SUBSTRUCTURING ANALYSIS FOR IMPACT DYNAMICS USING ANSYS FINITE ELEMENT PACKAGE

by

Chang Ho Cho

Thesis Submitted to the Faculty of the Graduate School of
the New Jersey Institute of Technology in Partial fulfillment of
the Requirements for the Degree of
Master of Science in Mechanical Engineerig
1986

APPROVAL SHEET

Title of Thesis: Substructuring Analysis for Impact Dynamics using ANSYS Finite Element Package

Name of Candidate: Chang Ho Cho
Master of Science, 1986

Thesis and Abstract Approval:

Dr. Sachio Nakamura — DATE
Assistant Professor
Department of Mechanical Engineerig

Signatures of other members of the thesis committee.

_____ DATE

_____ DATE

VITA

Name: Chang Ho Cho

Degree and Data to be Conferred: Master of Science
Mechanical Engineering
May, 1986

Secondary Education: Jinju High School, 1967-1970

Collegiate Institutions Attended:

	Dates	Degree	Data of Degree
Korea University	1970 to 1975	BSME	February 1975

Major: Mechanical Engineering

ABSTRACT

Title of Thesis: Substructuring Analysis for Impact Dynamics using ANSYS Finite Element Package

Chang Ho Cho, Master of Science, 1986

Thesis Directed by : Dr. Sachio Nakamura
Assistant Professor
Mechanical Engineering Department

This thesis is intended to serve as an introductory guide to ANSYS finite element package for the analysis of impact dynamics. Substructuring technique has been used to reduce the degrees of freedom.

Finite element method is essentially a numerical technique to calculate elastic deformations. However, by introducing some techniques like gap element, the technique can be easily extended to rigid body dynamic analysis. To solve the equation of motion both for rigid boby and elasitc deformations, direct integration technique is used in this thesis, i.e., Houbolt method.

ANSYS is a large scale general purpose computer program for finite element analysis. Because of its generality and capability of analysis, the package is becoming one of the most popular and poweful tools in structural analysis. However, since the package is so big that it takes a while for the beginner to fully understand its operation.

This thesis is hoped to provide basic concepts and numerical theories for the beginners to use this package in

a consistant manner, especially in the case of impact dynamics, since ANSYS manual fails to give a clear explanation on the topic.

_____ _____

Signature of Adviser Date

TABLE OF CONTENTS

LISTS OF FIGURES

CHAPTER I. INTRODUCTION

A. Introductory Comments

Impact dynamics is of growing interest in morden technology. To guarantee the safety of passengers against collision, striking and all kinds of impact, the crash impact condition has been added to the traditional set of structural design criteria either by contract or by safety law. According to the U.S. Fedral Motor Vehicle Safety Standards, for example, the passengers must be safe in frontal barrier crashed with 30 mph[1].

Although the safety must be confirmed by physical experiments, numerical simulations such as finite element method are widely used in the design phase of product development to reduce development cost. Many kinds of general purpose as well as special purpose finite element codes are available on the market today, e.g., ADINA, ANSYS, NASTRAN, MARC, KRASH, DYCAST, WRECKER, etc.[1,2]

Two methods are commonly used to solve impact dynamics. One approach is the gap element technique and the other is Hughes type approach. Both methods have their own advantages as well as disadvantages, respectively. Thus all sorts of reseaches are still being carried on[3-8].

When two elastic bodies come into collision with each other, the contact stresses are usually beyond the elastic limit in the local contact surface. This high stress tran-

smits to the whole regions with time. The large displacements and rotations are also developed in the local area. Therefore the problem can be divided into the wave propagation problem and the structural vibration problem involving geometrical and material nonlinearities[9]. Thus the contact forces and kinematic behaviours need to be studied in the time domain.

The finite element method for nonlinear dynamic analysis requires a large amount of computational works compared with linear stastic analysis. Because of the nature of the step-by-step time integration solution scheme, the computation is a time consuming process. However, in many practical problems, usually a small portion of the solutions is of interest. The nonlinearity is also limited within local area. Thus either the substructuring technique or the mode superposition technique is widely used to reduce the problem size as well as the computation time. Substantial amount of researches have been done for the both methods [4,9-11].

The computer program ANSYS which is used in this thesis is a general purpose finite element code developed and distributed by Swanson Analysis System, Inc. Unfortunately, this program has limited cabpabilities of handling impact dynamics and therefore user must be careful in using ANSYS for this purpose.

B. Literature Survey

Many general purpose finite element codes have the capabilities of handling gap elements and contact loadings[2]. In the following, ANSR-II which is an extended version of ANSR-I is selected to be compared with ANSYS. ANSR (Analysis of Nonlinear Structural Response) is developed and maintained by Earthquake Engineering Research Center at University of California, Berkeley. It has a gap-friction element to handle gap conditions. This element is developed on the basic assumptions of trilinear inelastic forcedeformation relationship, constant friction coefficient and zero internal viscous damping. Although the gap-friction element of ANSR is quite similiar to the 3-D interface element of ANSYS, the ANSYS allows linear force-deformation relationship only.

The gap-friction element of ANSR is defined by one bearing plane and one node as shown in Fig. 1-1. The bearing plane may be defined by three nodes and may be arbitrarily oriented in space[12]. The 3-D interface element of ANSYS is defined by two nodes as shown in Fig. 1-2. The interface plane is assumed to be perpendicular to the connecting line of two nodes[13]. Both elements have the capability of resisting deformation normal and shear to a bearing and a interface plane[12,13]. ANSYS has additionally few more different elements and one independent module having the capability to handle gap condition. This will be further discussed in Chapter II. ANSR has limited element libra-

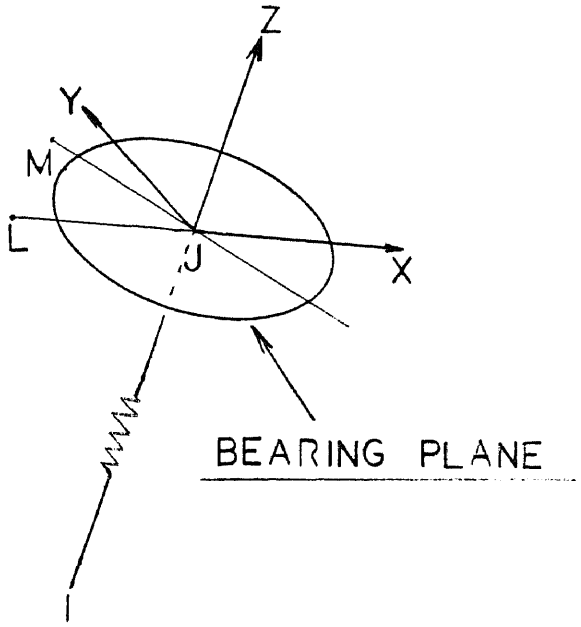

Fig. 1-1: Gap-Friction Element

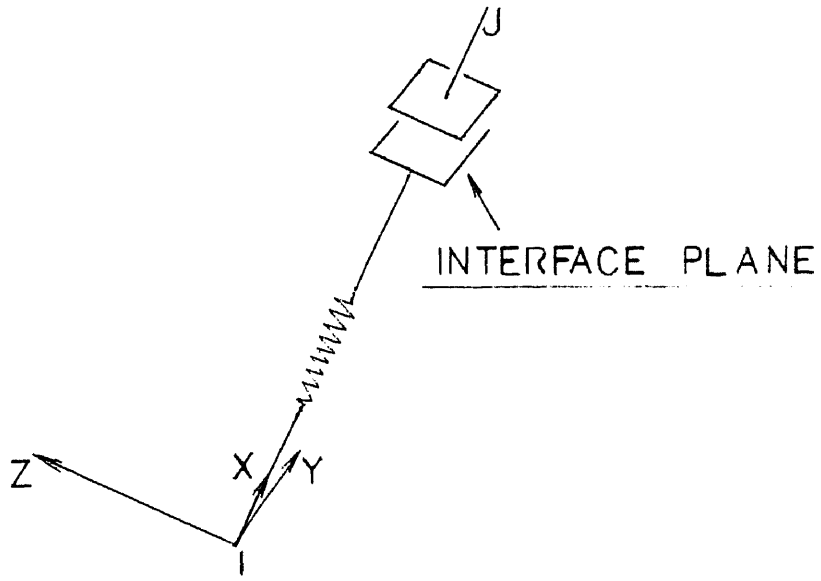

Fig. 1-2: 3-D Interface Element

ries and more element libraries are under development[2]. Thus the current version of ANSR-II has pretty limited capability in handling impact dynamics.

C. Scope of the Thesis

This thesis is intended as an introdutory guide to ANSYS package for the analysis of impact dynamics. The main focus is to predict the contact forces as well as the kinematic behaviour of elastic bodies during the impact. The damping effect and the material nonlinearity are ignored. The large displacement is allowed and the stress stiffening effect is included. The substructuring technique is used to reduce the computer memory requirement and computation time.

Many works have been done to solve the impact dynamics using gap elements[1,3,7]. However, many of those works involve only a single gap element. This thesis attempts to use two gap elements in series to simulate practical situations using ANSYS to study the numerical behaviors and to search for better numerical technique. In the following, the gap condition is simulated using a combination element, JSTIF=40, or a gap condition module in ANSYS.

Since the capability of ANSYS is pretty limited, e.g., it has no capability to check the penetration of a body, the selected examples are all quite simple. The first example simulates rigid body dynamics. The second example calculates stresses in the time domain to simulate collision.

The third example figures out the kinematic behaviours of a impact of two gap elements in series to simulate helmet impact dynamics.

CHAPTER II. COMPUTER PROGRAM ANSYS

A. Introductory Comments

The ANSYS is a proprietary general purpose computer program developed and maintained exclusively by Swanson Analysis System, Inc. The first edition was released in 1972, and it is continuously being updated with the addition of improved modeling and problem solving techniques. The present edition installed at Mechanical Engineering Department of NJIT is Version 4.2 released in 1985.

The ANSYS program is a large scale finite element code using the matrix displacement method to solve the several classes of engineering problems. Analysis capabilities cover (1) structural mechanics including static and dynamic, linear and nonlinear, (2) heat transfer including steady-state and transient, (3) fluid flow and (4) electro- and magnetostatics. The overall flow chart of ANSYS analysis type is given in Fig. 2-1. It also has the capability to create a superelement, substructuring, as shown in Fig. 2-2. The element library contains more than forty elements for structural analysis and more than twenty elements for heat transfer asnalysis.

The solution techniques used in ANSYS are: (1) wavefront (or frontal) method to solve a system of linear simultaneous equations, (2) Guyan reduction technique for dynamic matrix condensation, (3) Jacobian method for the solution of

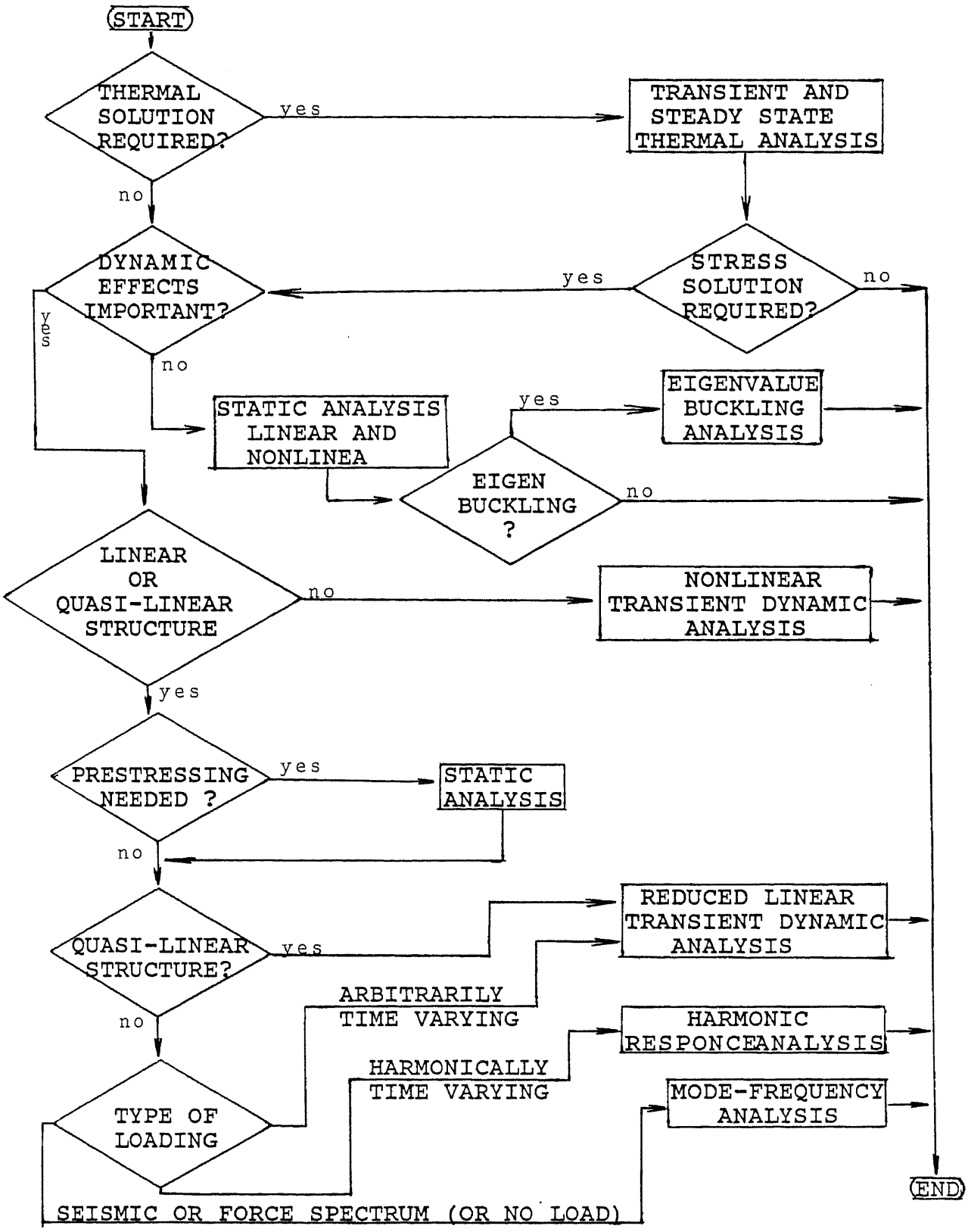

Fig. 2-1: Flow Chart of ANSYS Analysis

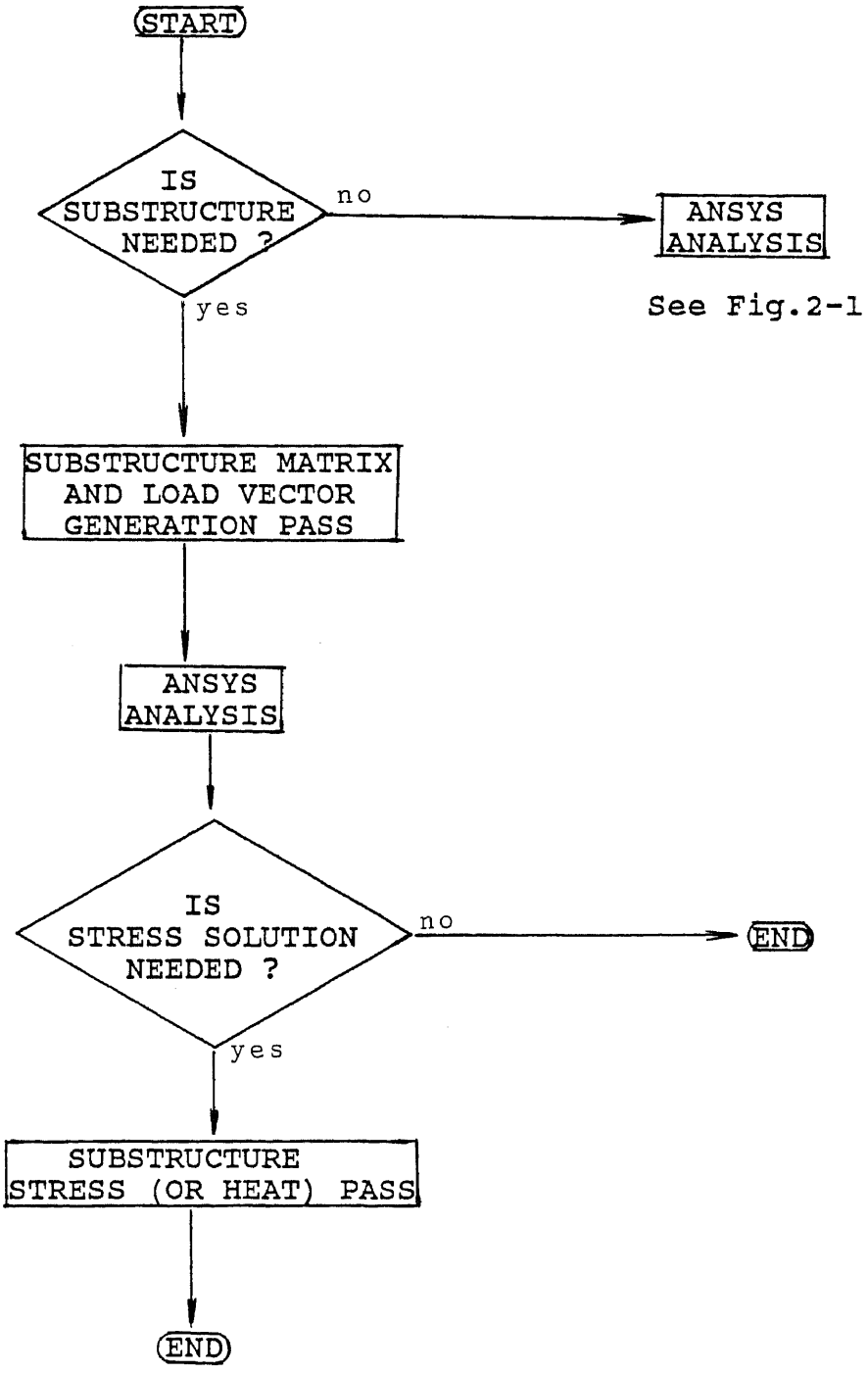

Fig. 2-2: Flow Chart of Substructuring

eigen value problems (EVP) and (4) an implicit numerical integration method (i.e., Houbolt method) for initial value problems (IVP)[2,13,14].

ANSYS may be run in a fully interactive mode, in a fully batch mode or in a combination mode of both. The program is user oriented and self-contained. It does not require special knowledge of system operations or computer programming. However, because of the flexibility and generality of its analysis capability, user must know the basic assumptions as well as the theories based on for correct use.

B. Structure of ANSYS

The computer program ANSYS consists of several routines, such as preprocessing routines, postprocessing routines, auxiliary routines, etc. Each of the routines is a group of related modules, and a module is also a group of related commands. The commands are divided into two types, slash(/) type and ordinary type. The slash commands are usually used for supplying general control instructions such as selecting the run mode and the type of run, printout controls, plot controls, file controls, etc. The ordinary commands are used for supplying general specification and action instructions such as specifying all input data, generating mesh and plot, etc[13].

