

1991 NASA/ASEE SUMMER FACULTY FELLOWSHIP PROGRAM

**JOHN F. KENNEDY SPACE CENTER
UNIVERSITY OF CENTRAL FLORIDA**

INVESTIGATION OF THE FINITE ELEMENT SOFTWARE PACKAGES AT KSC

PREPARED BY:	Dr. Chu-Ho Lu
ACADEMIC RANK:	Assistant Professor
UNIVERSITY AND DEPARTMENT:	Memphis State University Department of Mechanical Engineering
NASA/KSC	
DIVISION:	Data Systems
BRANCH:	CAD/CAE
NASA COLLEAGUE:	Mr. Hank Perkins Mr. Eddie Bertot
DATE:	August 9, 1991
CONTRACT NUMBER:	University of Central Florida NASA-NGT-60002 Supplement: 6

ACKNOWLEDGMENTS

I would like to thank Eddie Bertot and Hank Perkins of DL-DSD-22 for their support in obtaining the necessary resources to do this research, Dave Paton of MDSSC and Karl Meyer of LSOC for suggesting NASTRAN test cases, Mark Sorger, Shawn Riley and Richard Neely of EG&G for fixing VAX workstation and Intergraph workstation and providing the documentation of MSC/NASTRAN and SDRG/I-DEAS.

Additionally, I would like to thank the administrative support of Dr. E. Ramon Hosler and Mrs. Kari Stiles, University of Central Florida, and Dr. Mark Beymer and Mr. Dennis Armstrong, the director of the summer faculty fellowship program at NASA/KSC.

ABSTRACT

This research report concerns a discussion of the useful and powerful features of NASTRAN and documents three real world problems for the testing of the capabilities of different NASTRAN versions. The test problems involve direct transient analysis, nonlinear analysis and static analysis. The experiences in using highly graphics software packages are also discussed. It is found that (1) MSC/XL can be more useful if it can be improved to generate picture files of the analysis results and to extend its capabilities to support finite element codes other than MSC/NASTRAN, (2) the current version of SDRC/I-DEAS (version VI) may have bugs in the module "Data Loader".

TABLE OF CONTENTS

Sections	Title
I.	INTRODUCTION
II.	NASTRAN DEVELOPMENT AND ITS FUNCTIONS
2.1	Introduction
2.2	Program Architecture
2.3	Rigid Formats
2.4	Direct Matrix Abstraction Program (DMAP)
2.5	Checkpoint/Restart Capability
2.6	Nonlinear Capabilities
2.7	Superelement
2.8	Cyclic Symmetry
III.	TEST PROBLEMS
3.1	Introduction
3.2	Rolling Beam Umbilical System
3.3	SRB Hold-down Post
3.4	Single Pallet Rotation Device Cover Rivet Analysis
IV.	DISCUSSIONS AND CONCLUSIONS

LIST OF ILLUSTRATIONS

Figures	Title
1	An Isometric View of Rolling Beam Umbilical System
2	X_R Displacement History at Carriage/Truss Interface
3	X_R Velocity History at Carriage/Truss Interface
4	X_R Acceleration History at Carriage/Truss Interface
5	The Location and Finite Element Model of SRB Hold-down Post
6	The Deformed Configuration of The SRB Hold-down Post
7	The Stress Field of The SRB Hold-down Post
8	The Finite Element Model of A Single Pallet Rotation Device
9	The Deformed Shape of The Single Pallet Rotation Device
10	The Stress Field of The Single Pallet Rotation Device
11	A Different Angle View for The Stress Field of The Single Pallet Rotation Device

I. INTRODUCTION

The finite element method has been established as a powerful and popular numerical procedure for analyzing structures and continua governed by differential equations. The technique, in general, discretizes the domain of the problem under consideration into a number of geometrically simple regions (elements). The governing equations of the problem are then approximated over each element by continuous functions to formulate so-called element equations. By doing this, a continuous model with infinite degrees of freedom is converted into a discrete model having finite degree of freedom, and a mathematical model including differential equations is generally converted into a mathematical model involving algebraic equations. After these algebraic equations are assembled and the constraints are introduced, the solution and the derived variables can be obtained with the aid of computers.

Instead of writing a finite element program to analyze a specific problem, the modern engineering analysts will turn to commercial finite element packages. In fact, hundreds of commercial finite element programs are available, from small to large. The most well known large general-purpose analysis software packages are NASTRAN, ANSYS and ABAQUS. They provide many different element types, so that almost any conceivable structure, loads and boundary conditions can be treated. Linear problems of statics and dynamics are certainly included. Several nonlinear capabilities are also provided. At John F. Kennedy Space Center (KSC), such computer packages are MSC/NASTRAN (provided by The MacNeal-Schwendler Corporation), SDRC/I-DEAS (provided by Structural Dynamics Research Corporation) and I/FEM (provided by Intergraph Corporation).

Using the commercially available finite element programs to solve problems, one does not begin with differential equations. Instead, there are three basic phases which can be identified (i.e., preprocessing, analysis modeling and solution, and postprocessing). In the preprocessing phase, a continuous media is discretized, element types are selected, loads and constraints are provided, and material and physical properties of the problem are specified. Then the problem is solved in the analysis phase. The derived variables are also computed. The results are finally analyzed and managed in the form of reports or plots in the postprocessing phase. The process is repeated if mesh refinement is necessary.

Although the large general-purpose finite element software packages offer an extremely versatile capability, engineers typically consume more than 65% of their time in the pre- and post-processing processes. It is clear that the effort in these processes can be reduced if other softwares having highly interactive graphics capability can be interfaced. Three such computer software packages are currently available at KSC, i.e., I-DEAS/FEM, I/FEM and MSC/XL. The interfaces between the former two computer packages and MSC/NASTRAN were investigated [1] in which four different combinations of the three software packages were studied in designing a solid rocket booster fixture.

Different from I-DEAS/FEM and I/FEM, which provide linear analysis capability and

can also be interfaced with COSMIC/NASTRAN, ANSYS, ABAQUS for nonlinear analysis, MSC/XL (introduced by MSC during 1990) is a pure pre- and post-processing software package and only supports MSC/NASTRAN. Although the combination of MSC/XL and MSC/NASTRAN provides powerful finite element modeling, analysis and results processing, the cost in using these software packages seems expensive. It would be interesting to know the possibility of replacing MSC/NASTRAN by different general-purpose finite element packages. Because the finite element users at KSC have been familiar with the features of NASTRAN, the computer packages suggested are different versions of NASTRAN.

Due to the complicate nature of the program, It is impossible to have a comprehensive study to all aspects of NASTRAN in a short period of time. In this research report several problems which employ some useful features of MSC/NASTRAN, suggested by engineering analysts at KSC, will be examined. The purpose of this work is to document test cases so that the capabilities of other versions of NASTRAN can be investigated. In the following sections the development history of NASTRAN and its functions will first be briefly described. Then the test problems will be illustrated. The last section contains discussions and conclusions.

II. NASTRAN DEVELOPMENT AND ITS FUNCTIONS

2.1 Introduction

Different from the current general purpose finite element software packages developed by individual companies and universities, NASTRAN (NASA STRucture ANalysis) was developed under NASA's sponsorship in 1964-1969. During that period NASA had a NASTRAN project management office which administered the software development, released and updated periodically the voluminous documentation, and reported any systematic error and correction procedure. The responsibility was shifted in 1969 to the Computer Software Management and Information Center (COSMIC) at the University of Georgia.

In using COSMIC/NASTRAN people often found that it was very difficult to make any correction when they realized some bugs in the program. Instead of trying to debug the program they reported the problems to the COSMIC which in turn gave subcontracts to individuals to solve the problems. The solutions to the problems might not be available until the release of the next version of COSMIC/NASTRAN.

