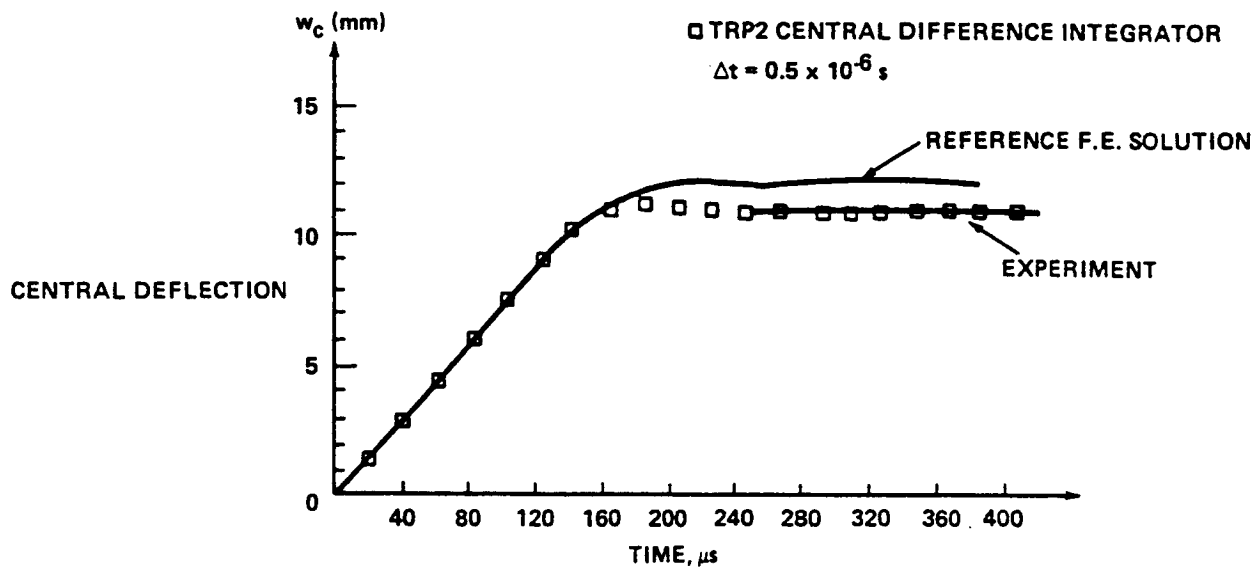
CENTRAL DEFLECTION

EXPERIMENT

TIME, μs

Fig. 6.5.3 .Central Deflection Versus Time for Example Problem No. 5

Standard Bibliographic Page

1. Report No. NASA CR-4040	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle DYCAST — A Finite Element Program for the Crash Analysis of Structures		5. Report Date January 1987
		6. Performing Organization Code
7. Author(s) A. B. Pifko, R. Winter, and P. Ogilvie		8. Performing Organization Report No.
9. Performing Organization Name and Address Grumman Aerospace Corporation Bethpage, NY 11714		10. Work Unit No.
		11. Contract or Grant No. NAS1-13148
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546		13. Type of Report and Period Covered Contractor Report
		14. Sponsoring Agency Code 505-63-41-01

15. Supplementary Notes

Langley Technical Monitor: Martha P. Robinson
Final Report

16. Abstract

DYCAST is a nonlinear structural dynamic finite element computer code developed for crash simulation. The element library contains stringers, beams, membrane skin triangles, plate bending triangles and spring elements. Changing stiffnesses in the structure are accounted for by plasticity and very large deflections. Material nonlinearities are accommodated by one of three options: elastic-perfectly plastic, elastic-linear hardening plastic, or elastic-nonlinear hardening plastic of the Ramberg-Osgood type. Geometric nonlinearities are handled in an updated Lagrangian formulation by reforming the structure into its deformed shape after small time increments while accumulating deformations, strains, and forces. The nonlinearities due to combined loadings are maintained, and stiffness variation due to structural failures are computed. Numerical time integrators available are fixed-step central difference, modified Adams, Newmark-beta, and Wilson-theta. The last three have a variable time step capability, which is controlled internally by a solution convergence error measure. Other features include: multiple time-load history tables to subject the structure to time dependent loading; gravity loading; initial pitch, roll, yaw, and translation of the structural model with respect to the global system; a bandwidth optimizer as a pre-processor; and deformed plots and graphics as post-processors.

17. Key Words (Suggested by Authors(s)) Crash Analysis Dynamic Analysis Nonlinear Analysis Large Displacement Analysis Finite Element Analysis	18. Distribution Statement Unclassified-Unlimited Subject Category 64

19. Security Classif.(of this report) Unclassified	20. Security Classif.(of this page) Unclassified	21. No. of Pages 348	22. Price A15