Analysis of any engineering problem basically consists of

three phases: preprocessing phase, solulution phase and postprocessing phase. The preprocessing is performed using PREP7 and PREP6 routines. In this phase all input data and solution schemes may be defined. The PREP7 routine is used to define the all general input data. The PREP6 routine is used to define the transient load data. The solution phase is designed to be automatically intiated, so that the basic user needs only two or three commands to activate this phase(/INPUT,27, /INPUT,23 and FINISH). In this phase the validity of input data can be checked and the solutions of analysis may be obtained.

The postprocessing phase is optional. In this phase the solutions of analysis may be reorganized, tabulated and plotted. The ANSYS has several postprocessing routines, such as POST1, 26, 27, 28, 29 and 30. The functions of each routines may be summarized as follows[13].

ROUTINES	FUNCTIONS
POST1	General Database Results Postprocessing
POST26	Time-History Results Postprocessing
POST27	Post Data File Operations and Results Postprocessing
POST28	Response Spectrum Generation
POST29	2-D Solid Harmornic Element Postprocessing
POST30	2-D Shell Harmornic Element Postprocesasing

The ANSYS run may be completed with three routines in most engineering analyses. But linear transient dynamic

analysis, KAN=5, and harmonic response analysis, KAN=6, consist of four routines, because these two analyses have two solution passes. In these two analyses, a stress pass is additionally required to obtain the whole solutions.

Many local files are created during ANSYS run. Some of these files are created according to user's commands, and all other files are automatically created during run. All of those files contain their own data needed for the solution of analysis and may also be needed for related phases, such as a solution restart, a solution stress pass, postprocessing, etc. The data transfer from one routine to another routine may be achieved by means of these local files[13].

C. Solution Scheme

ANSYS is developed using finite element method employing matrix displacement technology based on the energy principle. According to the principle of virtual work,

$$\delta U = \delta V \tag{2-1}$$

Here δU is the first variation of total strain energy given by

$$\delta U = \int_V \delta \varepsilon^T \sigma \, dV \tag{2-2}$$

and δV is the first variation of total potential energy given by

$$\delta V = -\delta u F + \int_V \delta u^T b \, dV + \int_A \delta u^T s \, dA \tag{2-3}$$

where u, F, b and s are nodal displacement, external force, body force and surface traction vectors, repectively.

Substituting equations (2-2) and (2-3) into equation (2-1), one obtains the following equiliblium equations[15].

$$K\,u + f = F \tag{2-4}$$

where K is an element stiffness matrix given by

$$K = \int_V B^T D\, B\, dV \tag{2-5}$$

Here B and D are a strain shape function and a stress matrix, respectively. f is a nodal element force vector given by

$$f=-\int_V N^T b\, dV-\int_A N^T s\, dA-\int_A B^T D\, \varepsilon_o\, dV+\int_V B^T \sigma_o\, dV \tag{2-6}$$

Here N is a displacement shape funtion. Equation (2-4) can be rewritten in partitioned matrix form:

$$\begin{bmatrix} K_{ii} & K_{ib} \\ K_{bi} & K_{bb} \end{bmatrix} \begin{Bmatrix} u_i \\ u_b \end{Bmatrix} = \begin{Bmatrix} F_i \\ F_b \end{Bmatrix} - \begin{Bmatrix} f_i \\ f_b \end{Bmatrix} \tag{2-7}$$

The subscripts i and b are used to refer the degrees of freedom that are free and imposed displacements, respectively. The first part of equation (2-7) may be solved for basic unknouns $\{u_i\}$.

$$\{u_i\} = -[K_{ii}]^{-1}\,[K_{ib}]\{u_b\} + [K_{ii}]^{-1}\,\{R_i\} \tag{2-8}$$

where R=F-f.

The main portion of this solution procedure is to calculate the inverse of the stiffness matrix K. The Gaussian elimination procedure is one of the most popular technique for this purpose, and the total stiffness matrix is usually assembled before performing this procedure. In this case back-up storage for the total unreduced stiffness matrix is required. From a view point of storage manipulation, this technique is not efficient. ANSYS uses more memory efficient technique, wave front direct solution method. This method deals with the only related stiffness matrices that are actually reqired to eliminate a specific degree of freedom and then it statically condenses out that degree of freedom[14,16].

The active equations are represented by

$$\sum_{j=1}^{n} K_{lj}\, u_j = R_l \qquad (2\text{-}9)$$

Here n is a total number of equations and l is an equation number, i.e., row number. To eliminate a typical equation p=l, the equation is first normalized to[14]:

$$\sum_{j=1}^{n} \frac{K_{pj}}{K_{pp}}\, u_j = \frac{R_p}{K_{pp}} \qquad (2\text{-}10)$$

This equation may be rewritten as follows and stored in a file for later back-substitution.

$$\sum_{j=1}^{n} K^*_{pj}\, u_j = R^*_p \qquad (2\text{-}11)$$

where

$$K^*_{pj} = K_{pj}/K_{pp} \tag{2-12}$$

$$R^*_p = R_p/K_{pp} \tag{2-13}$$

The remaining equations are modified and the p's equation, i.e., p's row, is eliminated from the following equations.

$$K^*_{lj} = K_{lj} - K_{lp}\, K^*_{pj} \tag{2-14}$$

$$R^*_l = R_l - K_{lp}\, R^*_p \tag{2-15}$$

$$\sum_{j=1}^{n-1} K^*_{lj}\, u_j = R^*_l \tag{2-16}$$

where $l \neq p$.

This process is repeated for all other equations to be eliminated. Since this procedure is performed in the order of elements, the size of wave front is directly affected by the numbering scheme of elements. Thus an effective numbering of elements is necessary to reduce the size of memory storage. The other solution scheme adopted in ANSYS, i.e., Houbolt method, is considered in Chapter III.

CHAPTER III. IMPACT DYNAMICS

A. General Description

Impact dynamics may be defined as a dynamic response problem in which the applied load F is developed by impact. Since the boundry conditions are continuously changing in time, it must be treated as a nonlinear problem regardless of the nonlinearities of geometry and materials[15]. In order to allow the rigid body motion in finite element analysis, the concept of gap element is introduced. Since this problem is to be solved using ANSYS, the problem is viewed from the point of ANSYS. ANSYS has two possibilities to analyze this problem. For the whole time domain, this problem must be solved by nonlinear transient dynamic analysis, KAN=4, with gap element. However, for a certain time domain, this problem may be solved by reduced linear transient dynamic analysis, KAN=5, with gap conditions.

A general dynamic equilibrium equation may be written by

$$M\ddot{u} + C\dot{u} + Ku = F \qquad (3\text{-}1)$$

This equation can be solved using numerical integration scheme based on the finite difference method. Among several numerical integration schemes for solving this equation, ANSYS adopts the Houbolt method.

The Houbolt method uses two backward difference formula to express the velocity and acceleration vectors in terms of

displacement vector[14,16]:

$$u_{t+\Delta t} = \frac{1}{6\Delta t}\{11u_{t+\Delta t} - 18u_t + 9u_{t-\Delta t} - 2u_{t-2\Delta t}\} \qquad (3\text{-}2)$$

$$u_{t+\Delta t} = \frac{1}{(\Delta t)^2}\{2u_{t+\Delta t} - 5u_t + 4u_{t-\Delta t} - u_{t-2\Delta t}\} \qquad (3\text{-}3)$$

The solutions of the governing equation at time $t + \Delta t$ are given in terms of u_t, $u_{t-\Delta t}$ and $u_{t-2\Delta t}$ by

$$\left(\frac{2}{(\Delta t)^2}M + \frac{11}{6\ t}C + K\right)u_{t+\Delta t} = F_{t+\Delta t} + \left(\frac{5}{(\Delta t)^2}M + \frac{3}{t}C\right)u_t$$

$$- \left(\frac{4}{(\Delta t)^2}M + \frac{3}{2\ t}C\right)u_{t-\Delta t} + \left(\frac{1}{(\Delta t)^2}M + \frac{1}{3\ t}C\right)u_{t-2\Delta t} \qquad (3\text{-}4)$$

The accuracy of this method dependens on the time step Δt, and the computation time is also directly propotional to the number of time steps used. Therefore, finding an optimum time step becomes a crucial factor in both the accuracy and computation cost.

B. Gap Element

The ANSYS has four kinds of elmements handling gap conditions: (1) compression or tension only spar element, JSTIF= 10, (2) 2-D interface element, JSTIF=12, (3) 3-D interface element, JSTIF=52, and (4) combination element, JSTIF=40. Although two interface elements can withstand the normal and shear forces, only normal forces are considered in this

thesis. However the compression or tension only spar element is not sufficient in funtion. Thus the combination element is used in this thesis.

This element consists of (1) two springs, (2) one damper, (3) one friction resistor and (4) one gap as shown in Fig. 3-1. According to the specification of nodes in a given coordinate system, it may act both as a hook element and a gap element. In a gap element a positive displacement is defined such that a node j moves away from nodei. In a hook element a positive displacement is defined such that a node i moves toward node j. In both elements the positive displacement is defined such a way of tending to open the gap. Thus the two elements act reversely each other. This point is illustrated in Fig. 3-2.

The force-deflection relation for this element is as shown in Fig. 3-3. The element force matrix for this element is

$$\{F_e\} = F \begin{Bmatrix} -1 \\ 1 \end{Bmatrix} \tag{3-5}$$

where

$$F = F_1 + F_2$$

and

$F_1 = F_{slide}$ if slider was sliding in previous iteration.

$F_1 = (u_{gap} - u_{slide}) * K_1$ if slider was not sliding in previous iteration.

$F_2 = u_{gap} * K_2$

K_i = stiness of spring i

The element mass matrix is defined by user, depending upon the value of KEYOPT(6):

$$[M_e] = \begin{cases} M \begin{bmatrix} 1 & 0 \\ 0 & 0 \end{bmatrix} & \text{if KEYOPT(6)=0} \\ \frac{M}{2} \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} & \text{if KEYOPT(6)=1} \\ M \begin{bmatrix} 0 & 0 \\ 0 & 1 \end{bmatrix} & \text{if KEYOPT(6)=2} \end{cases} \qquad (3\text{-}6)$$

where M is a total element mass.

The element stiffness matrix is

$$[K_e] = K \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \qquad (3\text{-}7)$$

where

$K = S_1 + S_2$

and

$S_1 = K_1$ or zero, if slider was not sliding or sliding in previousiteration, respectively

$S_2 = K_2$

In the following, the damping matrix C and friction force F_{slide} are set to zero. This means that the friction and damping effects are neglected. This could be rationalized by the fact that at the instant of impact, which is usually very short time period, the viscous damping has no signifi-

cant effect.

C. Time Step Optimization

As stated before, the selection of an appropriate time step is crucial to solve a dynamic equilibrium equation using direct integration methods. For selecting an appropriate time step, the stability of the solutions must be considered as well. The analysis of those characteristics results into guidelines for the selection of an appropriate time step.

One solution procedure is to decouple the equations into the bases of eigenvectors, instead of dealing with the coupled general dynamic equlibrium equations, to analyze the stability and the acuracy of the solution. By doing so, one may consider only a typical equation of one degree of freedom.

$$\ddot{x} + 2\xi\omega\dot{x} + \omega^2 x = r \qquad (3\text{-}8)$$

The variables to be considered for this analysis are only time step Δt, eigenvalues for free vibration ω and damping ratio ξ. In the following, Houbolt method used in ANSYS is discussed in detail.

The Houbolt method belongs to direct and implicit integration method. The equilibrium equations are considered at time instant $t+\Delta t$ instead of at time t for solutions at time $t+\Delta t$. Substituting the equations (3-2) and (3-3)

after changing variables into equation (3-8), one can get the following relationships among x's in different time steps[16].

$$\begin{Bmatrix} x_{t+\Delta t} \\ x_t \\ x_{t-\Delta t} \end{Bmatrix} = A \begin{Bmatrix} x_t \\ x_{t-\Delta t} \\ x_{t-2\Delta t} \end{Bmatrix} + H\, r_{t+\Delta t} \qquad (3\text{-}9)$$

where

$$A = \begin{bmatrix} \frac{5\beta}{\omega^2(\Delta t)^2} + 6\gamma & -\frac{4\beta}{\omega^2(\Delta t)^2} + 3\gamma & \frac{\beta}{\omega^2(\Delta t)^2} + \frac{2\gamma}{3} \\ 1 & 0 & 0 \\ 0 & 1 & 0 \end{bmatrix}$$

$$\beta = \left(\frac{2}{\omega^2(\Delta t)^2} + \frac{11}{3\omega\Delta t} + 1\right)^{-1}, \qquad \gamma = \frac{3\beta}{\omega\Delta t}$$

and

$$H = \begin{Bmatrix} \frac{\beta}{\omega^2} \\ 0 \\ 0 \end{Bmatrix}$$

Since the stability problem must be considered for any arbitrary initial conditions, one may consider free vibration mode, i.e.,r=0. In this case the equation (3-9) can be simplified as follows.

$$x_{x+n\Delta t} = A^n x_t \qquad (3\text{-}10)$$

This is an eigenvalue problem, and the criterion of stabili-

ty is given by eigenvalues. The spectral radius of matrix A is defined by the eigenvalses λ_i of A.

$$\rho(A)=\text{Max.}|\lambda_i|, \quad i = 1,2,.....,n \tag{3-11}$$

Then the stability criterion is given by[16]

$$\rho(A) \leq 1 \quad \text{for} \quad n \longrightarrow \infty \tag{3-12}$$

Generally the small damping is insignificant upon the overall stability characteristics. Therefore the stability of the Houbolt method as well as many of the other integration schemes actually depends on the ratio of time step to period T, i.e., $\Delta t/T$. As shown in Fig. 3-4, the spectral radius $\rho(A)$ is always less than 1.0 regardless of the values of $\Delta t/T$. This proves the fact that Houbolt method is unconditionally stable. That means there is no restriction on the time step as long as the stability is concerned.

For the consideration of accuracy, the criterion of $\Delta t/T$ must be drawn by analyzing the period elongation and the amplitude decay of the solutions in the time domain. Bathe claimes that the solutions of numerical integration using the Houbolt method is generally accurate when $\Delta t/T$ is smaller than 0.01[16].

In practical problems, the time step must be chosen from the view point of accuracy and cost. ANSYS recommends the use of time step of one tenth to one fortieth of period T, according to the contact stiffness, for transient dynamic analysis using gap element.

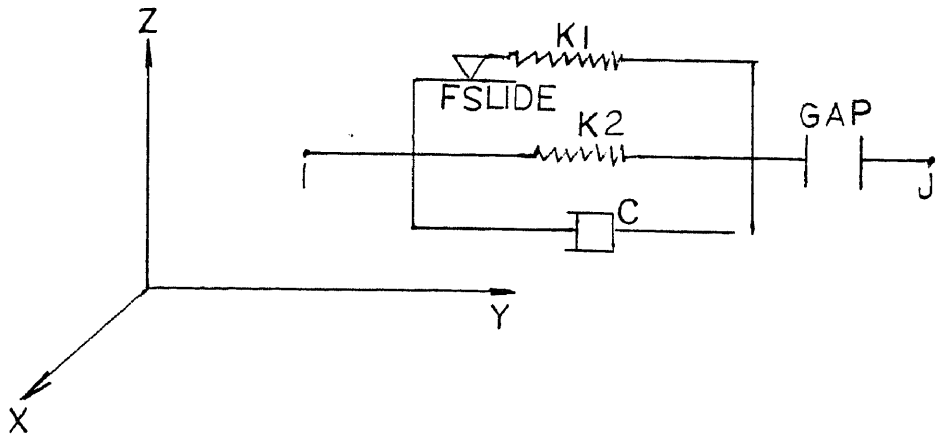

Fig. 3-1: Combination Element

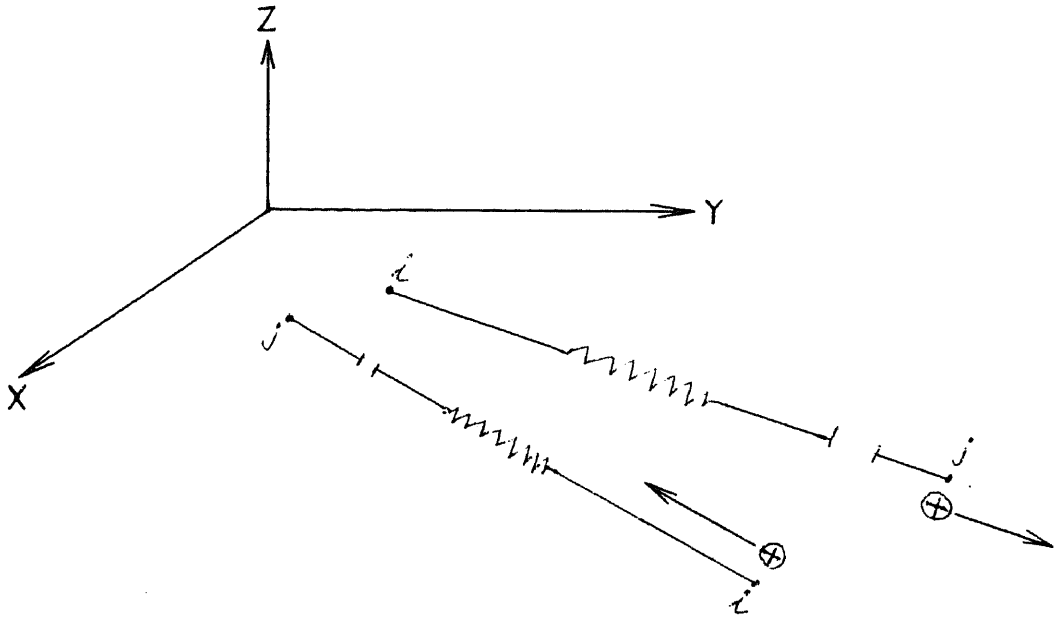

Fig. 3-2: Direction of Positive Displacement

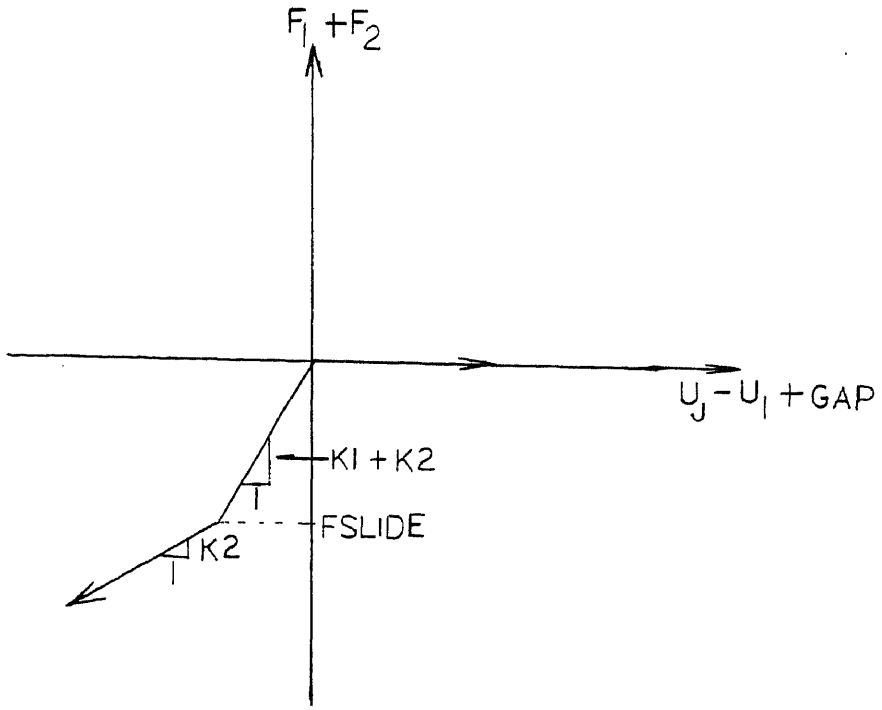

Fig. 3-3: Force-Deflection Relation

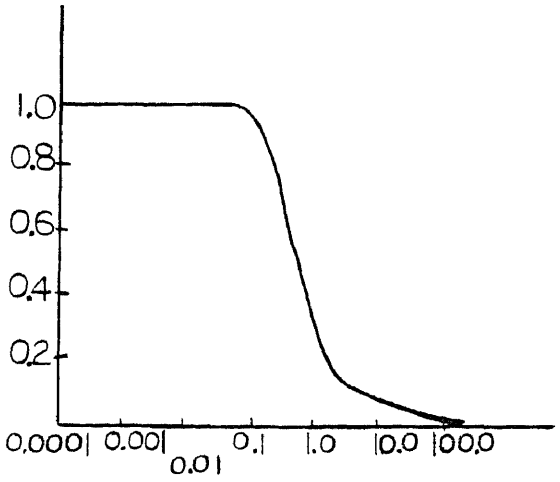

Fig. 3-4: Spectral Radius for ξ =0.0

CHAPTER IV. SUBSTRUCTURING TECHNIQUE

A. Introductory Comments

The matrix method of structural analysis as well as more general finite element method leads to a set of large number of linear algebraic equations in matrix form. Fortunately, those matrices are sparce, symmetric and positive definite. Many efficient numerical methods have been developed, taking advantage of these characteristics. To increase the efficiency further, substructuring technique was developed in 1960s [16].