Although only media costs were required in using COSMIC/NASTRAN, the ineffective software management and the lack of hot-line user's service made a number of finite element users shift to commercial software packages in early seventies. The most popular one inside (and/or outside) the United States is MSC/NASTRAN which was described in the previous section. The other versions are UAI/NASTRAN (offered by Universal Analytics Incorporation), CSA/NASTRAN (offered by Computerized Structural Analysis and Research Corporation.), PM/NASTRAN (offered by MARC Analysis Research Corporation), etc. In this section the general features of NASTRAN will be briefly discussed.

2.2 Program Architecture

The original NASTRAN was designed to meet certain criteria [2,3,4], such as: the ease of input, the minimization of user's intervention during the execution of the program, the independence of functions in solution modules, the restart capability, etc. Based on these goals NASTRAN is operated in the batch mode and is divided into four components which are given in order as follows [5]:

NASTRAN Card. This card is designed for changing the default values of certain operational parameters such as buffer size (keyword: BUFFSIZE), working space (keyword: HICORE), the amount of real memory available (keyword: REAL) and the numbers of lines printed per page (keyword: NLINES), etc. To get a solution of a large problem the first three parameters may need to be modified.

The Executive Control Deck. This deck is the heart of NASTRAN. It defines the type of solution and the general conditions to be performed, the allowed maximum execution

time, the types of system diagnostics required, the execution of DMAP alter library and the restart conditions. The general concept in designing this deck is the reliability and efficiency. Few changes were anticipated in the Executive system as NASTRAN grew. This leads to the result that the executive system is a sophisticated, but difficult to modify system.

The Case Control Deck. The function of this deck is to provide the selections of the data from the Bulk Data Deck, the subcase structure, the boundary conditions and loading cases, and the output requests for printing, punching and /or plotting.

Finally, the Bulk Data Deck provides all the data necessary to define the geometry of the structural model, the constraint and the loading conditions. It is the major portion of the input data for NASTRAN. The various pools of data which are selected by Case Control Cards will be used at execution time. For example, constraint conditions, loading conditions and thermal conditions may be specified to sets identified by number, but only those sets which are selected for a particular subcase in the Case Control Deck will be used in the solution.

2.3 Rigid Formats

A rigid format is an established sequence of DMAP instructions stored in the Executive system to perform some standardized analysis. It is usually identified with an integer number and an analysis approach. In COSMIC/NASTRAN there are 22 rigid formats [6] in which 16 rigid formats (1 to 16) in the displacement approach, 3 rigid formats (1, 3 and 9) in the heat transfer approach, and 3 rigid formats (9, 10, 11) in the aerodynamic approach. On the other hand there are more than 45 solution sequences in MSC/NASTRAN [7]. The rigid formats provide users with a very large range of analysis options. Section 3.1 of Reference 7 contains excellent tables which summarizes the various modeling options offered versus rigid body format number. For some particular problems, the user can also modify the rigid formats by DMAP alters.

2.4 Direct Matrix Abstraction Program (DMAP)

DMAP is a unique feature of NASTRAN and is one of the most powerful matrix manipulation tool offered in any code. The use of DMAP alters in rigid formats provides users with new analysis capability and increased efficiency. The DMAP is a program language of NASTRAN, which contains macro instructions of matrix operations. A DMAP has the following form:

(Module Name) (Input Data Block)/(Output Data Block)/(Parameter)\$

where a slash is a delimiter between data blocks and a dollar sign terminates the DMAP instruction. The sequence shown below gives as example [4].

To compute $[C] = [A] + [B]$ and $[D] = [A][C]$, the DMAP sequence is,
BEGIN\$ Start DMAP
ADD A,B/C\$ Adds [A] to [B]

MATPRN C//	Print [C]
MPYAD A,C/D/	Compute [D]
MATPRN D//	Print [D]
END\$	Terminate

To relieve users in preparation of matrix manipulation, NASTRAN provides in each new version a library of useful predefined DMAP alters which are called RFALTERS. The user has to insert these alters into the Executive Control Deck by using a machine-dependent operation system utility for merging files.

2.5 Checkpoint/Restart Capability

NASTRAN offers a sophisticated checkpoint/restart capability. The advantages of this feature are: (1) continuation of an execution of a job which terminated due to data errors or time limitation; (2) requesting additional information output for a job already completed; (3) restarting to run additional load cases in static analysis; and (4) extracting real eigenvalue for additional frequency ranges in normal mode analysis. In using this capability the NASTRAN card "CHKPNT YES" in the Executive Control Deck must be provided. This card makes the system save information on a file called NPTP (New Problem Tape) which will be renamed as OPTP (Old Problem Tape) in the future run.

2.6 Nonlinear Capabilities

Nonlinear analysis should be performed whenever problems involve nonlinear materials and finite deformation. The nonlinear capabilities are different among the various versions of NASTRAN and are all primitive when compared with those offered by current nonlinear codes such as ANSYS and ABAQUS. There are two types of static nonlinearities offered in COSMIC/NASTRAN, i.e., the differential stiffness of rigid format 4 and the material plasticity of rigid format 6.

Nonlinear capabilities have been improved in the latest version (66) of MSC/NASTRAN [7]. The static nonlinear solution methods, SOLs 64 and 66, provide for modeling inelastic effects such as material plasticity and creep, "gaps" in the structure which open, close, and slide (if the friction force is smaller than the product of the normal force and the friction coefficient), and nonlinear elastic material properties. Furthermore the rigid format 99 offers transient dynamic solutions for material nonlinearity and large deformation. The nonlinear capabilities in the UAI/NASTRAN is said to be comparable to those offered in MSC/NASTRAN. This argument can not be judged because of unsuccessful access of the UAI/NASTRAN User's Manual.

2.7 Superelement

A structure is divided into pieces called substructures or superelements. The automated multi-stage substructuring system for NASTRAN [8], developed by Universal Analytics, Inc., is very useful and user oriented. The reasons for subdividing a structure into superelements are that: (1) the reduction in the computational cost when design changes

are of localized nature because the design changes for single substructure may not affect matrices of all other substructures; (2) the simplification in data generation and in checkout; (3) the efficiency in computation to form the condensed stiffness matrix of the typical substructure because those internal degree of freedom need to be processed only once; and (4) the different design groups can study different substructures simultaneously.

In using this feature the user should keep in mind that the boundaries of substructures must be small, so that the ratio of masters (the retained degree of freedom) to slaves (the condensed degree of freedom) is small and the overall stiffness matrix can be kept in manageable size. Accordingly, a long cylinder can be more efficiently substructured than a sphere [9]. An excellent discussion of superelements can be found in Reference 10.

2.8 Cyclic Symmetry

For a structure which can be generated through rotation of a superelement (fundamental region), its circumstance is called cyclic symmetry. A structure such as an propeller in a centrifugal pump is a good example. The analytical model required for cyclic symmetry analysis, in general, is substantially smaller. In fact, the cyclic symmetry analysis can be treated as a special form of superelement analysis. The difference between these two techniques are that the cyclic symmetry analysis deals with problems having all identical superelements while the superelement analysis involves the subdivision of a structure into superelements of arbitrary shapes. The cyclic symmetry analysis offered in NASTRAN can automatically impose boundary conditions and also deals with the applied loads which may act at arbitrary locations throughout the entire structure.