Even with the advanced hardware and software technique in computer technology today, the substructuring technique is still an attractive tool in computational and logical aspects of structural analysis. This technique can be also applied to the structures having geometric symmetry and repeated systems to reduce the modeling time. In the case of nonlinear dynamic analysis, the stiffness matrix K can be divided into linear and nonlinear parts. Substructuring technique can be applied to the linear part in the same way as in static linear analysis.

This technique has three major advantages[13,17]. The first advantage is that the computation time can be reduced, because a large amount of degrees of freedom are condensed out and the non-interesting part of solutions can be skipped. The second one is that the large structures can be

analyzed with relatively small memory size, because the total degrees of freedom dealt with in substructure analysis are much less than those of entire structure. The third one is that the modeling of large scale structures become easier, since the seperate modeling scheme can be adopted and one substructure can be used repeatedly in repeated systems or symmetric parts.

B. General Theory

The basic idea of substructuring tehenique is the following[17]:

(1) The whole structure is divided into several substructures.

(2) The displacements of common boundries with adjoint substructures are assumed to be completely fixed.

(3) Each substructure created is analyzed seperately.

(4) All common boundries are relaxed simultaneously.

(5) The actual displacements of commonboundriesare determined from equlibrium equations of forces at boundries.

(6) Each substructure can then be treated as one independent structure from others and thus it can be analyzed seperately under eachreal substructure loadings and boundary displacements.

To review those concepts, one considers the following

equilibrium equations given by matrix form:

$$[K]\ \{u\} = \{F\} \tag{4-1}$$

Applying this equation to one typical substructure, one can rewrite the equation in partitioned form:

$$\begin{bmatrix} K_{bb} & K_{bi} \\ K_{ib} & K_{ii} \end{bmatrix} \begin{Bmatrix} u_b \\ u_i \end{Bmatrix} = \begin{Bmatrix} F_b \\ F_i \end{Bmatrix} \tag{4-2}$$

The subscripts b and i refer to the boundary and internal nodes, respectively.

Solving equation (4-2) with $u_b = 0$, the first part of solutions are:

$$\bar{u}_i = K_{ii}^{-1}\ F_i \tag{4-3}$$

$$\bar{F}_b = K_{bi}\ \bar{u}_i \tag{4-4}$$

Here $\bar{F}_b$ are the constrain forces at boundries necessary to maintain $u_b=0$ when the external forces F_i are applied to the internal nodes.

When the boundries are relaxed, the correction terms can be determined by setting $F_i=0$ and $F_b = F_b - \bar{F}_b$ in equation (4-2).

$$u_i' = -K_{ii}^{-1}\ K_{ib}\ u_b' \tag{4-5}$$

$$u_b' = K_b^{-1}\ F_b' \tag{4-6}$$

where

$$K_b = K_{bb} - K_{bi}\ K_{ii}^{-1}\ K_{ib} \tag{4-7}$$

$$F_b' = F_b - \bar{F}_b \tag{4-8}$$

Then the real solutions u are given by

$$u = \bar{u} + u' \tag{4-9}$$

Substituting $\bar{u}_b=0$ to the equation (4-9), we can get the following solutions for u_b.

$$u_b = K_b^{-1}(F_b - K_{bi}\, K_{ii}^{-1}\, F_i) \tag{4-10}$$

Substituting these results into equation (4-5), we may also obtain the other solutions for u_i.

$$u_i = K_{ii}^{-1}(F_i - K_{ib}\, u_b) \tag{4-11}$$

The internal degrees of freedom u_i can be statically condensed out from equation (4-11). Thus the overall structural equilibrium equation may be rewritten as

$$[K_b]_{total}\ \{u_b\}_{total} = \{F_b\}_{total} \tag{4-12}$$

Here K_b is called a boundary stiffness matrix and represents the stiffnesses of boundary degrees of freedom. Because the boundary stiffness matrix K_b is much smaller than the total stiffness matrix K, handling the stiffness matrix K_b is much easier than handling the total stiffness matrix K.

In the nonlinear dynamic analysis, the basic equation to be solved using Houbolt method is[9]

$${}^{t}\hat{K}\,\Delta u(i) = {}^{t+\Delta t}F - {}^{t+\Delta t}\hat{f}(i-1) \tag{4-13}$$

where ${}^{t}\hat{K}$ is an effective stiffness matrix at time t given

by

$$ {}^{t}\hat{K} = \frac{2}{(\Delta t)^2} M + \frac{11}{6\Delta t} C + K \qquad (4\text{-}14) $$

and

$$ \Delta u(i) = {}^{t+\Delta t}u(i) - {}^{t+\Delta t}u(i-1) \qquad (4\text{-}15) $$

$$ {}^{t+\Delta t}\hat{f}(i-1) = \text{sum of the right hand side of equation (3-4) except F} $$

The superscripts t, $t+\Delta t$ and (i) refer to time step and iteration number, respectively. The stiffness matrix K is given by a tangent stiffness matrix at time t, and the ${}^{t+\Delta t}\hat{f}(i-1)$ is an equivalent nodal force vector correspond to stresses of elements at $(i-1)^{th}$ iteration at time $t+\Delta t$.

In principle this equation is established repeatedly at each time step and solved again using iterative method. However many practical problems deal with local nonlinearity only. Therefore one can reduce the computational efforts by seperating linear portion from nonlinear part. In order to do this, the effective stiffness matrix ${}^{t}K$ may be divided into linear and nonlinear parts.

$$ {}^{t}\hat{K} = \hat{K} + {}^{t}K^{n} \qquad (4\text{-}16) $$

Here $\hat{K}$ and ${}^{t}K^{n}$ represent all linear and nonlinear degrees of freedom, respectivrly. Thus $\hat{K}$ is constant and ${}^{t}K^{n}$ is time dependent. If the subscripts 1 and n are employed to define the linear and nonlinear degrees of freedom, the basic governing equation (4-13) can be rewritten in partitioned

form:

$$\left[\begin{bmatrix} K_{11} & K_{1n} \\ K_{n1} & K_{nn} \end{bmatrix} + \begin{bmatrix} 0 & 0 \\ 0 & {}_tK^n \end{bmatrix}\right]\begin{Bmatrix} \Delta u_1^{(i)} \\ \Delta u_n^{(i)} \end{Bmatrix} = \begin{Bmatrix} t+\Delta t F_1 \\ t+\Delta t F_n \end{Bmatrix} - \begin{Bmatrix} t+\Delta t \hat{f}_1^{(i-1)} \\ t+\Delta t \hat{f}_n^{(i-1)} \end{Bmatrix} \quad (4\text{-}17)$$

Performing static condensation process, one obtains

$$(\hat{K}_r + {}_tK^n)\,\Delta u_n^{(i)} = {}^{t+\Delta t}F_r - {}^{t+\Delta t}\hat{f}_n^{(i-1)} - \hat{K}_r\, {}^{t+\Delta t}u^{(i-1)} \quad (4\text{-}18)$$

where

$$\hat{K}_r = \hat{K}_{nn} - \hat{K}_{n1}\,\hat{K}_{11}^{-1}\,\hat{K}_{1n} \quad (4\text{-}19)$$

$${}^{t+\Delta t}F_r = {}^{t+\Delta t}F_n - \hat{K}_{n1}\,\hat{K}_{11}^{-1}\,{}^{t+\Delta t}F_1 \quad (4\text{-}20)$$

Except for using an effective stiffness matrix and reduced load vector, equations (4-18) to (4-20) are equivalent to equations (4-6) to (4-8). The differences are that the nonlinear part of the effective stiffness matrix ${}_tK^n$ must be updated and factorized at each time step, and the reduced load vector ${}^{t+\Delta t}F_r$ must be newly calculated at the beginning of each time step. The iteration process is also required in each time step due to nonlinearity. Thus the substructuring technicque can be applied much in the same way as in the static analysis to reduce the degrees of freedom.

C. ANSYS Technique

Substructuring technique in ANSYS can be achieved by designating "master degrees of freedom" in using superelement. The superelement, a simple collection of elements that are reduced to act as one element, is develpoed upon the following assumptions within substructure[13]:

(1) All elements are linear.
(2) All material properties are constant, time independent.
(3) Unsymmetric element matrices are not permitted.

In principle, the solution procedures in static analysis are the same as those in the previous case, except for defining the master degrees of fredom as the boundary degrees of freedom. In dynamic analysis, however, the solution procedures are far more simplified because of the above basic assumptions.

ANSYS does not take the partitioning and static condensation procedures because of awkwardness to implement[14]. Instead ANSYS uses the Guyan reduction procedure to reduce the degrees of freedom in generating the superelement. This reduction technique gives the following reduced dynamic equilibrium equation[14], and the equation is solved directly.

$$M \ddot{u} + C \dot{u} + K u = F \qquad (4\text{-}21)$$

where

$$M = M_{mm} - K_{ms} K_{ss}^{-1} M_{sm} - M_{ms} K_{ss}^{-1} K_{sm} + K_{ms} K_{ss}^{-1} M_{ss} K_{ss}^{-1} K_{sm}$$

$$C = C_{mm} - C_{ms} K_{ss}^{-1} C_{sm} - C_{ms} K_{ss}^{-1} K_{sm} + K_{ms} K_{ss}^{-1} C_{ss} K_{ss}^{-1} K_{sm}$$

$$K = K_{mm} - K_{ms} K_{ss}^{-1} K_{sm}$$

$$F = F_m - K_{ms} K_{ss}^{-1} F_s$$

$$u = u_m$$

Subscripts m and s designate the correspondence of master and slave degrees of freedom.

The superelement can be used in any analysis type. The solution procedures of substructuring technique consist of three steps. The first step is a generation step. In this step, the reduced load vector and thereduced stiffness, mass and damping matrices are caculated. The second step is a use step. In this step, the superelement is used as one ordinary element with other elements. The only reduced displacements related to the master degrees of freedom. The third step is a stress step. In this step, the full set of displacements and stresses interior to the superelement are calculated. Since the internal displacements and stresses of any superelement are caculated independently from other superelements, one can get the solutions of any intereting sub-domain only without solving the whole equations[13].

CHAPTER V. NUMERICAL EXAMPLES

A. Pendulum Problem

1. Mathematical model

An initially rested pendulum, which is an equilibrium state, is released at time t=0. The problem is to predict its motions in time domain due to gravitation. It has two degrees of freedom as shown in Fig. 5-1-1.

This example is solved for two cases, (1) spring constant K = infinite (i.e., $3*10^5$ (lb/in)) and (2) K = 100(lb/in)

2. Data and Solutions

This problem is solved using KAN=4, nonlinear transient dynamicanalysis. The mass is defined using generalized mass element, JSTIF=21. The spring is modeled using spring-damper element, JSTIF=14. The input data are given in Appendix and the solution curves are given in Fig. 5-1-2 to Fig. 5-1-5, respectively.

3. Discussion

The equations of motion of this pendulum may be deriven using Lagrangean dynamics. Lagrangian L is defined by

$$L = T-V$$

$$= \frac{1}{2} m(\dot{q}_1^2 q_2^2 + \dot{q}_2^2) + mgq_2 \cos q_1 - \frac{1}{2} K(q_2 - l_0)^2 \qquad (5\text{-}1\text{-}1)$$

where l_0 is the length of spring at equilibrium state. Since the potential energy V is independent of the velocity, the Lagrange's equation can be written as

$$\frac{d}{dt}\frac{\partial L}{\partial \dot{q}_i} - \frac{\partial L}{\partial q_i} = 0 \quad , \; i = 1,2, \qquad (5\text{-}1\text{-}2)$$

Thus the governing equation is

$$q_2 \ddot{q}_1 + 2\dot{q}_1 \dot{q}_2 + g \sin q_1 = 0 \qquad (5\text{-}1\text{-}3)$$

$$m\ddot{q}_2 - m\dot{q}_1^2 q_2 - mg \cos q_1 + K(q_2 - l_0) = 0 \qquad (5\text{-}1\text{-}4)$$

If K is infinite as in case (1), the equations may be reduced to one degree of freedom:

$$\ddot{q}_1 + \frac{g}{l} \sin q_1 = 0 \qquad (5\text{-}1\text{-}5)$$

This equation is nonlinear and the solution is given by the Jacobian elliptic funtion[20]. For the given initial conditions of case (1), the solution is

$$\sin \frac{\theta}{2} = \frac{1}{2} \, \mathrm{sn}\, 6.216(t - 0.2712) \qquad (5\text{-}1\text{-}6)$$

where sn is the Jacobian elliptic funtion[21]. The period of this equation is given by

$$T = 4\sqrt{l/g}\, F(\pi/2\ ,\ k) = 1.085 \qquad (5\text{-}1\text{-}7)$$

where $F(\pi/2\ ,\ k)$ is the complete elliptic integral of the first order. The ampletute of this solution is also given by

$$|u_x| = 2*10\sin 60^{\circ} = 17.32 \qquad (5\text{-}1\text{-}8)$$

$$|u_y| = 10*(1 - \cos 60^{\circ}) = 5 \qquad (5\text{-}1\text{-}9)$$

These values are almost same as the numerical solutions given in Fig. 5-1-4.

In case (2), the solution procedure is not so simple. The one possible way is to use a computer program such as DE, which is a general purpose automatic computer program to solve ordinary differential equations. However, one can infer from the results of case (1) that the solution of case (2) may also be valid. Thus the more detail verification is omitted.

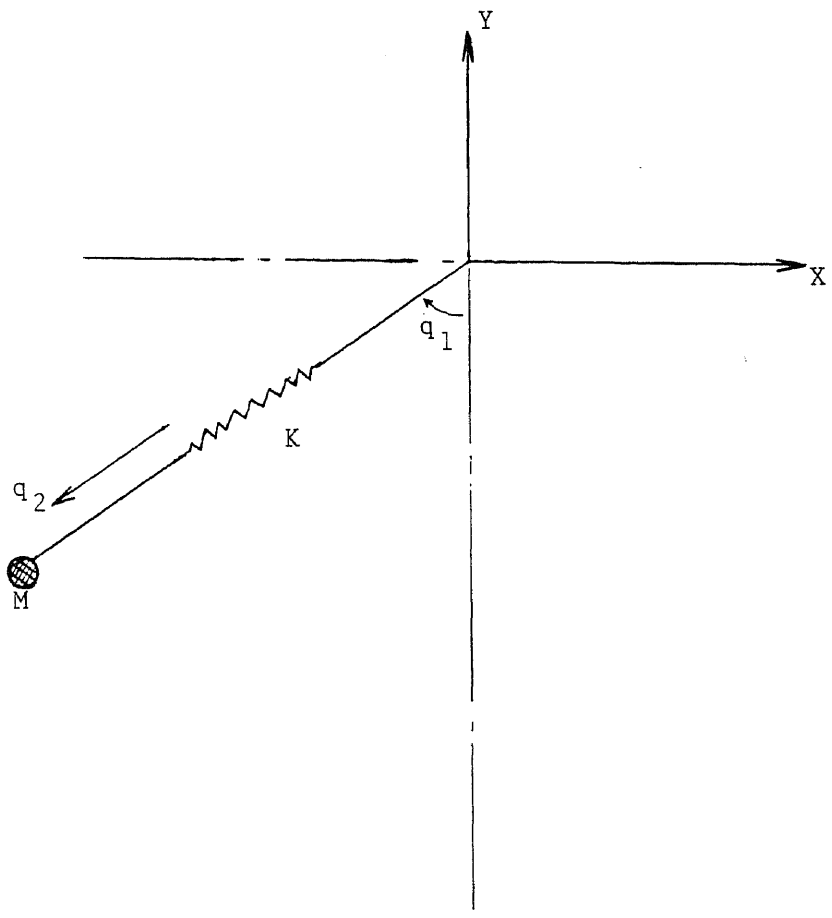

Fig. 5-1-1: Mathematical Model of Pendulum

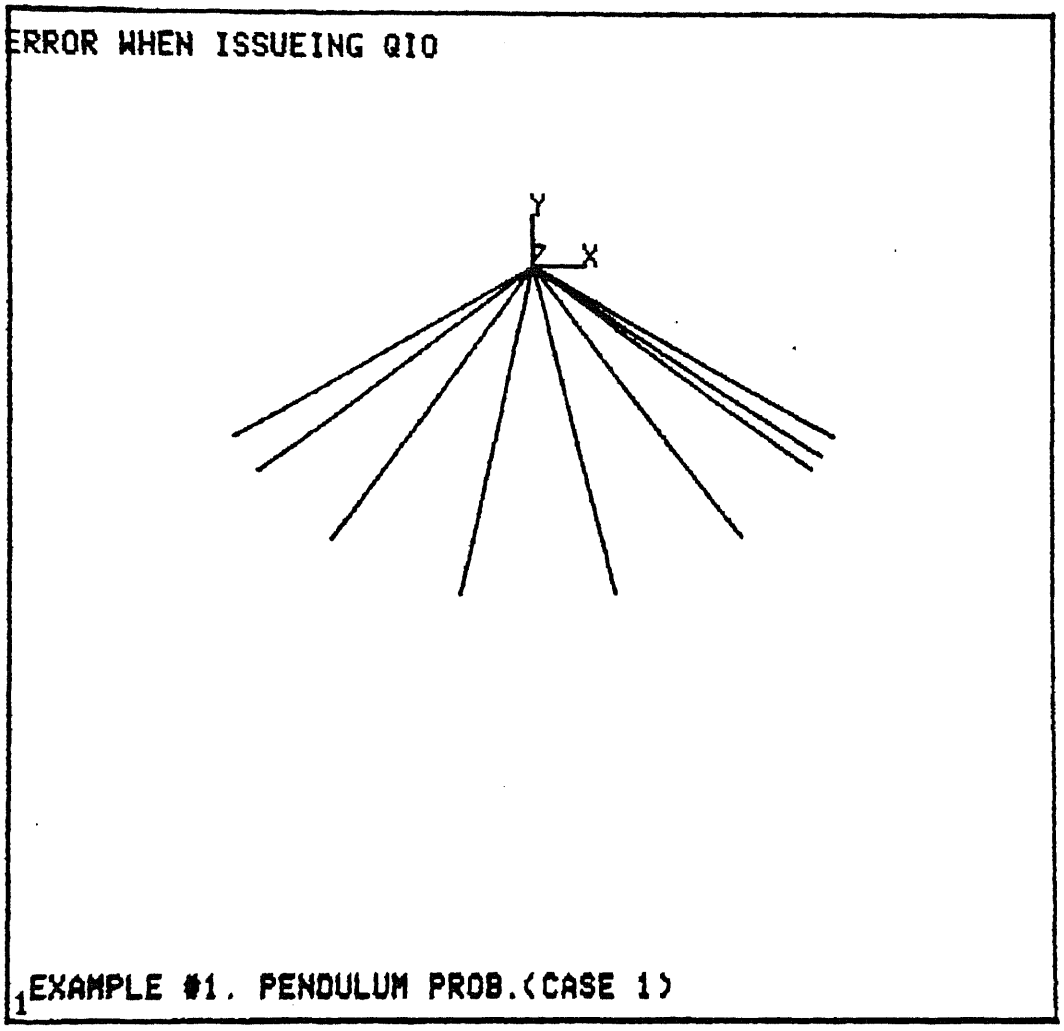

Fig. 5-1-2: Motion of Pendulum for case (1)

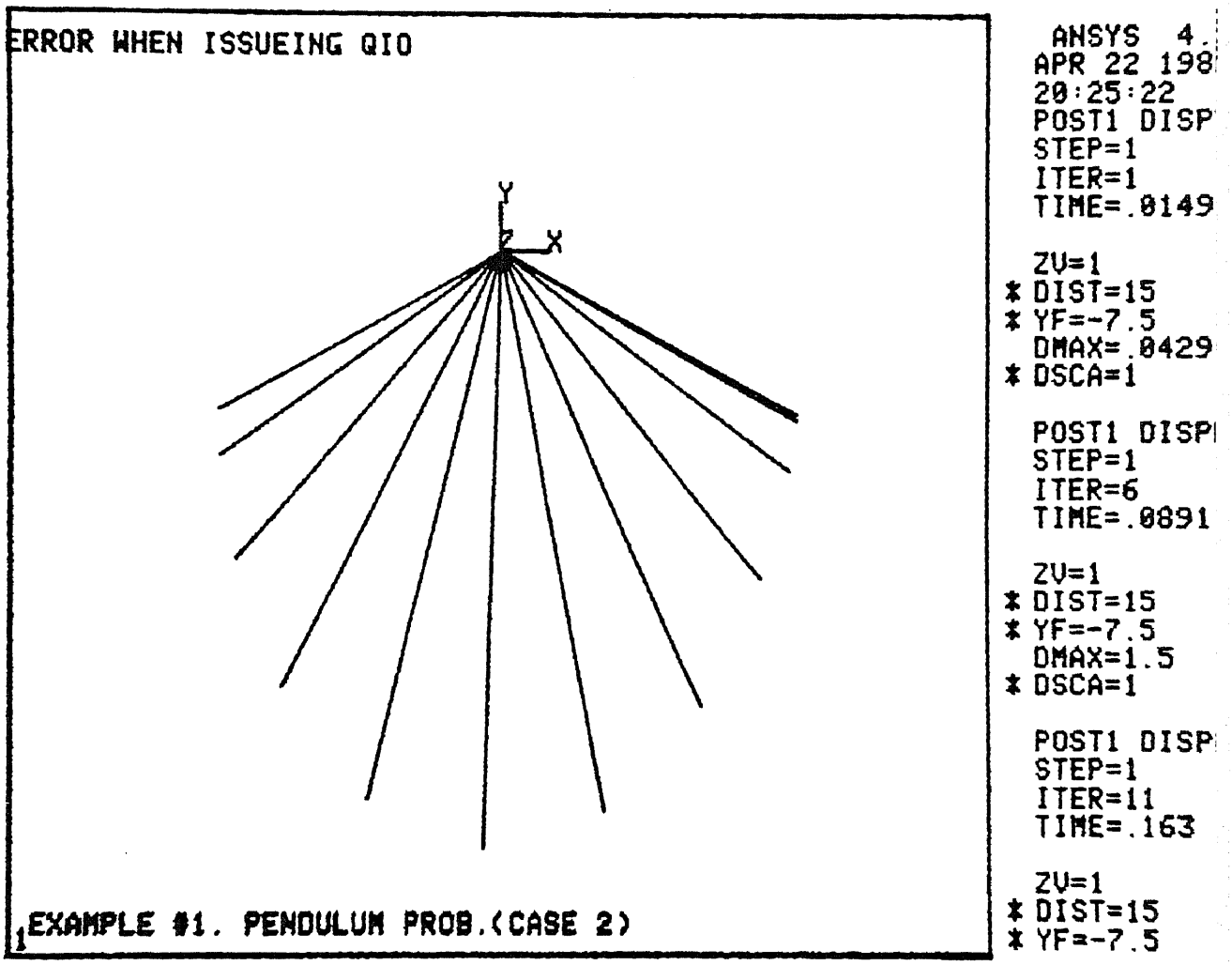

Fig. 5-1-3: Motion of Pendulum for Case (2)

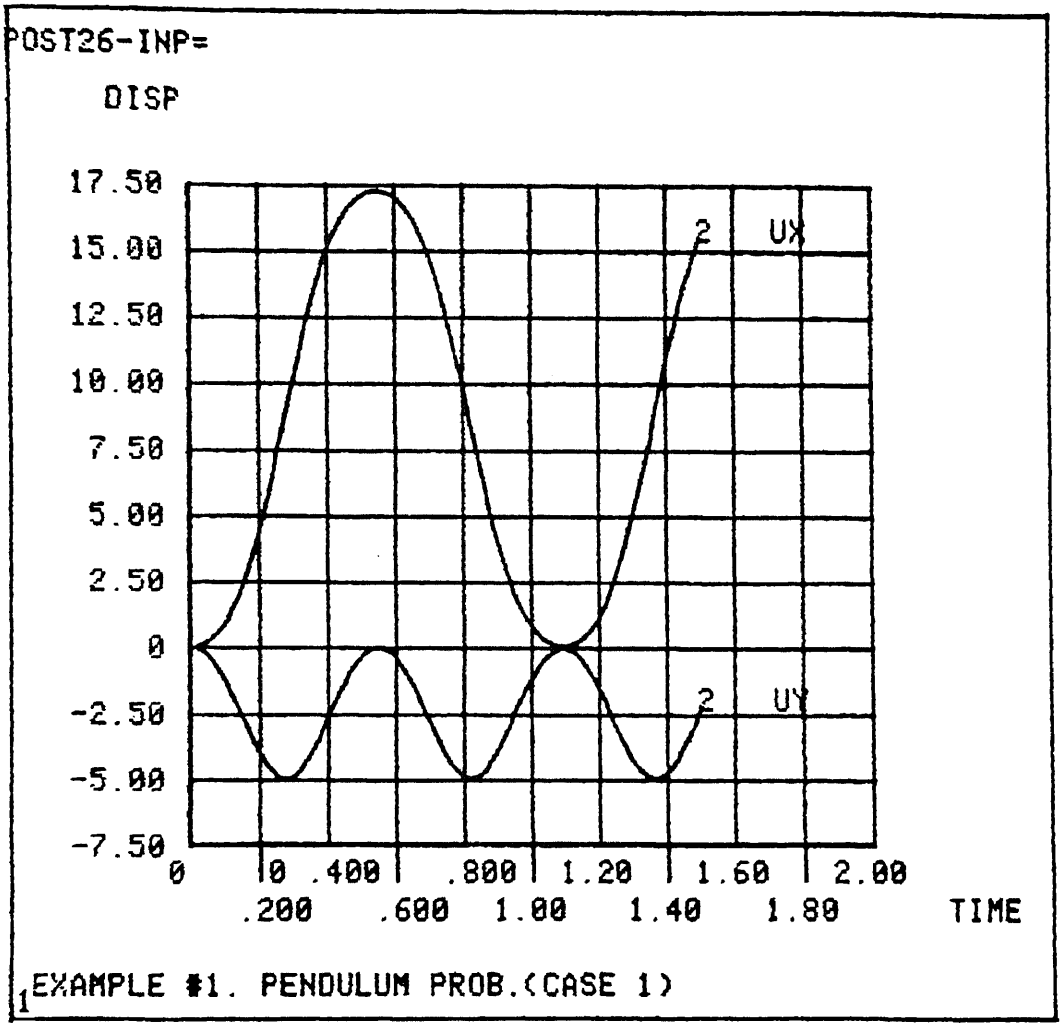

Fig. 5-1-4: Displacement vs Time for Case (1)

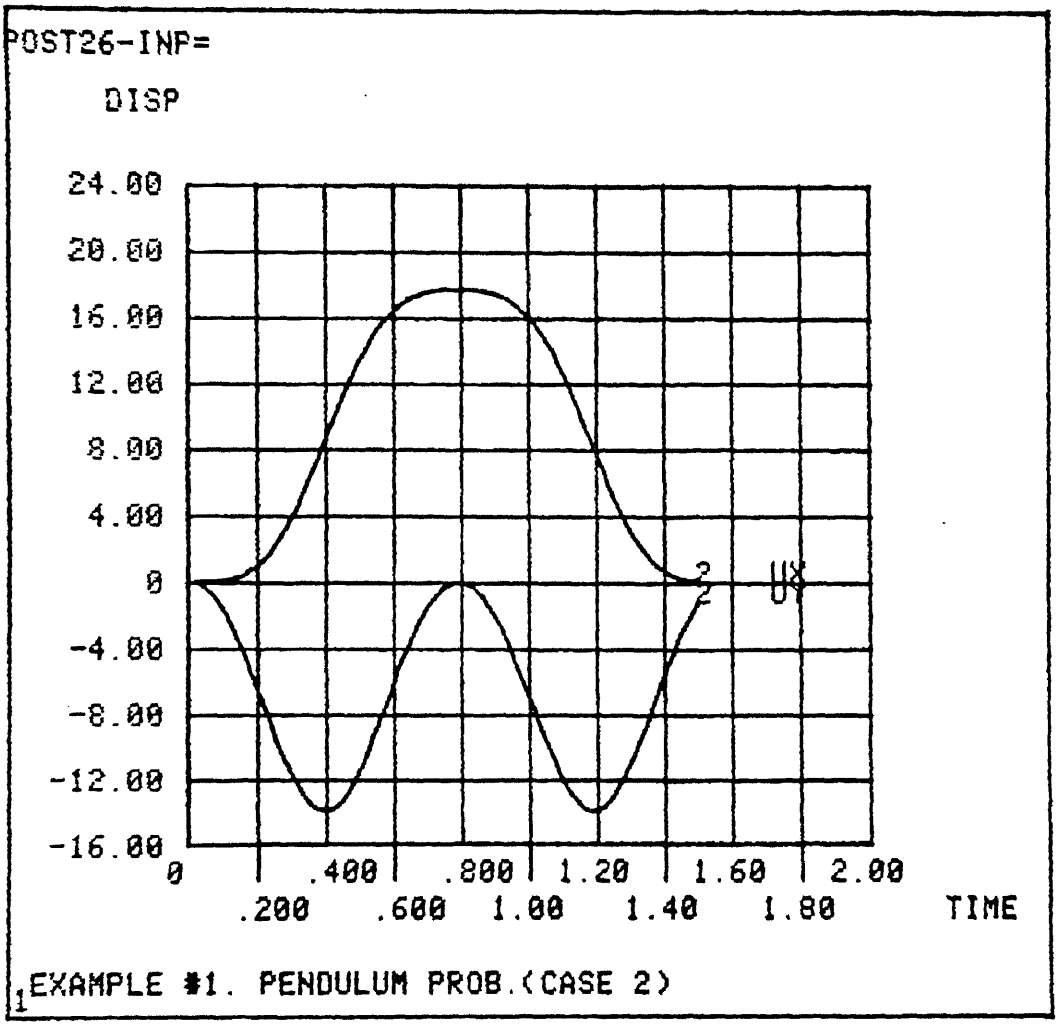

Fig. 5-1-5: Displacement vs Time for Case (2)

B. Car Crash Problem

1. Mathematical Model

The car is modelled as shown in Fig. 5-2-1 using a moving spar with constant velocity, v = 100 (in/sec). The material properties used are for steel. The nodal masses may be determined by lumped mass distribution. The problem is solved for three different contact stiffnesses.

Integration time step Δt and applied force F at each node are determined from 1/30f and $d(mv)/dt = (mv)/\Delta t$, where f is the highest frequency of intrest.

(1) Contact Stiffness K = 1000 (lb/in)

$$t = 1/30*f = 2\pi/30\sqrt{K/m} = 0.0001 \text{ (sec)}$$

$$F = d(mv)/dt = (mv)/\Delta t = m*10^6 \text{ (lb)}$$

(2) Contact Stiffness K = 10000 (lb/in)

$$t = 1/30*f = 2\pi/30\sqrt{K/m} = 0.00004 \text{ (sec)}$$

$$F = d(mv)/dt = (mv)/\Delta t = m*2.5*10^6 \text{ (lb)}$$

(3) Contact Stiffness K = 100000 (lb/in)

$$t = 1/30*f = 2\pi/30\sqrt{K/m} = 0.00001 \text{ (sec)}$$

$$F = d(mv)/dt = (mv)/\Delta t = m*10^7 \text{ (lb)}$$

The load steps for each case are choosen as shown in Fig. 5-2-2 to Fig. 5-2-4.

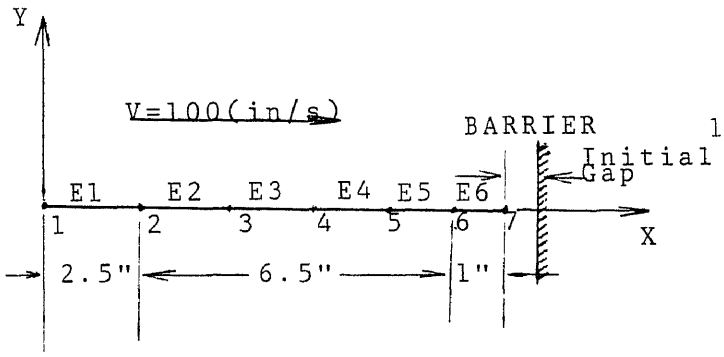

Fig. 5-2-1: Mathematical Model

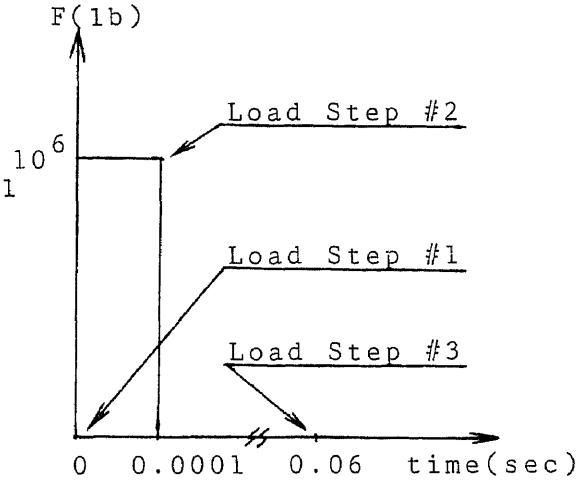

Fig. 5-2-2: Load, Case (1)

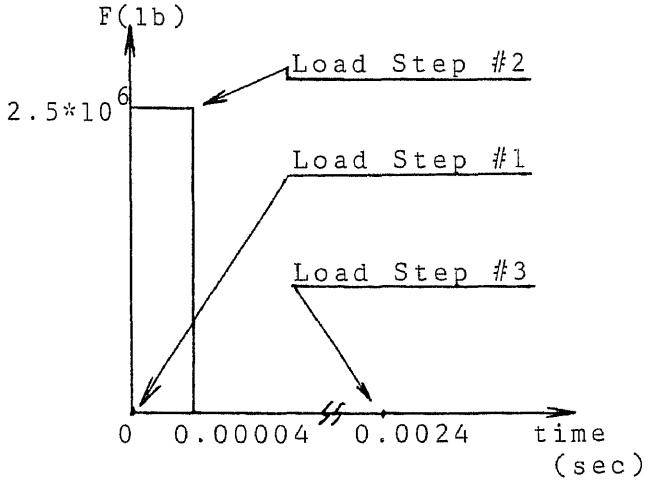

Fig. 5-2-3: Load, Case (2)

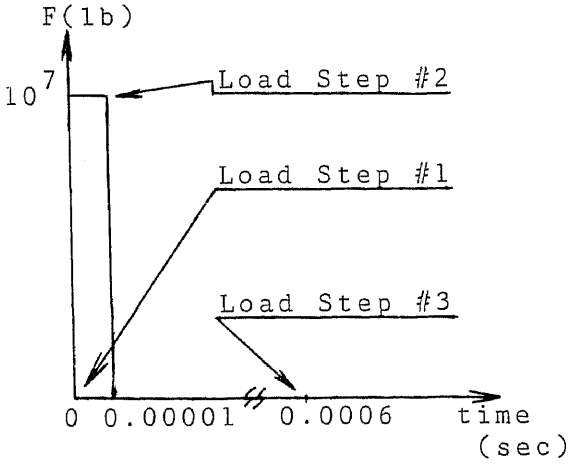

Fig. 5-2-4: Load, case (3)

2. Data and Solutions

The problem is solved using KAN=5, reduced linear transient dynamic analysis. The input data are given in Appendix and the solution curves are given in Fig. 5-2-5 to Fig. 5-2-10, respectively.