III. TEST PROBLEMS

3.1 Introduction

Even with a noncomprehensive discussion of the features offered in NASTRAN, as shown in the previous section, one will be very impressed with the complexity of a problem which can be handled by NASTRAN. In this section three real world problems, which have been solved by MSC/NASTRAN and verified by experiments, will serve as test cases for other versions of NASTRAN. The problems are selected so that a number of different type elements, the medium-to-large size of problems, and transient or nonlinear analysis may be included. Since the documentation and source codes of the other NASTRAN versions are not available at KSC, the capabilities of those software packages in handling the three problems are not studied. Due to the lengthy output of the results, the solution printout to these problems won't be shown in this report, however it can be found from the VAX/VMS computer at KSC.

Before we start to discuss test cases, it may be interesting to know some problems encountered on the interface between the latest versions of pre- (post-) processing softwares and NASTRAN. Here, we will briefly discuss two interactive engineering modelling and results interpretation tools: MSC/XL (version 1A) and SDRC/I-DEAS (version 6).

MSC/XL runs on DEC VAX/VMS processors with Tektronix (4111, 4129 and 42xx) or X windows version 11 from the VAX workstation. MSC/XL has made MSC/NASTRAN be more powerful and attractive, however the current version of MSC/XL cannot generate picture files. Therefore the quality of hard-copy plots can be very poor, nor be in color if the Tektronix or VAX workstation are used. The color plots for displacements and stress fields are the important means for clearly presenting analysis results. Its discrepancy can be overcome if one uses Intergraph workstation as a graphic device.

On the other hand the use of SDRC/I-DEAS is more involved than that of MSC/XL. Once a model is developed in I-DEAS, the command sequence shown in any task: Mange_Model, Write, MSC/NASTRAN, can generate Case Control Deck and Bulk Data Deck. To have the computational results be able to transfer back to I-DEAS, the user need to use functional module OUTPUT2 in the Executive Control Deck to store all the data blocks. In using MSC/NASTRAN version 65, those data blocks must be specified and then written into a FORTRAN unit which in turn assigned to a file [1]. This procedure has been improved in version 66 which contains certain RFALTERS supporting rigid formats 3, 5, 24, 26, 27, 47, 48, 61, 63 and 66. As an example, the Executive Control Deck for nonlinear material analysis [12] is shown below:

```
ASSIGN OUTPUT2=out.OP2,NEW,UNIT=12,UNFORMATTED
ID MATERIAL NONLINEAR
SOL 66
TIME 20
COMPILE SOL66, SOLIN=MSCSOU
RFALTER RF66D66
CEND
```

This deck together with an additional "Param, Post, -2" card in the Bulk Data Deck will make all the resulting data block be stored in out.OP2 which can be accessed by I-DEAS through the command "Data Loader" in the post-processing.

Although I-DEAS version VI has been claimed to be much better than its previous versions, the functional module, Data Loader, in the current version does not work well. In one of the three test problems - Rolling Beam Umbilical System, both geometry data and analysis data can be loaded to I-DEAS when version V is used, however none of the data can be processed if version VI is used.

3.2 Rolling Beam Umbilical System

A rolling beam retract mechanism in the primary disconnect of the liquid hydrogen (LH₂) umbilical carrier plate was modeled by means of MSC/NASTRAN [12]. The dynamic response was performed through the direct transient analysis (SOL 27). Figure 1 shows the isometric view of the structure. The model includes 786 one-dimensional elements, 237 two-dimensional elements, 8 one-dimensional elbow elements, 121 spring elements and one general element. The rolling beam motion is initiated by firing a pyro bolt. The dead-load, time dependent forcing function was applied to the rolling beam in vertical direction (Z_R), while the time dependent displacement function was applied to the LH₂ umbilical carrier plate in the vertical direction (X_O) and horizontal direction (Y_O). The MSC/NASTRAN input deck and output files for this problem can be accessed from the filename "Roll" of the directory [LU] in KSCDL1. The typical displacement, velocity and acceleration histories are respectively shown in Figures 2, 3 and 4.

3.3 KSC SRB Hold-down Post

The location of the solid rocket booster (SRB) hold-down post on the launch pad is shown in Figure 5. During the shuttle launch the ignition of the main engine starts at t=-7 seconds, while that of the SRB starts at t=0. There will be some forces built up on the SRB hold-down post due to the small oscillation of the whole system. The maximum values of the loading applied to the bearing of the hold-down post were given as 1457 kip in compression and 500 kip in shear. Nonlinear analysis sequence (SOL 66) have been used in the study. The structure model is shown in Figure 5 containing 3449 nodes, 319 GAP elements, 1633 HEXA elements and 3264 TETRA elements. Figure 6 shows the deformed shape of the SRB hold-down post (in solid line) superimposed with the undeformed configuration. The displacement and the stress fields are respectively shown in Figures 7 and 8. The details of the input deck and outputs can be found from the file "SRBHP" of the directory [LU] in KSCDL1.

3.4 Single Pallet Rotation Device Cover Rivet Analysis

A single pallet rotation device (SPRD) panel lifting lug with extra rivets was studied. The finite element model which includes 349 BAR elements, 319 BEAM elements, 1008 HEXA elements, 1153 QUAD4 elements and 27 TRIA3 elements. The total number of the nodes is 3944. Static analysis (SOL 101) was performed and the results for the deformed shapes and stress fields are shown in Figures 9, 10 and 11. The MSC/NASTRAN input deck and output for this problem are stored in the files "SPRD" of the directory of [LU] of KSCDL1.

IV. DISCUSSIONS AND CONCLUSIONS

Although NASTRAN has been reported too costly for running small to medium-size problems, it is generally accepted that NASTRAN solves medium to large structures as well or better than most general purpose program [4]. For the usage of finite element codes, NASTRAN has been reported to be the most widely used in the world and may have the most complete documentation.

In this report some of the important features of NASTRAN have been summarized and three cases were documented for the testing of the different versions of NASTRAN. The test problems, including direct transient analysis, nonlinear analysis and static analysis, were modeled by means of MSC/NASTRAN. Due to unsuccessful access to the software package of the other commercial versions of NASTRAN, their capabilities are not studied.

For the use of the highly graphics software packages in the pre- and post-processing phases, it has been found that MSC/XL cannot generate picture files for producing good quality plots nor support the finite element codes other than MSC/NASTRAN. On the other hand, SDRC/I-DEAS provides more capabilities to interface a number of finite element packages and can produce high quality plots. However, there may exist errors in the functional module "Data Loader" of the current version (VI).

In the course of surveying finite element users at KSC, one has commented good capabilities of UAI/NASTRAN in handling direct transient problems such as the first test case. It would be beneficial to KSC to have future study to investigate the capabilities of the other latest general purpose finite element packages as long as the reduced cost in the replacement versus the cost of MSC/NASTRAN is substantial.