3. Discussion

The differential equation of a lateral vibration for elastic body is

$$E \frac{\partial^2 u}{\partial x^2} = \rho \frac{\partial^2 u}{\partial t^2} \qquad (5\text{-}2\text{-}1)$$

Replacing E by $\partial\sigma/\partial\varepsilon$, differential equation (5-2-1) may be modified for the case in which plastic deformation occurs.

$$\frac{\partial \sigma}{\partial \varepsilon} \frac{\partial^2 u}{\partial x^2} = \rho \frac{\partial^2 u}{\partial t^2} \qquad (5\text{-}2\text{-}2)$$

This equation is nonlinear whose analytical solution has never been obtained[18]. Equation (5-2-1) is analytically solved for simple case by S. Timoshenko[19]. However the initial and boundary conditions make the analyticcal solution almost impossible to obtain. The solution may be macroscopically checked by maximum displacement obtained from energy theorem. From the principle of energy conservation,

Kinetic Energy(T) + Potential Energy(V) = Constant (5-2-3)

where

$$T = \frac{1}{2} mv^2 \quad \text{and} \quad V = \frac{1}{2} K(\Delta u)^2 \qquad (5\text{-}2\text{-}4)$$

The maximum displacements caculated from equations (5-2-3) and (5-2-4) for each case are given in Table. 5-2-1 with the results of numerical analysis. If the displacement field is properly determined, the strain and stress may be automatically and correctly determined by the thory of elasticity.

For the third case, displacements between two ends, node 1 and 7, are slightly different, so that the stresses developed at the each side, element 1 and 6, have some time lag and show the waves (see Fig. 5-2-7 and 5-2-10). These phenomena result from the fact that time step (Δt) is less than the velocity of stress wave. The velocity of stress wave may be caculated by

$$c = \sqrt{E/\rho} = 202721.2 \text{ (in/sec)} \qquad (5\text{-}2\text{-}4)$$

The time required for stress transmission from one end to the other end may be determined from:

$$t = \text{length/velocity} = 4.93*10^{-5} \text{ (sec)} \qquad (5\text{-}2\text{-}5)$$

This is larger than time step $\Delta t = 0.00001$. The stress at element 6 is initiated at time t=0.0001(sec), but the stress at element 1 is initiated at time t=0.000148(sec). There-

Table. 5-2-1: Max. Values of Displacement

		Case 1	Case 2	Case 3
Theoretical	Results	Initial Gap	0.1	0.04	0.01
		u	0.27	0.085	0.027
		Total Disp.	0.37	0.125	0.037
Numerical	Results	ANSYS	0.37	0.1254	0.0373

fore the time lag between two points is

$$T_{lag} = 0.0000148 - 0.0001 = 0.000048 \text{ (sec)} \quad (5\text{-}2\text{-}6)$$

The agreement between equation (5-2-5) and (5-2-6) is excellent. However these results must be verified by physical experiments, since numerical experiment is based on the assumed contact stiffnesses and forces.

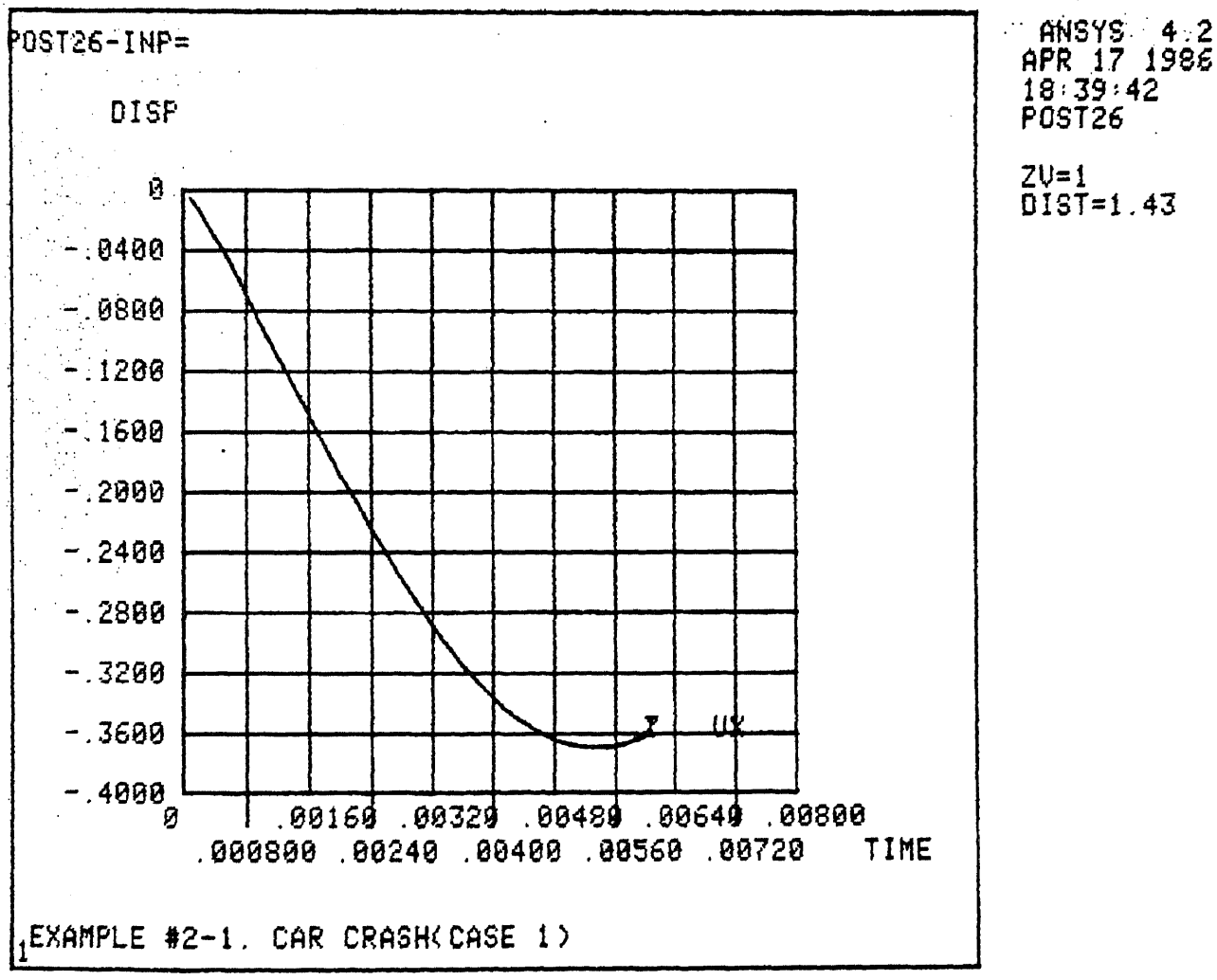

Fig. 5-2-5: Displacement vs Time for case (1)

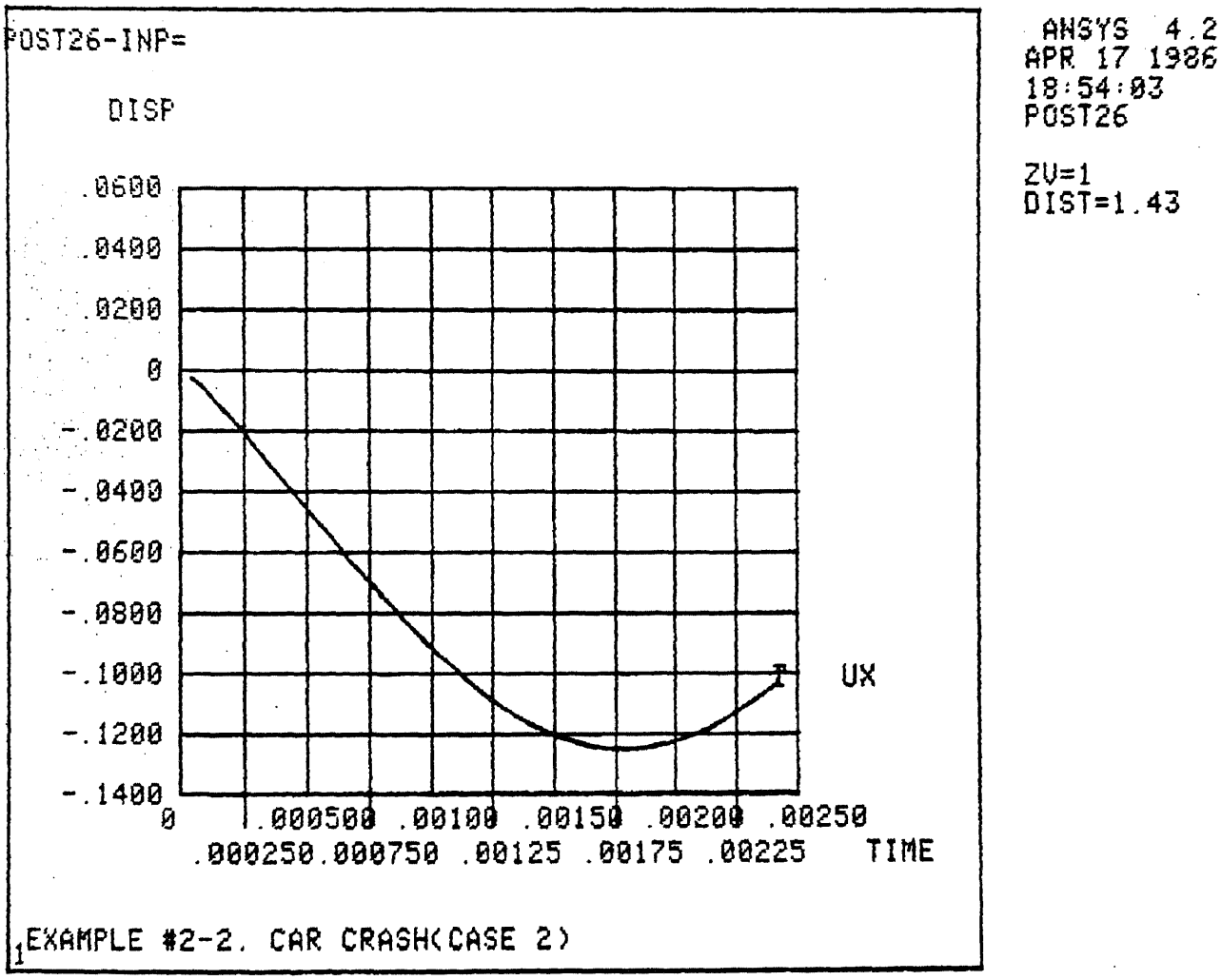

Fig. 5-2-6: Dipslacement vs Time for case (2)

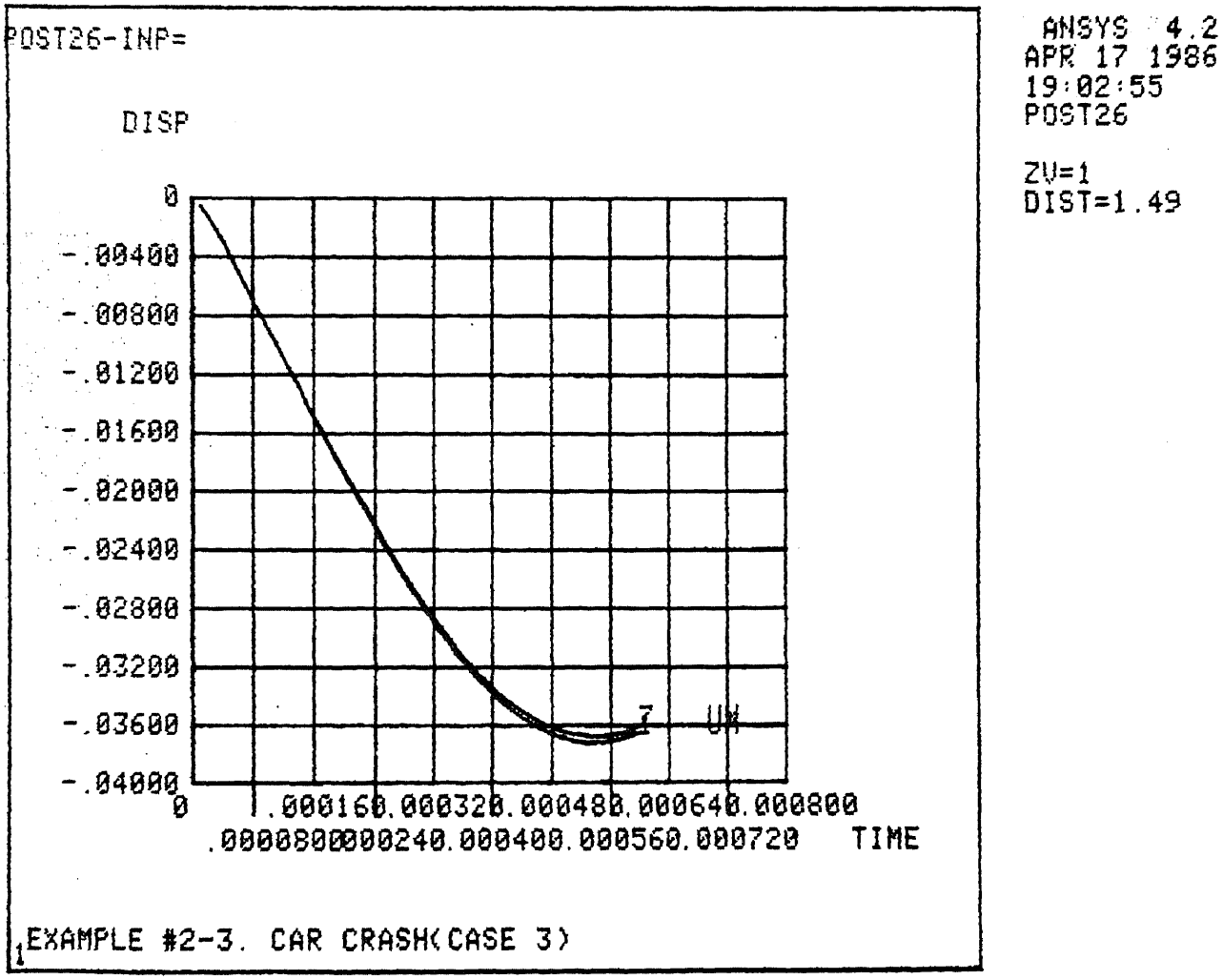

Fig. 5-2-7: Displacement vs Time for Case (3)

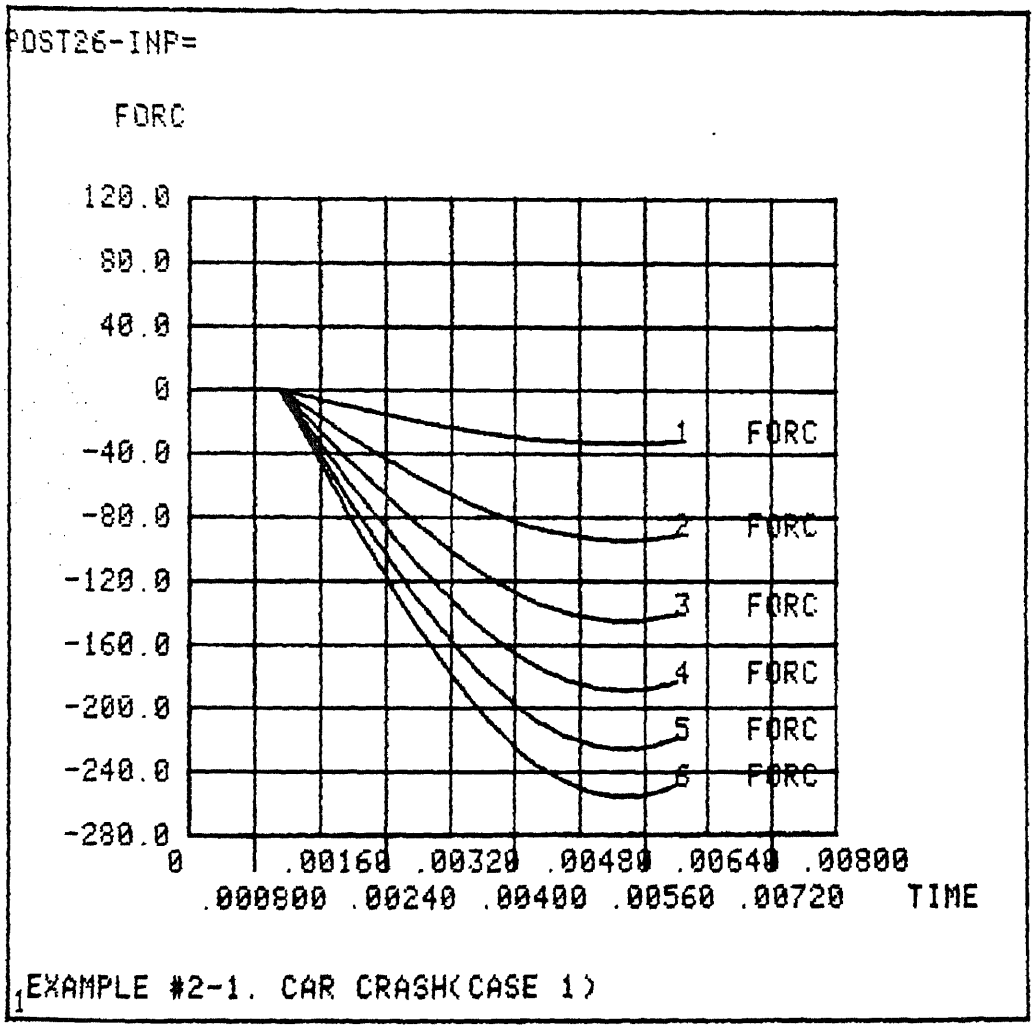

Fig. 5-2-8: Stress vs Time for Case (1)

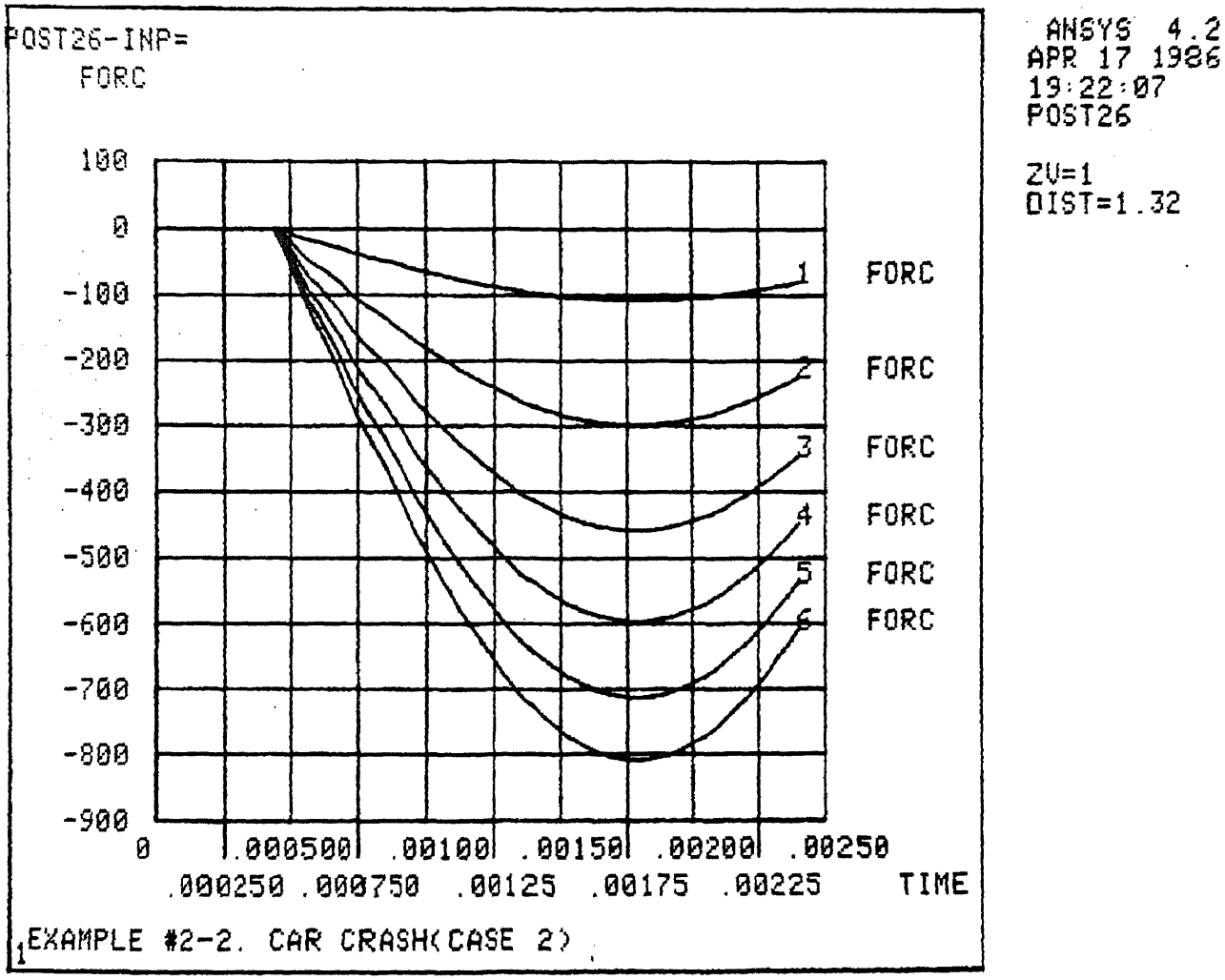

Fig. 5-2-9: Stress vs Time for Case (2)

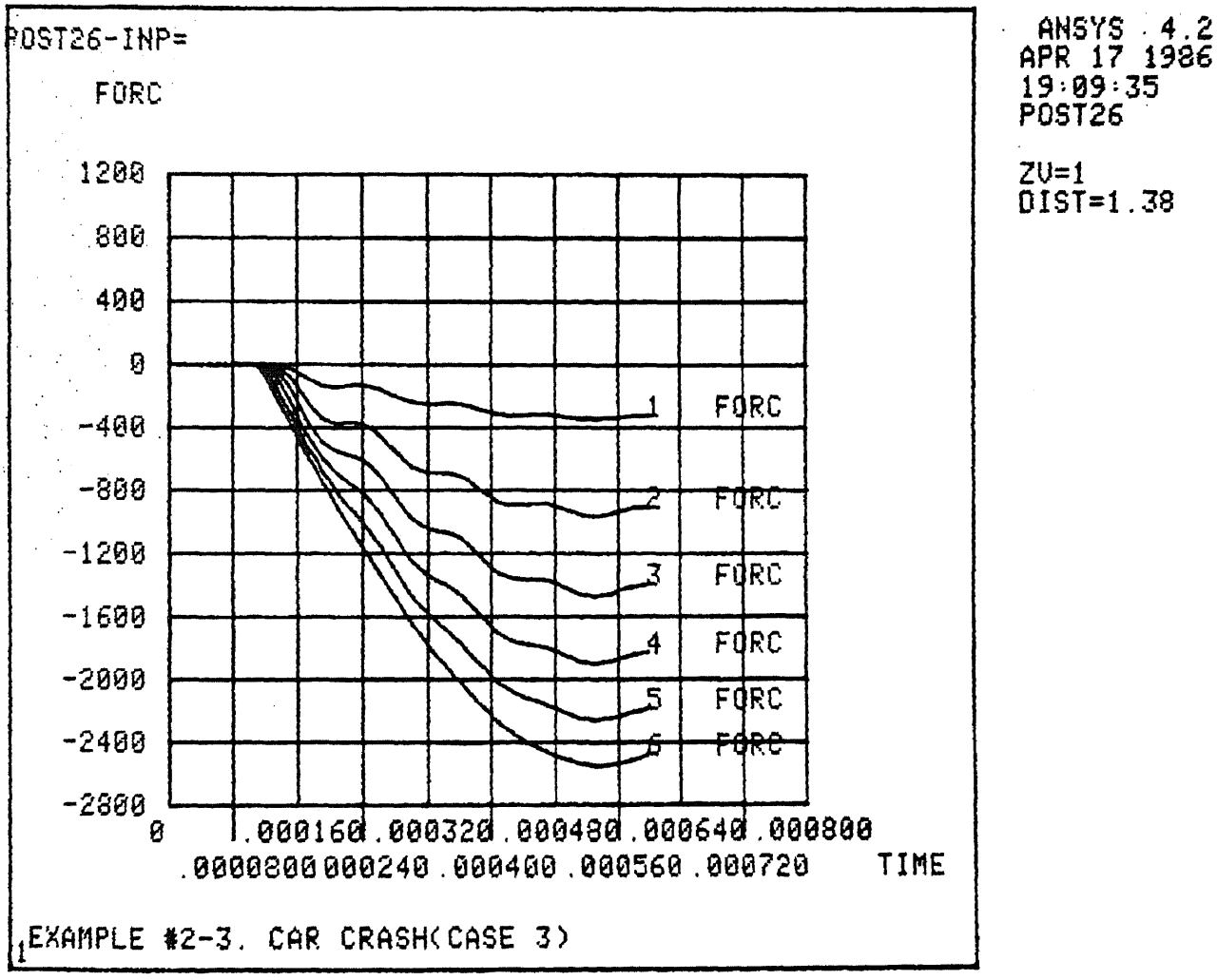

Fig. 5-2-10: Stree vs Time for Case (3)

C. Helmet Problem

1. Mathematical Model

The problem dealt with in this example involve two gap elements in series. The first gap exits between a helmet and ground. The second gap exits between a head and a helmet. The helmet is crudely approximated by a small curved plate which is a part of SPH-4 US Army helmet and modelled using quadrilateral shell elements, JSTIF=63. The head is also crudely approximated by a small rectangular aluminum block and 3-D isoparametric solids, JSTIF=45, is used. This is shown in Fig. 5-3-1.

The contact stiffnesses between helmet and ground and the one between head and helmet are assumed to be 1000(lb/in) and 1650(lb/in), respectively, to give sufficient stiffnesses and to reduce integration time steps. The integration time step Δt is determined by

$$\Delta t = T/33 = 2\pi/33\sqrt{K/m} = 6.667*10^{-5} \text{ (sec)} \qquad (5\text{-}3\text{-}1)$$

There is no external force except for a gravitational force.

2. Data and Solutions

This problem is solved in two ways: (1) without using substructuring technique and (2) using substructuring technique. In the latter solution procedure, the whole quadri-

lateral shell elements approximating a helmet are condenced to one superelement used in solution phase of this analysis.

As can be expected, the former solution procedure requires large memory as well as CPU time, thus the analysis is limited to the first short time period. The CPU time and memory storage needed for each case are compared with in the next section. The input data for each case are given in Appendix. The numerical solutions using method (2) are illustrated in Fig. 5-3-2 to Fig. 5-3-7.

3. Discussion

This example is somewhat more complicated than the previous examples. Since damping effect is neglected, the differential equation has a form

$$M\ddot{u} + K u = M g \qquad (5\text{-}3\text{-}2)$$

To solve this equation, one may first calculate element stiffnesses and then integate in time using direct numerical integation technique. To briefly predict the behavior of the entire structure, one may use a single degree of freedom model whose closed form solution is easily obtained.

Until any of the the gap is closed, the motion is a complete free fall. It is easy to see that the gap between helmet and ground is the first to close. The time taken to close the gap is

$$t_{gap} = 2*Gap/g = 0.07194(sec) \qquad (5\text{-}3\text{-}3)$$

After the gap is closed, the equation of motion and its consistant initial conditions for helmet are

$$M\ddot{x} + K x = M g \qquad (5\text{-}3\text{-}4)$$

$$x=0, \quad \dot{x}=-27.798 \qquad \text{at } t'=0 \qquad (5\text{-}3\text{-}5)$$

here

$$x = u+1, \quad t' = t-0.07194 \qquad (5\text{-}3\text{-}6)$$

The solution satisfling equations (5-3-4) and (5-3-5) is

$$x=-9.7727*10^{-3}\sin\omega' t'+4.7759*10^{-5}(1-\cos\omega' t') \qquad (5\text{-}3\text{-}7)$$

Here ω' is an eigenvalue of this system given by $\omega' = \sqrt{K/M} = 2844.4$(rad./sec). The maximun displacement occurs at time t=0,0725 ($t'=5.522*10^{-4}$)(sec) with magnitude

$$u_{Max} = -1.0097 \text{ (in)} \qquad (5\text{-}3\text{-}8)$$

The maximum acceleration also occurs at time t=0.0725(sec) with magnitude

$$u_{Max} = 79067 \text{ (in/sec}^2\text{)} \qquad (5\text{-}3\text{-}9)$$

There is an excellent agreement between these numbers and numerical solutioins.

Since the spring is linear, the gap reopens at time t=0.073(sec). Thus the above solutions are valid during this time period only. The time when the gap between head and helmet is closed may be determined by

$$gt^2/2 + 27.798(t-0.073) - g(t-0.073)^2/2 = 1.1 \qquad (5\text{-}3\text{-}10)$$

Thus, the gap reopens at time t=0.07425(sec).

The numerical solutions show that the slope of the curves of displacements and accelerations changes its magnitude at this time instant. Furthermore, one can see that the period of those curves are approximately 0.002. This agrees well with exact solution, $T=2\pi/\omega=2.20*10^{-3}$(sec). The solution curve, Fig. 5-3-3, shows that the displacement of node 41 is slightly bigger than that of the crown of helmet, i.e, node 43. This is very reasonable. In conclusion, the numerical solution does make sense. However, the contact stiffeness which was assumed previously without verification must be determined by experiments.

TheVAX 11/780 computer at Mechanical Engineering Department of NJIT is used for this analysis. The CPU time and disk memory needed for this analysis of each case are:

	CPU(sec)	memory(block)
case (1)	5485.08	25644
case (2)	1152.31(56.69)	10391(1829)

In case (1), the above values are for carrying out 70 iterations, which means the analysis is only carried out for 0.072(sec) only. In case (2), the above values are for 190 iterations for full analysis, up to t=0.08. The values inside parentheses show portions for generating superelement. Thus one can conclude that the substructuring technique is a very powerluf technique, especially for large structural analysis.