REFERENCES

1. Lu, C.H., "Study of The Available Finite Element Software Packages at KSC," Research Report for 1990 NASA/ASEE Summer Faculty Fellowship Program at Kennedy Space Center, Contract Number NASA-NGT-6002 Supplement:4.
2. Mueller, G.E., "The Future of Data Processing in Aerospace," *Aeronautical Journal*, April 1979, PP 149-158.
3. "Trends in Computing: Applications for the 80's," *Fortune*, May 19, 1980, pp. 29-70.
4. Jones, J.W., Fong, H.H. and Blehm, D.A., "Evaluation of NASTRAN General Purpose computer Program," Final Report to The Office of Naval Research, Code 474, August, 1980.
5. MSC/NASTRAN Handbook for Linear Analysis, MSC/NASTRAN Version 64, The MacNeal-Schwendler Corporation, August, 1985.
6. COSMIC/NASTRAN User's Manual Updates, Section 5.5-43, 1987-1990.
7. MSC/NASTRAN User's Manual, Version 66, The MacNeal-Schwendler Corporation, November, 1989.
8. Field, E.I., Herting, D.N., Herendeen, D.L. and Hoesly, R.L., "The Automater Multi-stage Substructuring System for NASTRAN," NASTRAN: User's Experiences, NASA TM X-3278, Fouth NASTRAN User's Colloquium, NASA Langley Research Center, 9-11 September, 1975.
9. Cook, R.D., Malkus, D.S. and Plesha, M.E., *Concepts and Applications of Finite Element Analysis*, Third Edition, John Wiley & Sons, 1989.
10. MSC/NASTRAN Handbook for Superelement Analysis, MSC/NASTRAN Version 61, The MacNeal-Schwendler Corporation, April, 1982.
11. SDRC/I-DEAS Finite Element Modeling User's Guide, Level VI, SDRC, 1990.
12. Meyer, K., "Dynamic Response and stress analyses of the STS/Centaur Rolling Beam Umbilical System Rolling Beam," KSC-DD-803-TR, John F. Kennedy Space Center, NASA, June 1985

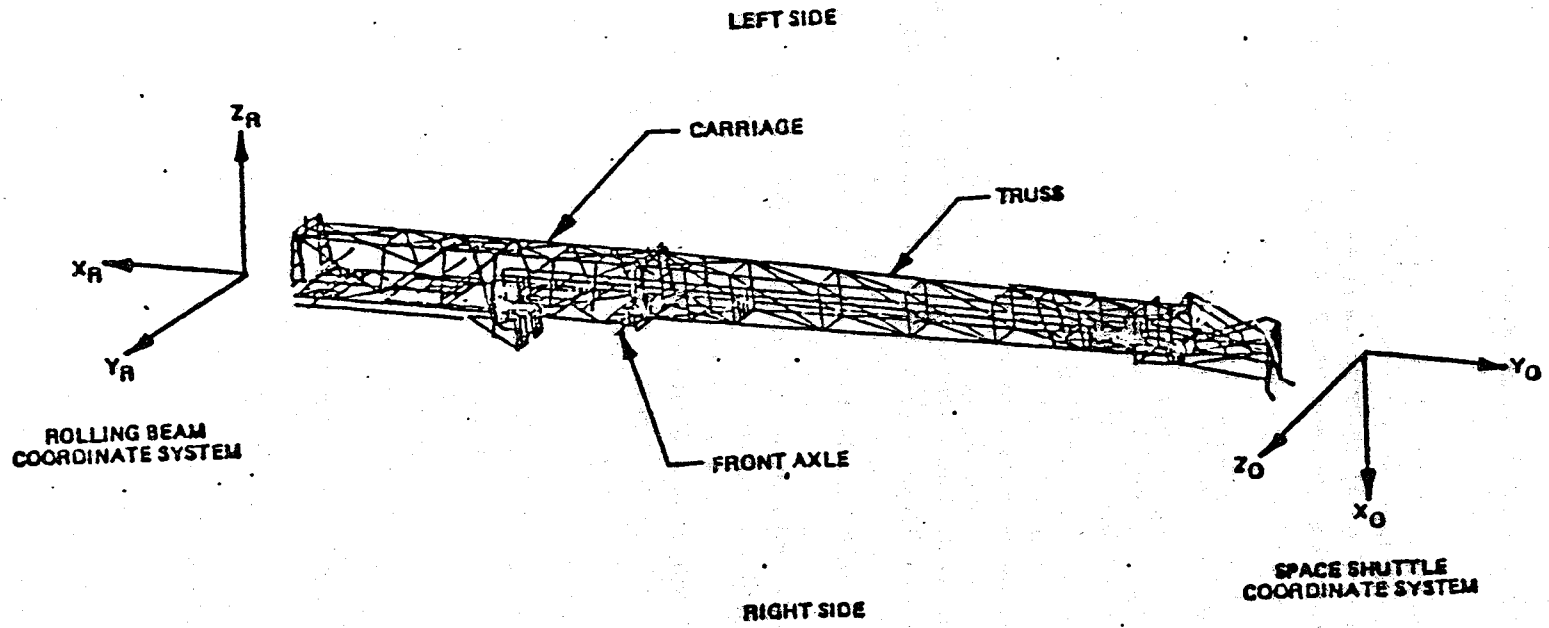


Figure 1 An Isometric View of Rolling Beam Umbilical System

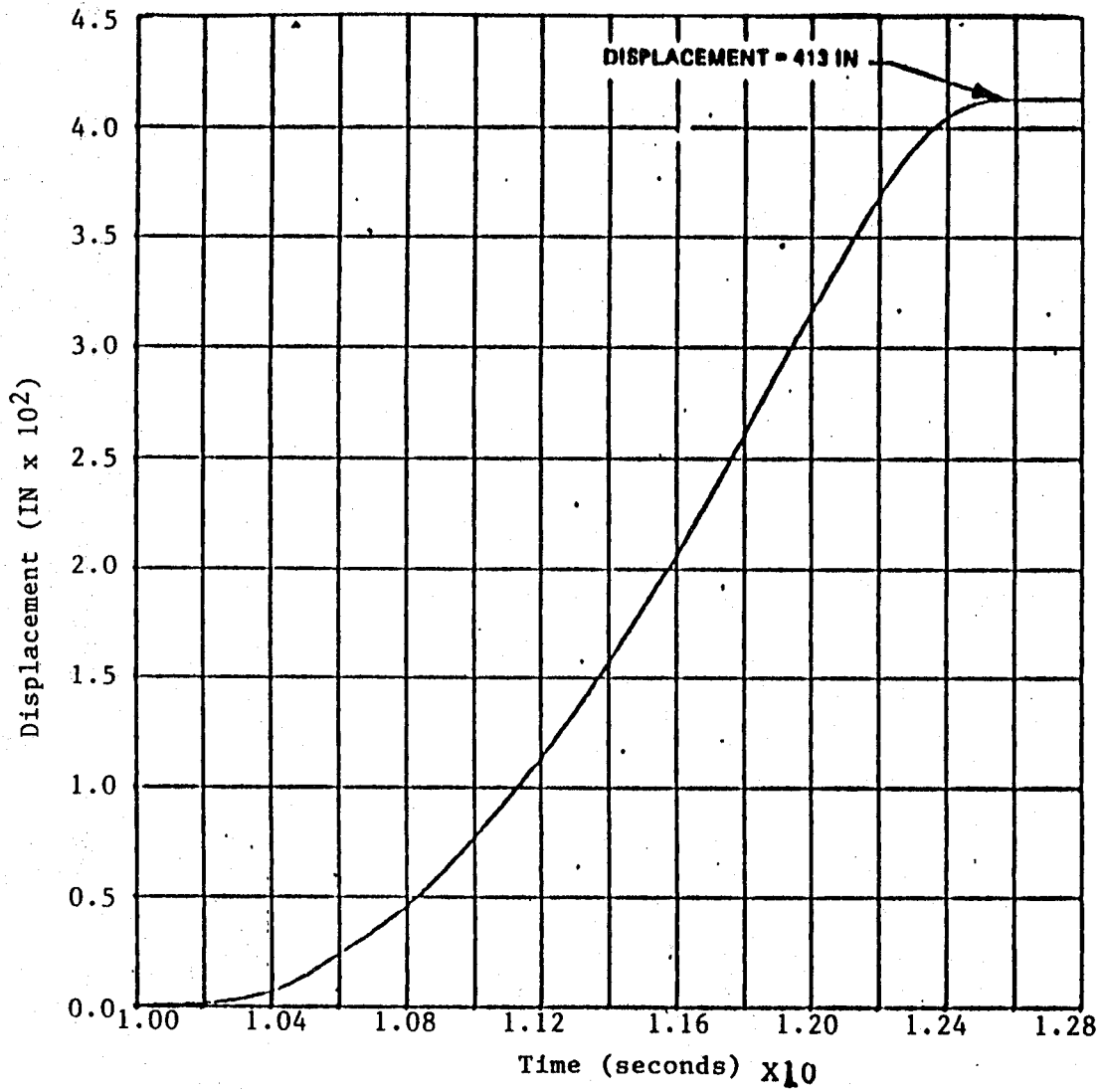


Figure 2. X_R Displacement History at Carriage/Truss Interface

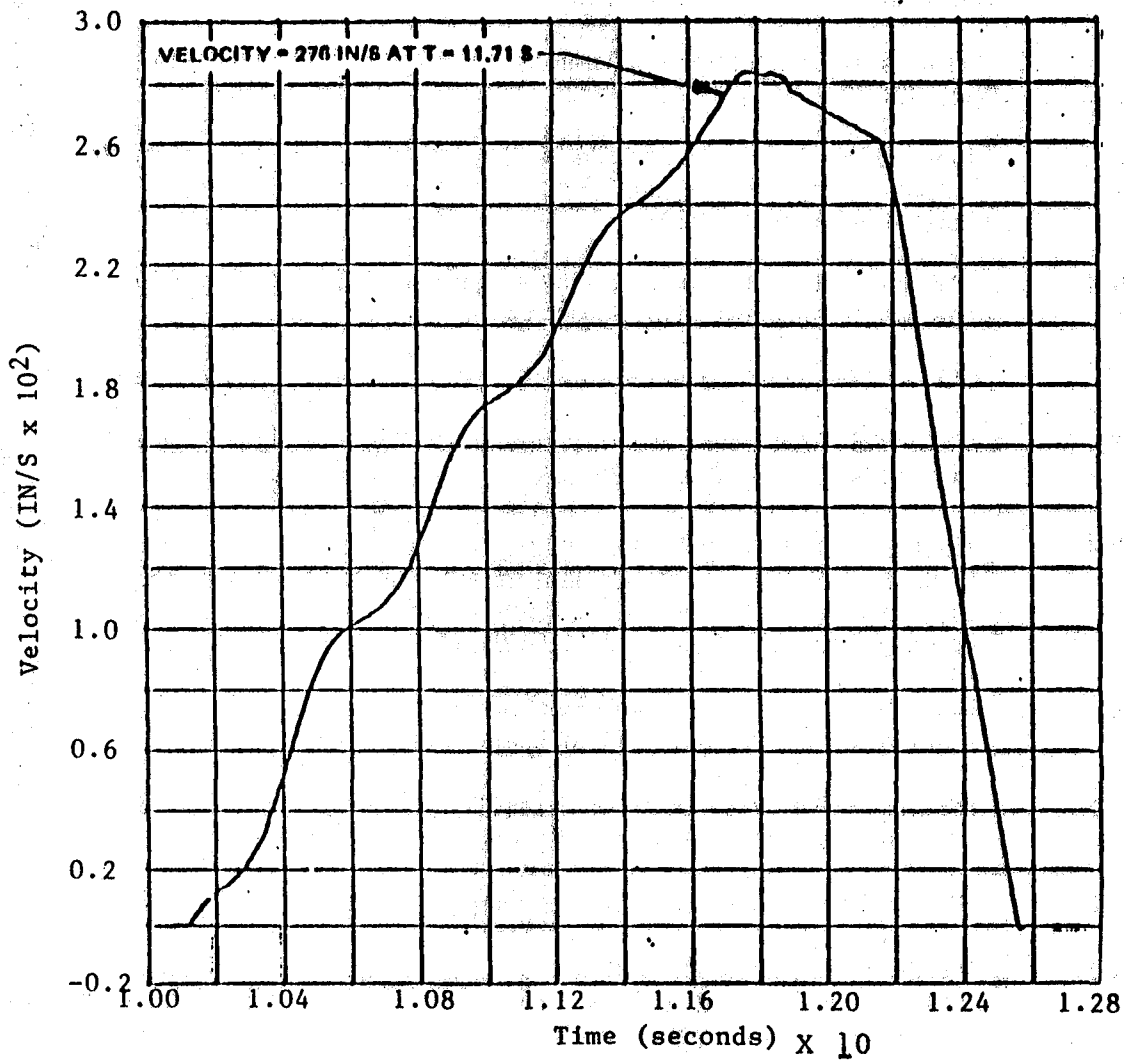