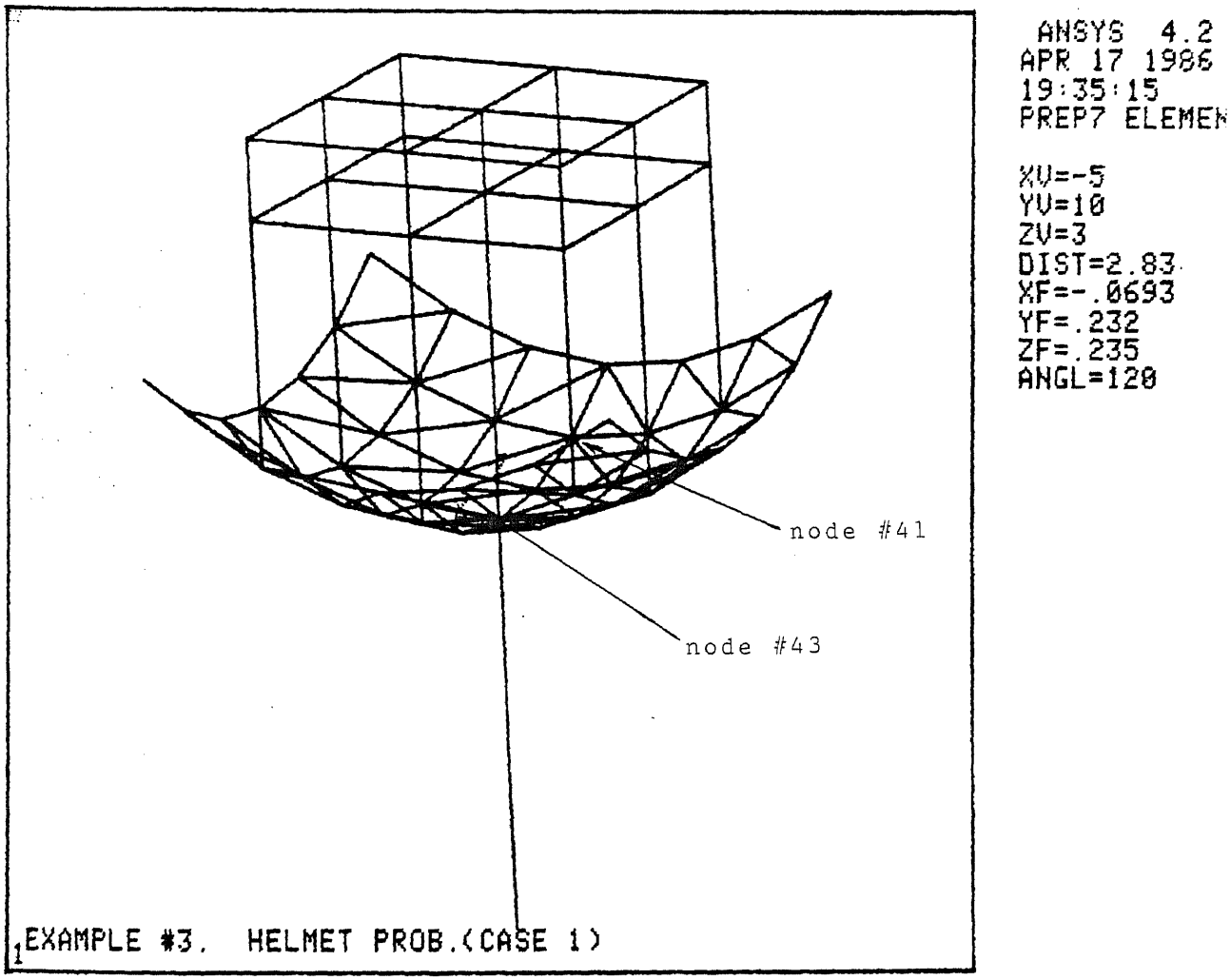

Fig. 5-3-1: Mathematical Model

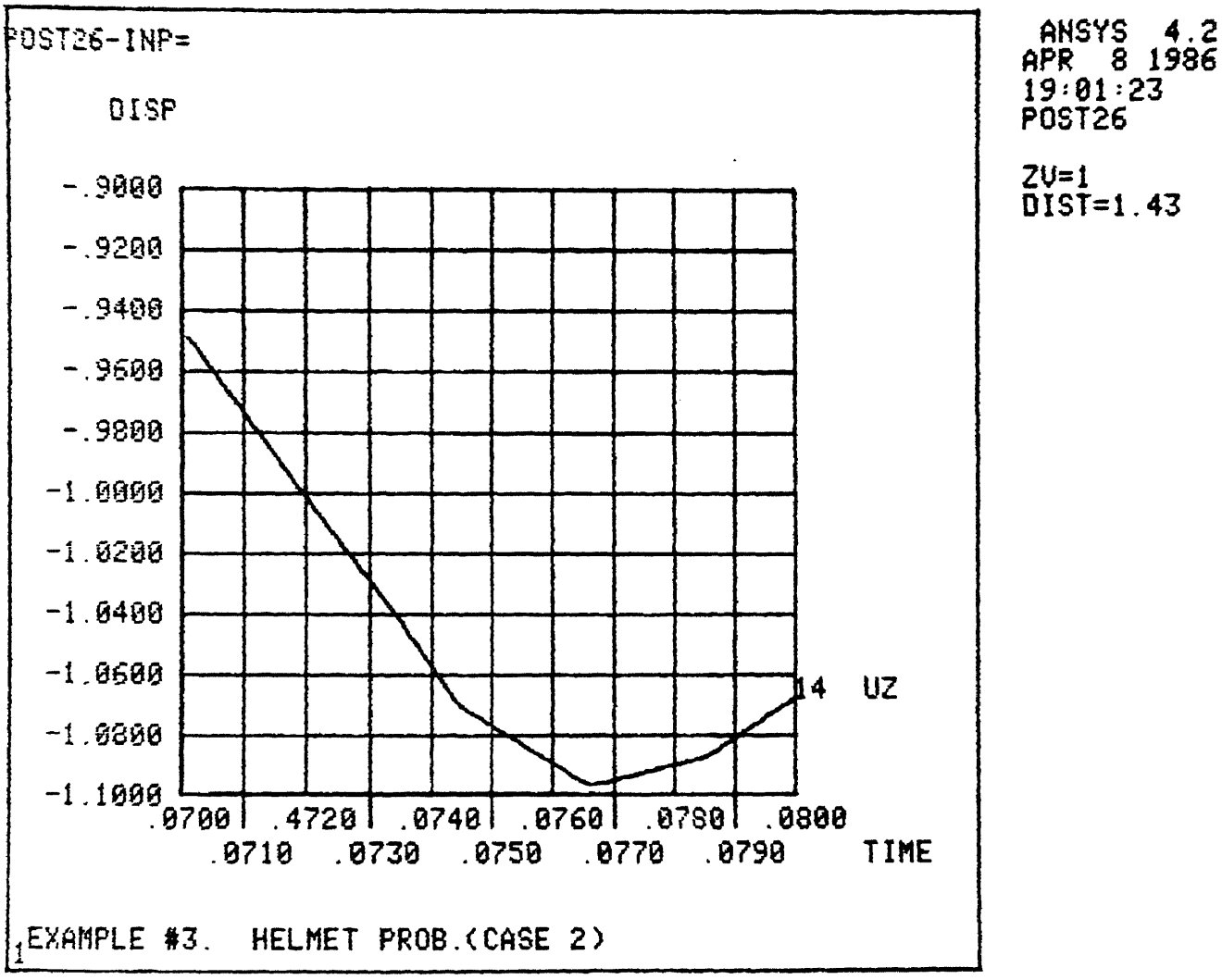

Fig. 5-3-2: Displacement of Head vs Time

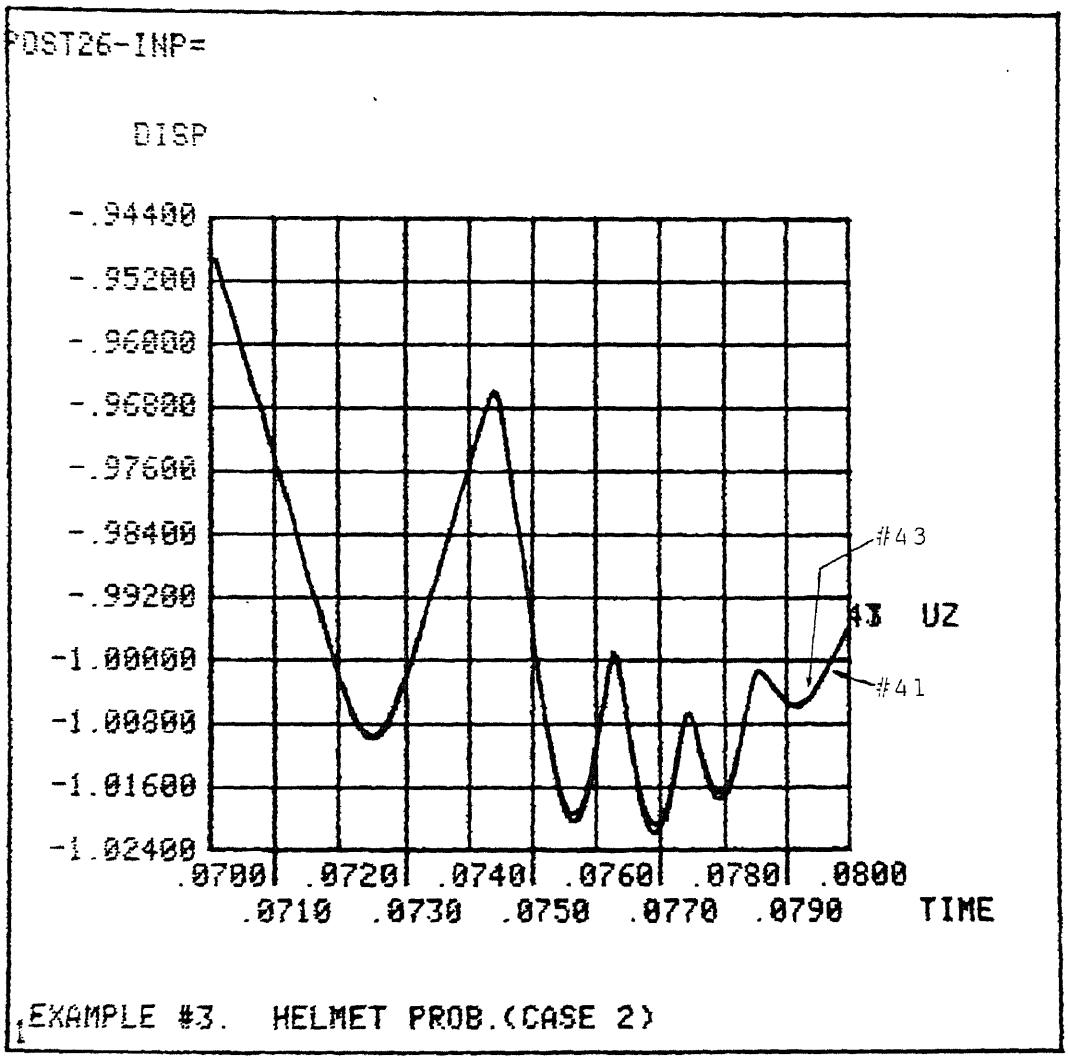

Fig. 5-3-3: Displacement of Helmet vs Time

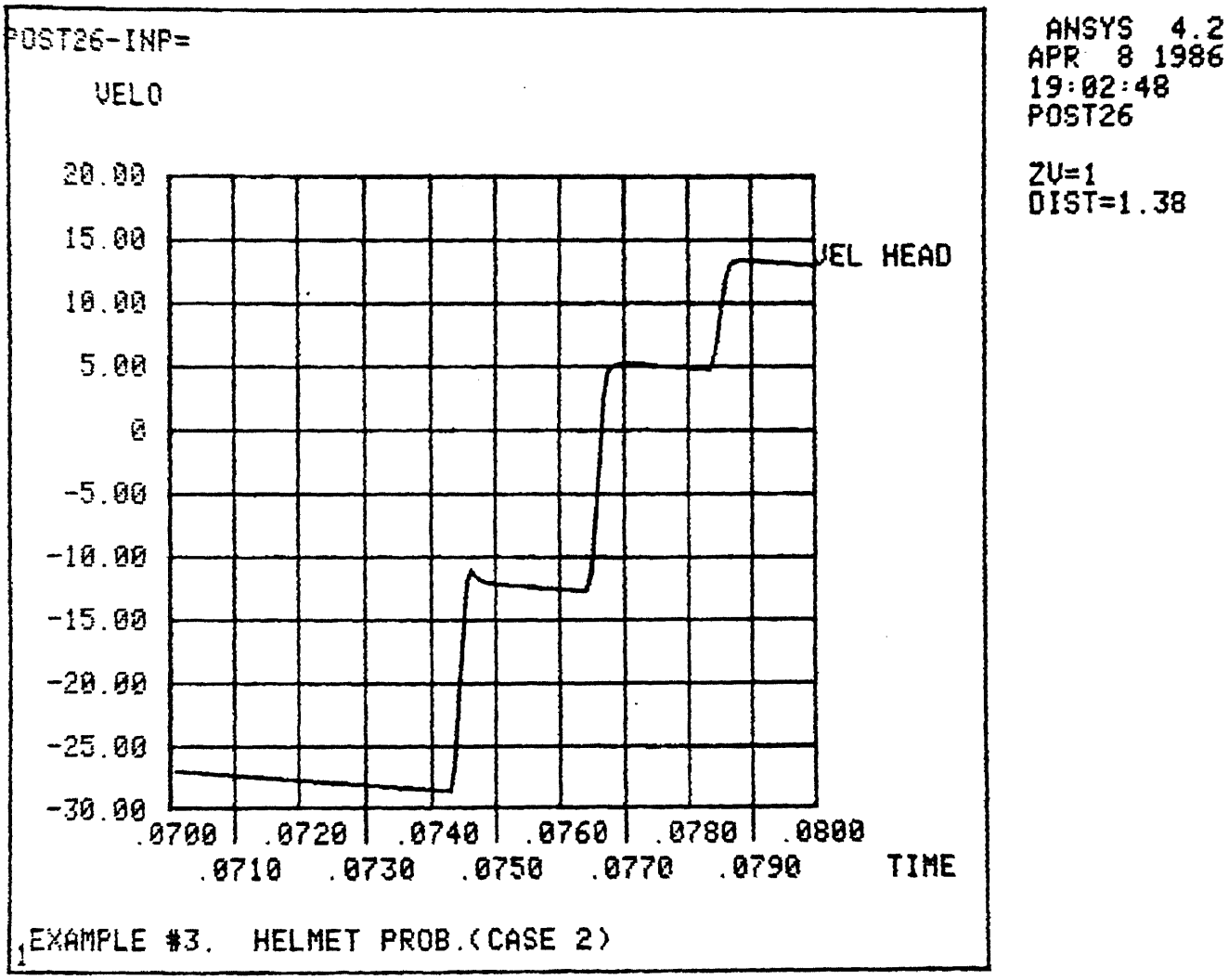

Fig. 5-3-4: Velocity of Head vs Time

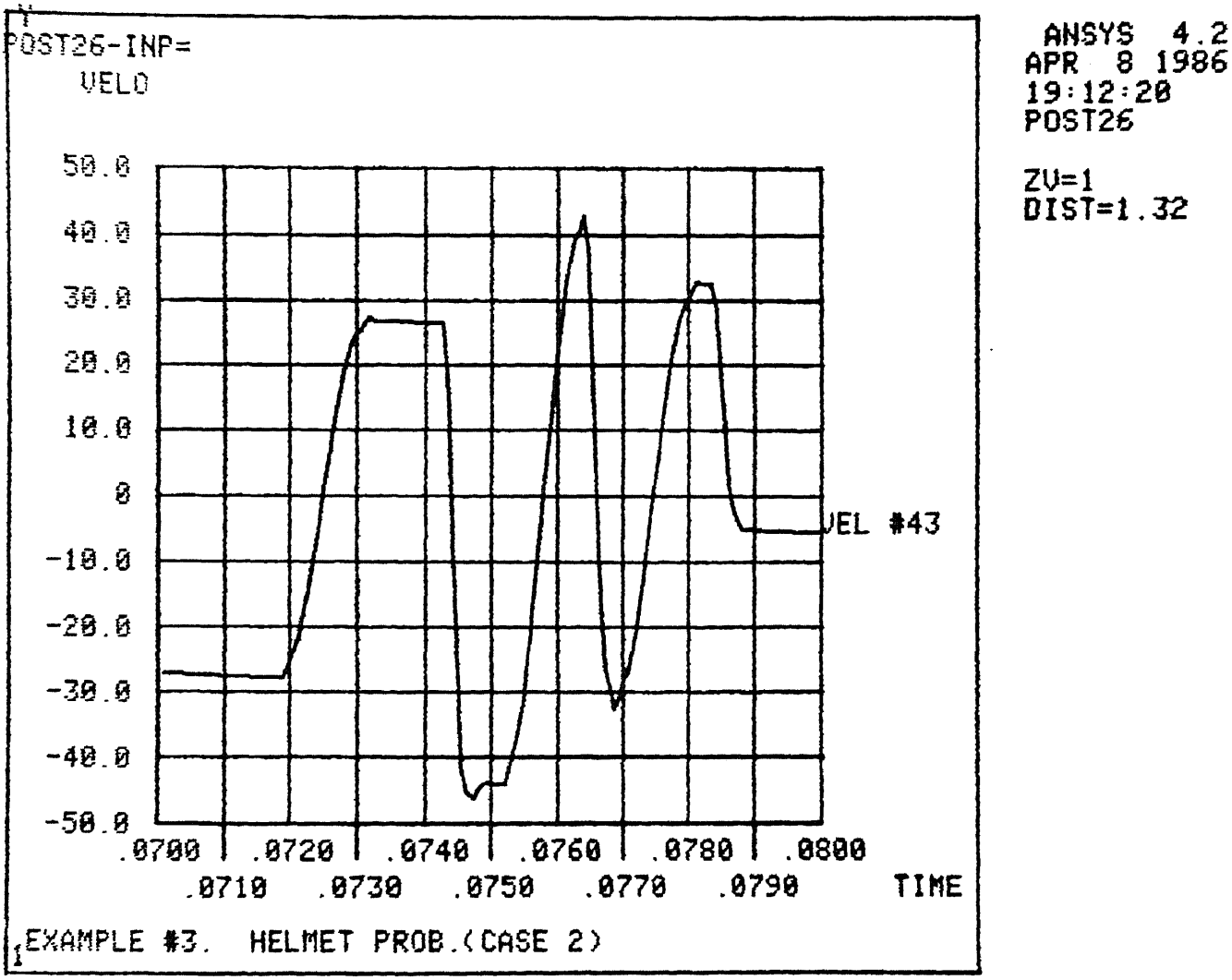

Fig. 5-3-5: Velocity of Helmet vs Time

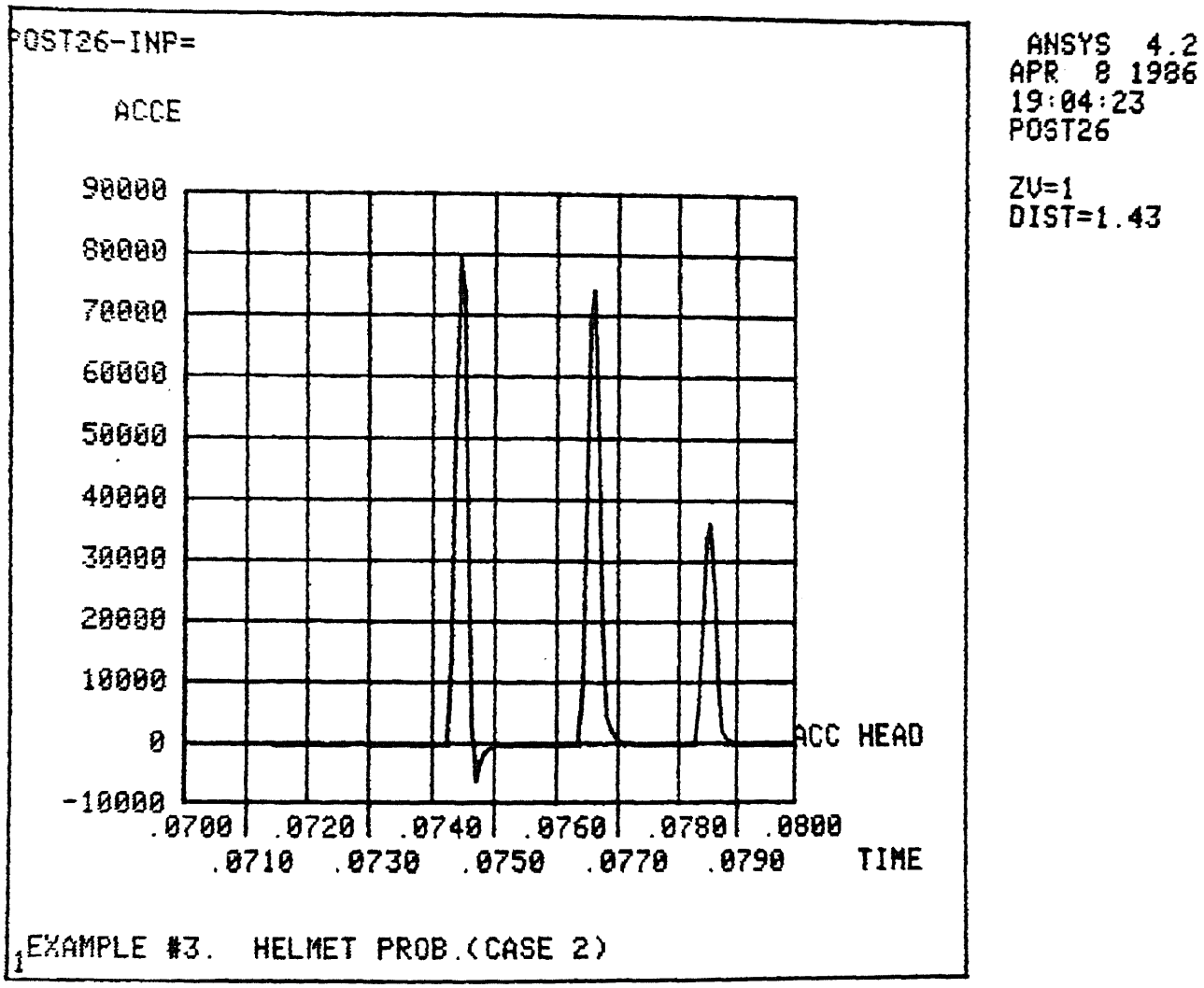

Fig. 5-3-6: Acceleration of Head vs Time

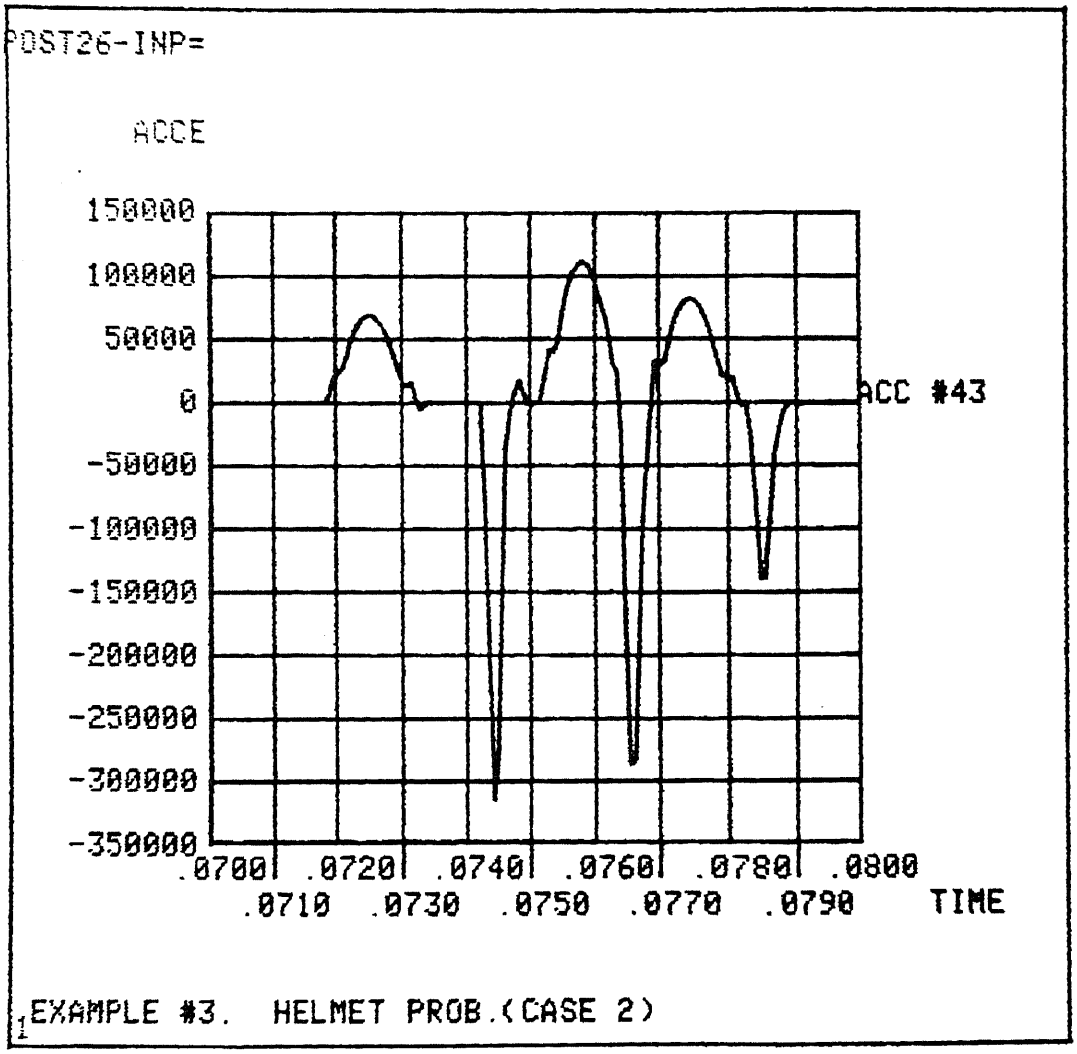

Fig. 5-3-7: Acceleration of Helmet vs time

CHAPTER VI. CONCLUSION

A. Concldsing Remarks

This thesis demonstrated the use of ANSYS package for the analysis of impact dynamics. Both substructuring analysis technique and gap element technique are fully discussed.