Figure 3. X_R Velocity History at Carriage/Truss Interface

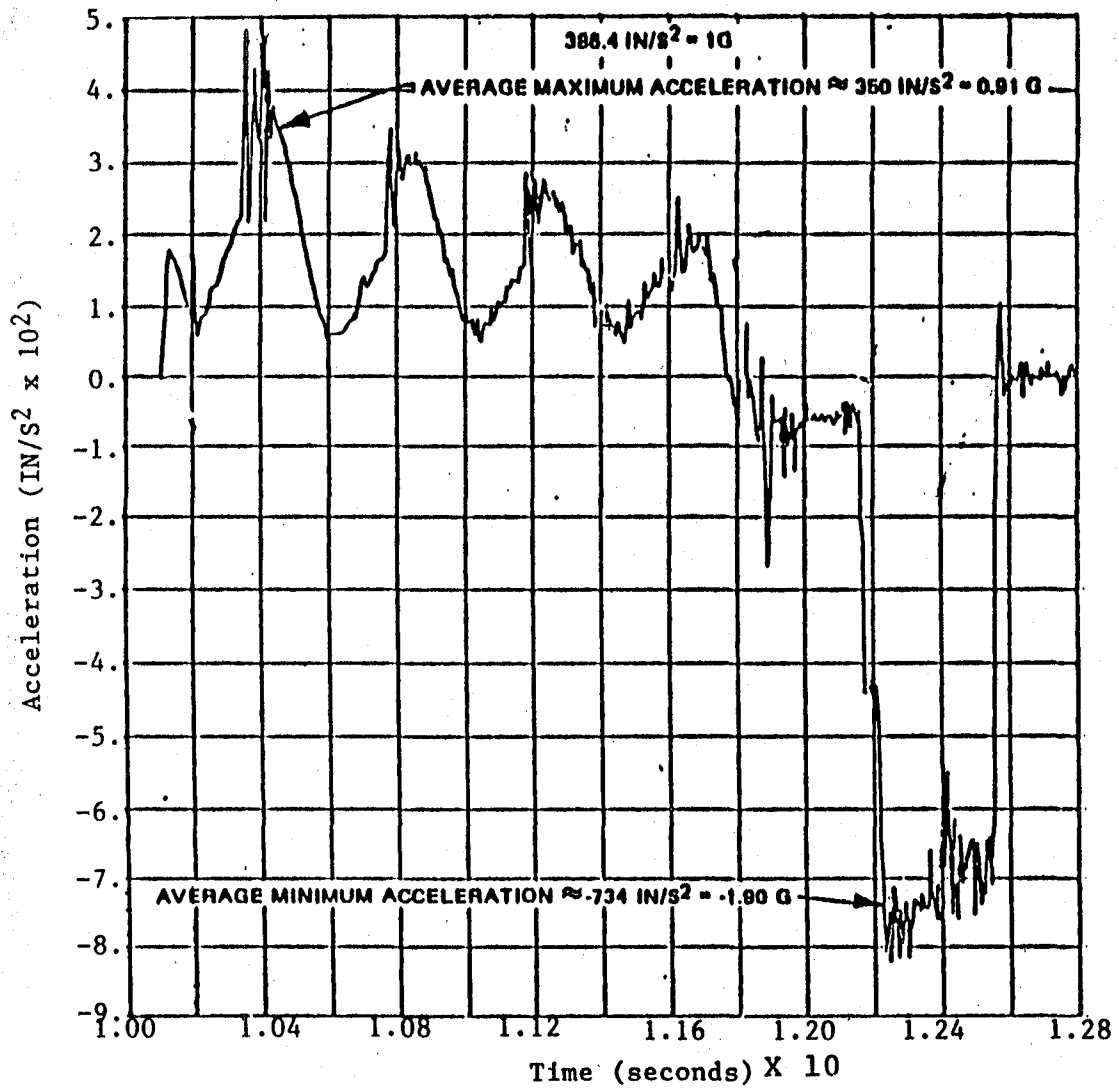


Figure 4. X_R Acceleration History at Carriage/Truss Interface

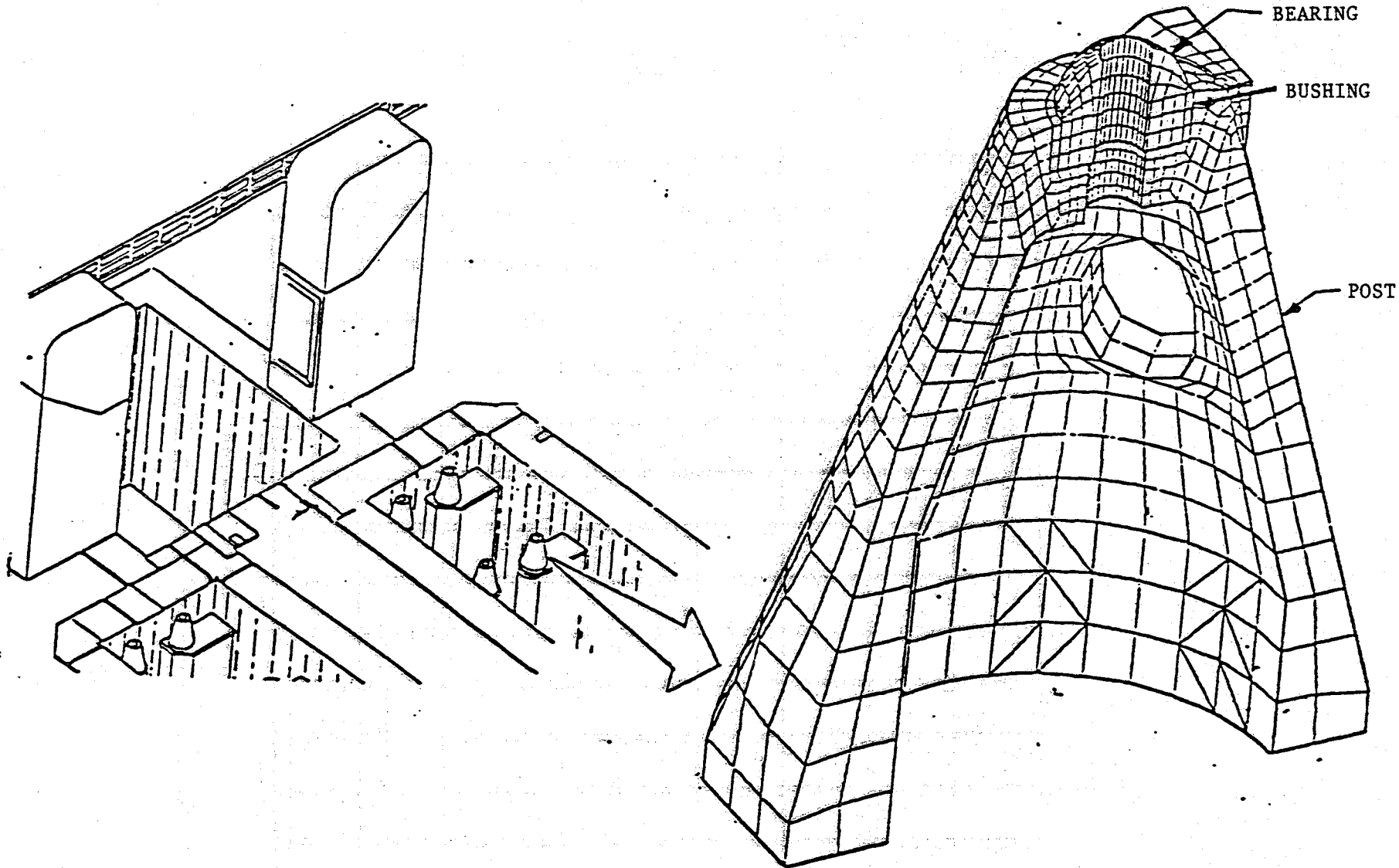
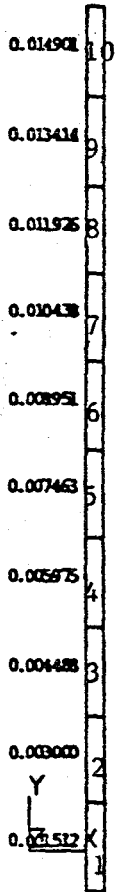
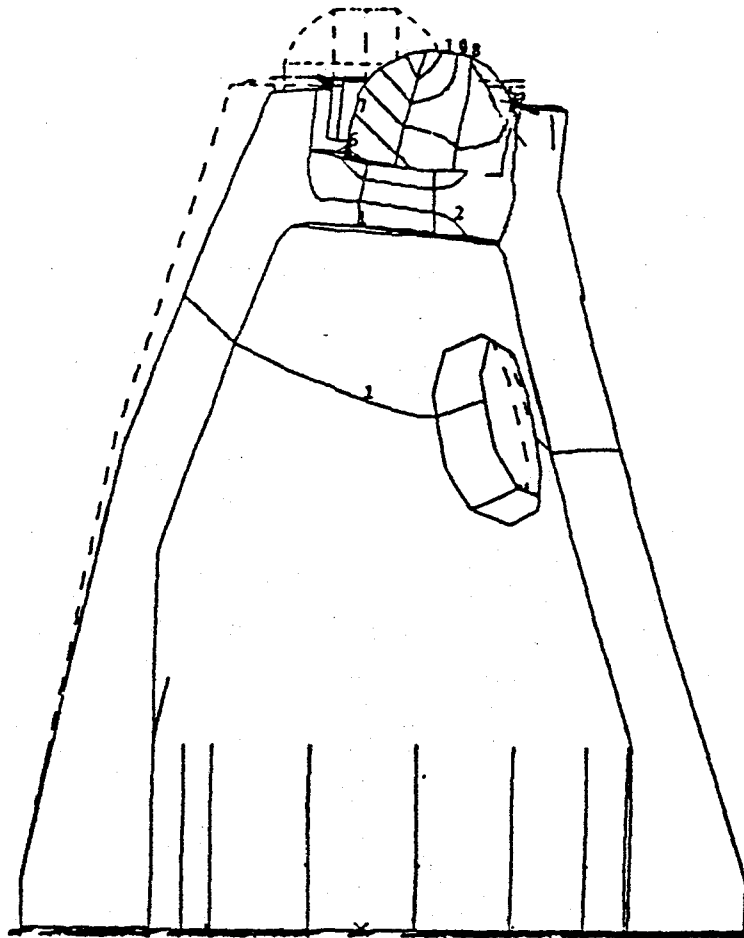


Figure 5. The Location and Finite Element Model of SRB Hold-down Post

ID NUMBER: 1
FRAME OF REF: GLOBAL
DISPLACEMENT - MPG MIN: 2.47E-05 MPX: 0.016389



230

Figure 6. The Deformed Configuration of SRB Hold-down Post

ID NUMBER: 2
FRAME OF REF: GLOBAL
STRESS - VON MISES MIN: 137.60 MAX: 6389.34

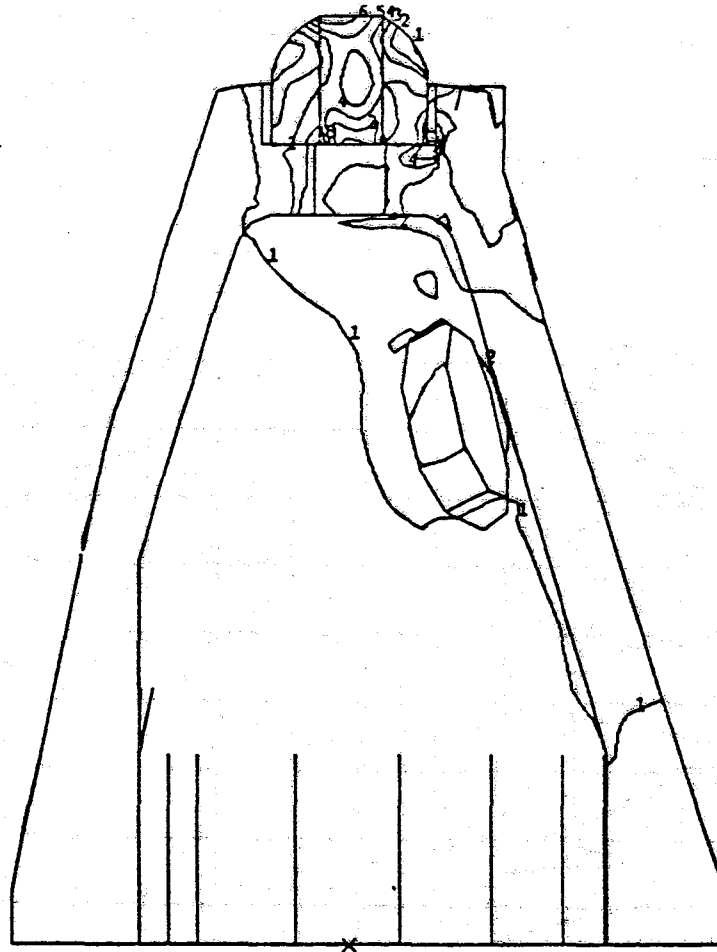


Figure 7. The Stress Field of The SRB Hold-down Post

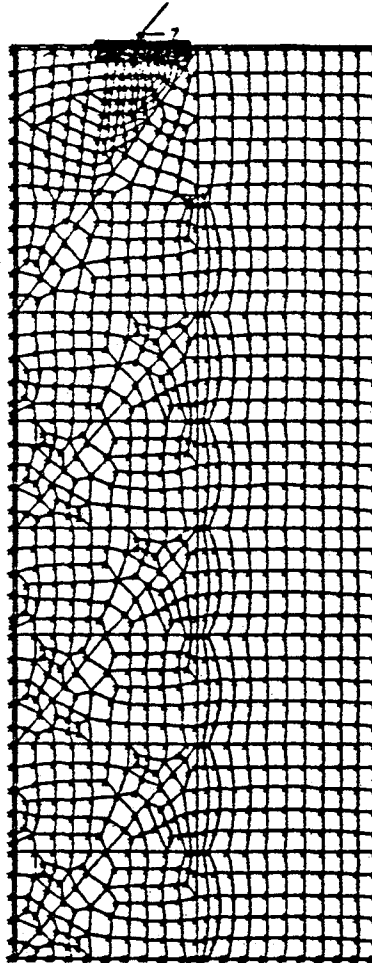


Figure 8. The Finite Element Model of A Single Pallet Rotation Device.