Looking at the growth of increasing needs of impact dynamics, the demands of numerical analysis technique for the problem is expanding rapidly. This thesis is hoped to facilidate the use of ANSYS finite element package for the beginner, especially in the are of impact dynamics for which the ANSYS manual fails to give clear explanations.

B. Future Study

In this thesis, all the impact was exclusively assumed to be elastic. However, in actual situation, this will not happen but involve local plasticity phenomina. Researches of developing mathematical model including this effect must be done for more meaningful numerical simulations.

The gap element technique explained in this thesis may be too crude to simulate reasonable impact situation, and also too restricted from numerical analysis point of view. Development of better techniques replacing gap element in finite element environment could be an excellent reseach topic.

APPENDIX. INPUT DATA LISTS FOR ANSYS

A. Pendulum Problem

```
/PREP7
/TITLE,EXAMPLE #1. PENDULUM PROB.(CASE 1)
KAN,4
ET,1,8
ET,2,21,,,2
KAY,5,1
KAY,6,1
R,1,0.1
EX,1,30E6
R,2,1.0
N,1
N,2,-8.66025,-5.
E,1,2
TYPE,2
REAL,2
E,2
ITER,101,5,1
TIME,1.5
ACEL,,386.4
KBC,1
D,1,UZ,,,2
D,1,UX,,,,,UY
AFWRITE
FINISH
/INPUT,27
FINISH
/INTER
/SHOW
/POST1
/NOPR
/DSCALE,,1
/FOCUSE,,,-7.5
/DIST,,15
*SET,NJIT,1,5
*BEGIN,MARC
SET,1,NJIT
PLDISP
*END
/NOERASE
*DO,MARC,1,20
FINISH
[EOB]
*
```

```
    1       /PREP7
    2       /TITLE,EXAMPLE #1. PENDULUM PROB.(CASE 2)
    3       KAN,4
    4       ET,1,14,,,2
    5       ET,2,21,,,2
    6       KAY,5,1
    7       KAY,6,1
    8       R,1,100
    9       R,2,1.0
   10       N,1
   11       N,2,-8.66025,-5.
   12       E,1,2
   13       TYPE,2
   14       REAL,2
   15       E,2
   16       ITER,101,5,1
   17       TIME,1.5
   18       ACEL,,,386.4
   19       KBC,1
   20       D,1,UZ,,,2
   21       D,1,UX,,,,,UY
   22       AFWRITE
   23       FINISH
   24       /INPUT,27
   25       FINISH
    1       /INTER
    2       /SHOW
    3       /POST1
    4       /NOPR
    5       /DSCALE,,1
    6       /FOCUSE,,,-7.5
    7       /DIST,,15
    8       *SET,NJIT,1,5
    9       *BEGIN,MARC
   10       SET,1,NJIT
   11       PLDISP
   12       *END
   13       /NOERASE
   14       *DO,MARC,1,20
   15       FINISH
[EOB]
*
```

B. Car Crash Problem

```
/PREP7
/TITLE,EXAMPLE #2-1. CAR CR
KAN 5
ET,,1,1
KAY,1,1
EX,1,30E6
DENS,1,.00073
NUXY,1,.33
R,1,1
N,1
,2,2.5
,3,4.5
,4,6.25
,5,7.75
,6,9.
,7,10.
,10,15
E,1,2
EGEN,6,1,1
M,1,UX,7
GP,10,7,FX,1000,.1
ITER,1,1,1
DTIME,.0001
KBC,1
F,1,FX,,,7
LWRITE
TIME,.0001
F,1,FX,-912.5
,2,FX,-1642.5
,3,FX,-1368.75
,4,FX,-1186.25
,5,FX,-1003.75
,6,FX,-821.25
,7,FX,-365
LWRITE
ITER,1,1,1
TIME,.006
F,1,FX,,,7
LWRITE
AFWRITE
FINISH
/INPUT,27
FINISH
/STRESS
ITER,1,1,1
NSTRES,1
TIME,.0001
ITER,1,1,1
NSTRS,60
TIME,.006
END
FINISH
[EOB]
*
```

```
/PREP7
/TITLE,EXAMPLE #2-2. CAR CR
KAN,5
ET,1,1,1
KAY,1,1
EX,1,30E6
DENS,1,.00073
NUXY,1,.33
R,1,1
N,1
,2,2.5
,3,4.5
,4,6.25
,5,7.75
,6,9.
,7,10.
,10,15
E,1,2
EGEN,6,1,1
M,1,UX,7
GP,10,7,FX,10000,.04
ITER,1,1,1
DTIME,.00004
KBC,1
F,1,FX,,,7
LWRITE
TIME,.00004
F,1,FX,-2281.25
,2,FX,-4106.25
,3,FX,-3421.875
,4,FX,-2965.625
,5,FX,-2509.375
,6,FX,-2053.125
,7,FX,-912.5
LWRITE
ITER,1,1,1
TIME,.0024
F,1,FX,,,7
LWRITE
AFWRITE
FINISH
/INPUT,27
FINISH
/STRESS
ITER,1,1,1
NSTRES,1
TIME,.00004
ITER,1,1,1
NSTRS,60
TIME,.0024
END
FINISH
[EOB]
*
```

```
/PREP7
/TITLE,EXAMPLE #2-3. CAR CR
KAN,5
ET,1,1,1
KAY,1,1
EX,1,30E6
DENS,1,.00073
NUXY,1,.33
R,1,1
N,1
,2,2.5
,3,4.5
,4,6.25
,5,7.75
,6,9.
,7,10.
,10,15
E,1,2
EGEN,6,1,1
M,1,UX,7
GP,10,7,FX,100000,.01
ITER,1,1,1
DTIME,.00001
KBC,1
F,1,FX,,,7
LWRITE
TIME,.00001
F,1,FX,-9125.
,2,FX,-16425.
,3,FX,-13687.5
,4,FX,-11862.5
,5,FX,-10037.5
,6,FX,-8212.5
,7,FX,-3650.
LWRITE
ITER,1,1,1
TIME,.0006
F,1,FX,,,7
LWRITE
AFWRITE
FINISH
/INPUT,27
FINISH
/STRESS
ITER,1,1,1
NSTRES,1
TIME,.00001
ITER,1,1,1
NSTRS,60
TIME,.0006
END
FINISH
[EOB]
*
```

C. Helmet Problem

```
*
/PREP7
/TITLE,EXAMPLE #3. CASE (1)
KAN,4
ET,1,63,,,,,,,1
ET,2,45,1
ET,3,40,,,3,,,2
KAY,5,1
KAY,6,1
KAY,8,1
EX,1,2.6E6
EX,2,10E6
NUXY,1,.25
NUXY,2,.33
DENS,1,1.965E-4
DENS,2,2.588E-4
R,1,.06
R,2,1650,,,,.1
R,3,1000,,,1.0
N,19,-1.5,-1.5,1.125
,20,-1.5,-1.0,0.8125
,21,-1.5,-0.5,0.625
,22,-1.5,,0.5625
,23,-1.5,0.5,0.625
,24,-1.5,1.0,0.8125
,25,-1.5,1.5,1.125
,26,-1.0,-1.5,0.8125
,27,-1.0,-1.0,0.5
,28,-1.0,-0.5,0.3125
,29,-1.,,,.25
,30,-1.,,.5,.3125
,31,-1.,1.,,.5
,32,-1.,1.5,.8125
,33,-.5,-1.5,.625
,34,-.5,-1.,,.3125
,35,-.5,-.5,,.125
,36,-.5,,,.0625
,37,-.5,,.5,,.125
,38,-.5,1.,,.3125
,39,-.5,1.5,,.625
,40,,-1.5,,.5625
,41,,-1.,,.25
,42,,-.5,,.0625
,43,,,,0
,44,,,.5,,.0625
,45,,1.,,.25
,46,,1.5,,.5625
NGEN,2,14,33,39,1,1
,2,28,26,32,1,2
,2,42,19,25,1,3
EN,1,20,19,26,26
,4,30,23,22,22
,7,26,27,20,20
,10,22,29,30,30
,37,48,41,40,40
,40,44,43,50,50
,43,40,47,48,48
,46,50,51,44,44
ENGEN,1,3,1,1
,1,3,1,4
,1,3,1,7
,1,3,1,10
,1,3,1,37
,1,3,1,40
,1,3,1,43
,1,3,1,46
ENGEN,12,3,7,1,3
,12,3,7,4,6
,12,3,7,7,9
,12,3,7,10,12
```

```
 70        ,12,3,7,37,39
 71        ,12,3,7,40,42
 72        ,12,3,7,43,45
 73        ,12,3,7,46,48
 74        N,1,-1,-1,1.5
 75        N,3,-1,1,1.5
 76        FILL
 77        NGEN,3,3,1,3,1,1
 78        N,10,-1,-1,1
 79        ,12,-1,1,1
 80        FILL
 81        NGEN,3,3,10,12,1,1
 82        N,27,-1.,-1,.5
 83        ,31,-1,1,.5
 84        ,29,-1,,.25
 85        ,41,,-1,.25
 86        ,43,,,,
 87        ,45,,1,.25
 88        ,55,1,-1,.5
 89        ,57,1,,.25
 90        ,59,1,1,.5
 91        ,68,,,-2.
 92        TYPE,2
 93        MAT,2
 94        EN,73,10,13,14,11,1,4,5,2
 95        ENGEN,1,2,1,73
 96        ENGEN,2,2,3,73,74
 97        TYPE,3
 98        REAL,2
 99        E,27,10
100        ,31,12
101        ,55,16
102        ,59,18
103        E,29,11
104        ,41,13
105        ,45,15
106        ,57,17
107        E,43,14
108        TYPE,3
109        REAL,3
110        E,68,43
111        M,10,UZ,18
112        M,27,UZ,31,2
113        ,41,UZ,45,2
114        ,55,UZ,59,2
115        ITER,15,5,1
116        TIME,0.06
117        ACEL,,,386.4
118        KBC,1
119        D,68,ALL
120        D,1,UX,,,18,,UY,ROTX,ROTY
          ,ROTZ
121        D,27,UX,,,31,2,UY,ROTX,RO
          TY,ROTZ
122        D,41,UX,,,45,2,UY,ROTX,RO
          TY,ROTZ
123        D,55,UX,,,59,2,UY,ROTX,RO
          TY,ROTZ
124        KRF,1
125        LWRITE
126        ITER,25,5,1
127        TIME,0.07
128        ACEL,,,386.4
129        KBC,1
130        KRF,1
131        LWRITE
132        ITER,150,10,1
133        TIME,0.08
134        ACEL,,,386.4
135        KBC,1
136        KRF,1
137        LWRITE
138        AFWRITE
139        FINISH
140        /EXEC
141        /INPUT,27
142        FINISH
```

```
/PREP7
/TITLE,GENERATION SUPERELEM
KAN,7
ET,1,63,,,,,,1
KAY,6,2
KAY,7,1
KAY,8,1
EX,1,2.6E6
NUXY,1,.25
DENS,1,1.965E-4
R,1,.06
N,19,-1.5,-1.5,1.125
,20,-1.5,-1.0,0.8125
,21,-1.5,-0.5,0.625
,22,-1.5,,0.5625
,23,-1.5,0.5,0.625
,24,-1.5,1.0,0.8125
,25,-1.5,1.5,1.125
,26,-1.0,-1.5,0.8125
,27,-1.0,-1.0,0.5
,28,-1.0,-0.5,0.3125
,29,-1.,,.25
,30,-1.,.5,.3125
,31,-1.,1.,.5
,32,-1.,1.5,.8125
,33,-.5,-1.5,.625
,34,-.5,-1.,.3125
,35,-.5,-.5,.125
,36,-.5,,.0625
,37,-.5,.5,.125
,38,-.5,1.,.3125
,39,-.5,1.5,.625
,40,,-1.5,.5625
,41,,-1.,.25
,42,,-.5,.0625
,43,,,0
,44,,.5,.0625
,45,,1.,.25
,46,,1.5,.5625
NGEN,2,14,33,39,1,1
,2,28,26,32,1,2
,2,42,19,25,1,3
EN,1,20,19,26,26
,4,30,23,22,22
,7,26,27,20,20
,10,22,29,30,30
,37,48,41,40,40
,40,44,43,50,50
,43,40,47,48,48
,46,50,51,44,44
ENGEN,1,3,1,1
,1,3,1,4
,1,3,1,7
,1,3,1,10
,1,3,1,37
,1,3,1,40
,1,3,1,43
,1,3,1,46
ENGEN,12,3,7,1,3
,12,3,7,4,6
,12,3,7,7,9
,12,3,7,10,12
,12,3,7,37,39
,12,3,7,40,42
,12,3,7,43,45
,12,3,7,46,48
M,27,UZ,31,2
M,41,UZ,45,2
M,55,UZ,59,2

ITER,1,1,1
D,27,UX,,,31,2,UY,ROTX,ROTY,ROTZ
D,41,UX,,,45,2,UY,ROTX,ROTY,ROTZ
D,55,UX,,,59,2,UY,ROTX,ROTY,ROTZ
LWRITE
AFWRITE
FINISH
/EXEC
/INPUT,27
FINISH
```

```
/PREP7
/TITLE,EXAMPLE #3. ANALYSIS USING SUPERELEM
KAN,4
ET,1,50
ET,2,45,1
ET,3,40,,,3,,,2
KAY,5,1
KAY,6,1
KAY,8,1
EX,2,10E6
NUXY,2,.33
DENS,2,2.588E-4
R,2,1650,,,,.1
R,3,1000,,,,1.0
N,1,-1,-1,1.5
N,3,-1,1,1.5
FILL
NGEN,3,3,1,3,1,1
N,10,-1,-1,1
,12,-1,1,1
FILL
NGEN,3,3,10,12,1,1
N,27,-1.,-1,.5
,31,-1,1,.5
,29,-1,,.25
,41,,-1,.25
,43,,,,
,45,,1,.25
,55,1,-1,.5
,57,1,,.25
,59,1,1,.5
,68,,,-2
E,1,8
TYPE,2
D,55,UX,,,59,2,UY,ROTX,ROTY,ROTZ
KRF,1
LWRITE
ITER,25,5,1
TIME,0.07
ACEL,,,386.4
KBC,1
KRF,1
LWRITE
ITER,150,10,1
TIME,0.08
ACEL,,,386.4
KBC,1
KRF,1
LWRITE
AFWRITE
FINISH
/EXEC
/INPUT,27
FINISH
*
```

```
MAT,2
EN,73,10,13,14,11,1,4,5,2
ENGEN,1,2,1,73
ENGEN,2,2,3,73,74
TYPE,3
REAL,2
E,27,10
,31,12
,55,16
,59,18
E,29,11
,41,13
,45,15
,57,17
E,43,14
TYPE,3
REAL,3
E,68,43
M,10,UZ,18
M,27,UZ,31,2
,41,UZ,45,2
,55,UZ,59,2
ITER,15,5,1
TIME,0.06
ACEL,,,386.4
KBC,1
D,68,ALL
D,1,UX,,,18,,UY,ROTX,ROTY
,ROTZ
D,27,UX,,,31,2,UY,ROTX,RO
TY,ROTZ
D,41,UX,,,45,2,UY,ROTX,RO
TY,ROTZ
*
```

Bibriography

1. A. B. Pifko, R. Winter; Theory and Application of Finite Element Analysis to Structural Crash Simulation. Computers & Structures, Vol. 13, Pa. 277-285, 1981

2. A. K. Noor; Survey of Computer Program for Solution of Nonlinear Structural and Solid Mechanics Problem. Computer & Structures, Vol. 13, Pa. 425-465, 1981

3. J. T. Stadter, R. O. Weiss; Analysis of Contact through Finite Element Gap. Computers & Structures, Vol. 10, Pa. 867-873, 1979

4. E. Stein, P. Wriggers; Calculation of Impact-Contact Problems of Thin Elastic Shells Taking Into Account Geometric Nonlinearity within the Contact Region. Compter Methods in Applied Mechanics & Engineering, Vol. 34, Pa. 861-880, 1982

5. J. Padovan, S. Tovichakchaikul, I. Zead; Finite Element Analysis of Steadily Moving Contact Fields. Computers & Structures, Vol. 18, Pa. 191-200, 1984

6. J. D. Lee, S.Du, H. Leibowitz; Three-Dimensional Finite Element and Dynamic Analysis of Composite Laminate Subject to Impact. Computers & Structures, Vol. 19, Pa. 807-813, 1984

7. J. Padovan, R. Moscarello, J. Stattord, F. Tabaddor; Pantographing Self Adaptive Gap Elements. Computers & Structures, Vol. 20, Pa. 745-758, 1985

8. K. J. Bathe, A. Chaudhary; A Solution Method for Planar and Axisymmetric Contact Problems. Int. J. for Numerical Methods in Engineering, Vol. 21, Pa. 65-88, 1985

9. K. J. Bathe, S. Gracewski; On Nonlinear Dynamic Analysis using Substructuring and Mode Superposition. Computers & Structures, Vol. 13, Pa. 699-707, 1981

10. S. M. Hamdan, N. Patamapongs; Stability and Convergence of Some FiniteElement Algorithms for Nonlinear Elastodynamics. Int. J. for Numerical Methods in Engineering, Vol. 21, Pa. 975-999, 1985

11. A. E. Elui, D. W. Murray; Skyline Algorithms for Multi-level Substructure Analysis. Int. J. for Numerical Methods in Engineering, Vol. 21, Pa. 465-495, 1985

12. D. P. Mondkar, G. H. Powell; ANSR-2 User's Manual. Report No. UCB/EERC - 80/23, Univ. of Cali., Berkey

13. G. J. DeSalvo, J. A. Swanson; ANSYS-User's Manual, Vol. 1,2. Swanson Analysis System, Inc., 1985

14. P. C. Kohnke; ANSYS-Theoritical Manual. Swanson Analysis System, Inc., 1983

15. O. C. Zienkiewicz; The Finite Element Method, 3rd Ed. McGraw-Hill Book Company, 1977

16. K. J. Bathe; Finite Element Procedures in Engineering Analysis. Prentice-Hall, 1981

17. J. S. Przemienieck; Theory of Matrix Structural Analysis. McGraw-Hill Book Company, 1968

18. C. M. Harris, C. E. Crede; Shock and Viration Handbook, Vol. 1. McGraw-Hill Book Company, 1961

19. S. Timoshenko; Viration Problems in Engineering, 2nd Ed. D. Van Nostran Company, Inc., 1937

20. D. T. Greenwood; Principles of Dynamics Prentice-Hall, Inc., 1975

21. W. H. Beyer, Editor Standard Mathematical Tables, 25th edition CRC Press, Inc., 1978