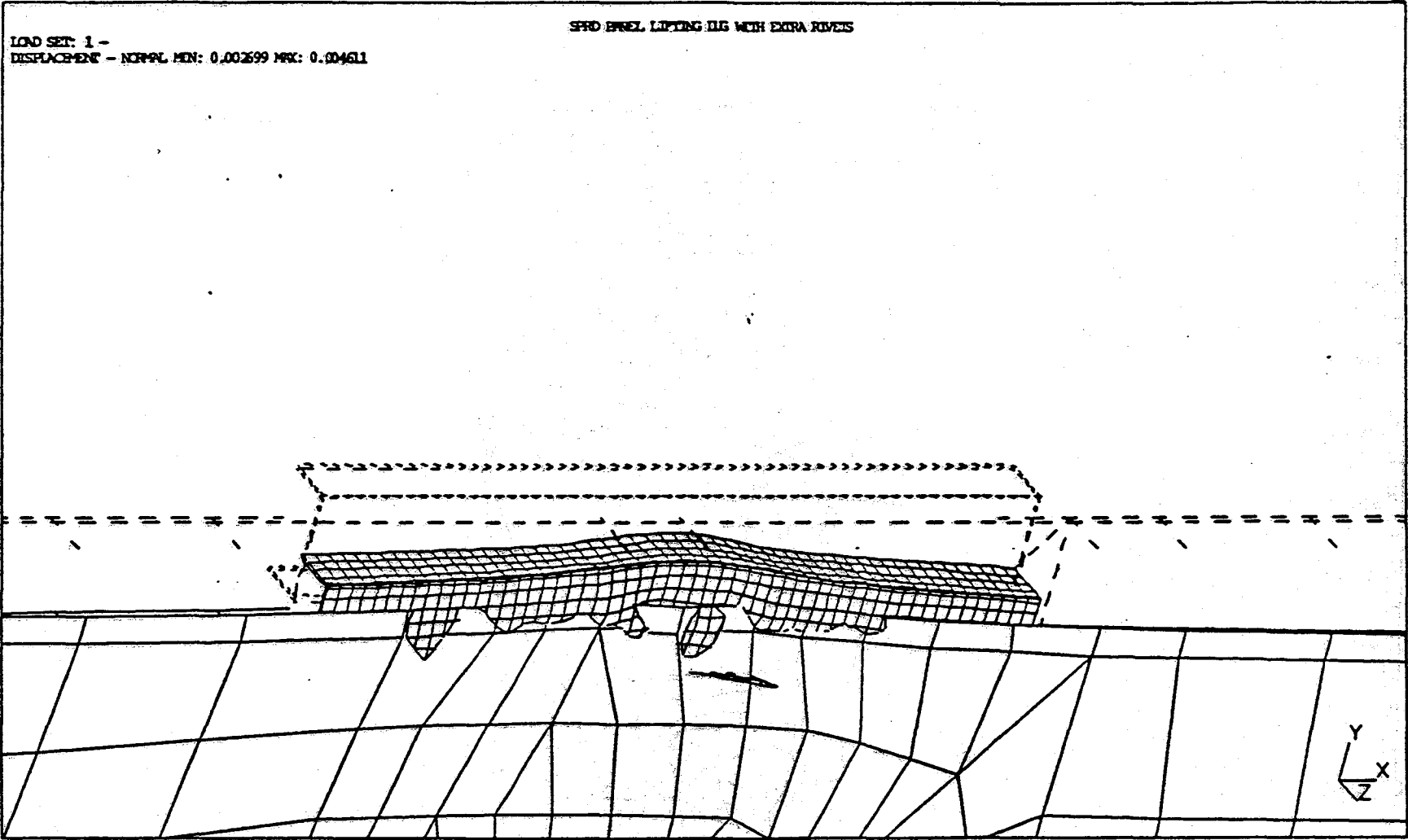


Figure 9. The Deformed Shape of The Single Pallet Rotation Device

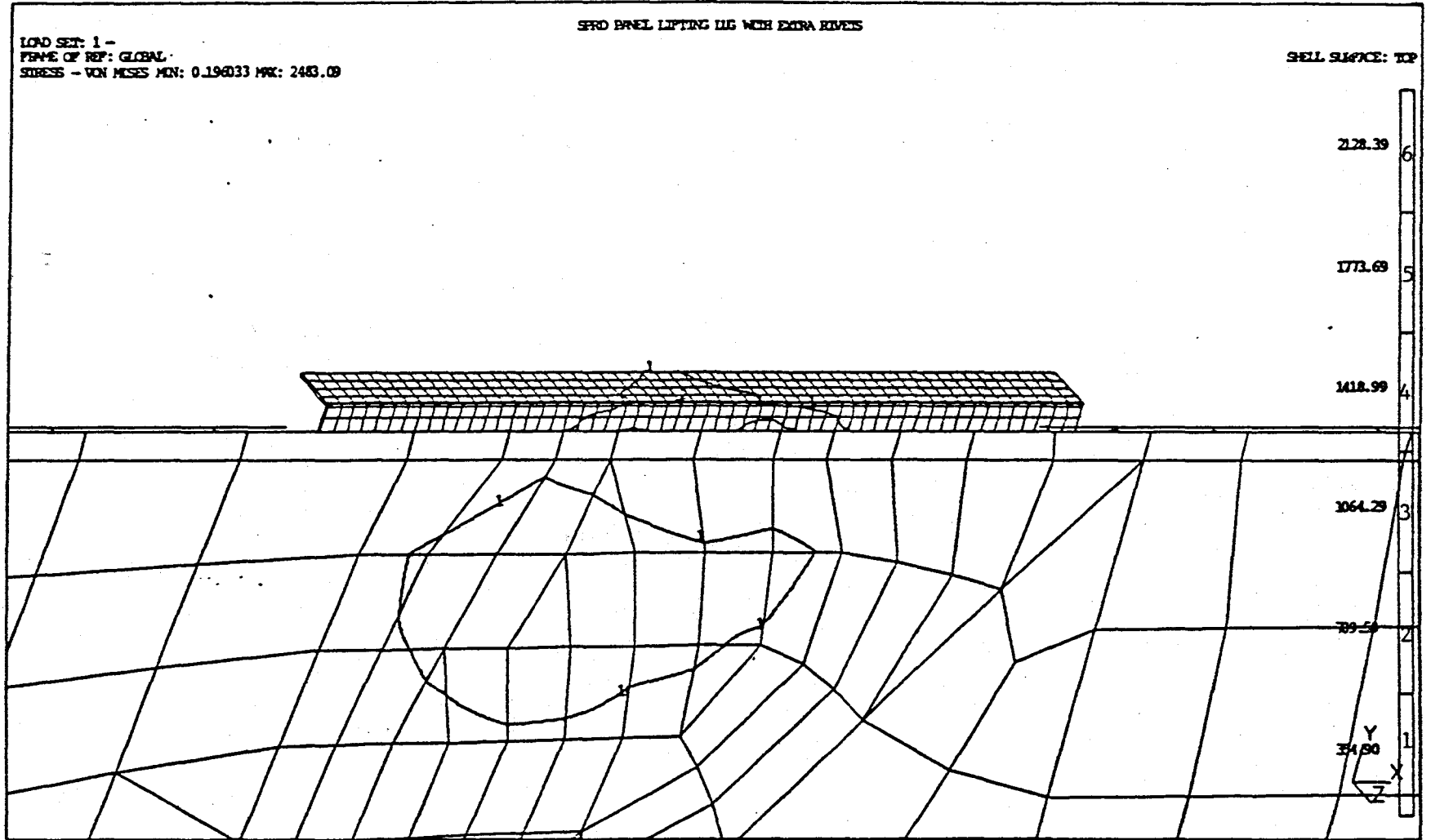


Figure 10. The Stress Field of The Single Pallet Rotation Device

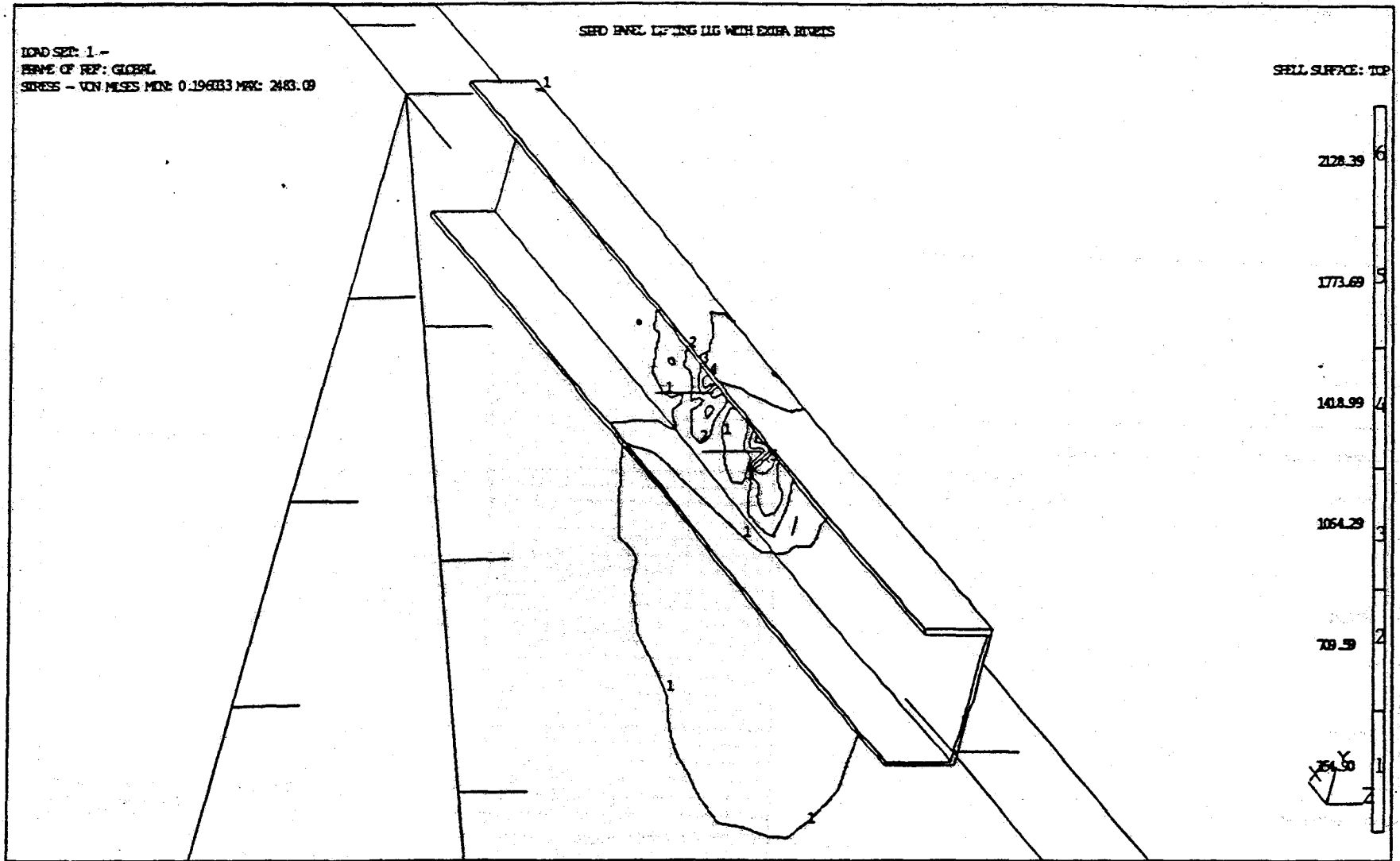


Figure 11. A Different Angle View for The Stress Field of The Single Pallet Rotation Device