N91-20029 p. 25

1990 NASA/ASEE SUMMER FACULTY FELLOWSHIP PROGRAM

JOHN F. KENNEDY SPACE CENTER
UNIVERSITY OF CENTRAL FLORIDA

STUDY OF THE AVAILABLE 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. Ed Bertot

DATE: August 10, 1990

CONTRACT NUMBER: University of Central Florida
NASA-NGT-60002 Supplement: 4

ACKNOWLEDGMENTS

I would like to thank my NASA colleagues, Mr. Eddie Bertot and Mr. Hank Perkins of DL-DSD-22 for their support in obtaining the necessary resources to do this research. The technical support from Mr. Drew Hope of EG&G in using I/FEM, and Mr. Rudy Werlink and Mr. Raoul Caimi of NASA DM-MED-11 of and Mr. Richard Hall and Mr. Lloyd Albright of EG&G in using SDRC/I-DEAS and MSC/NASTRAN was also deeply appreciated.

Additionally, I would like to thank the administrative support of Dr. Loren A. Anderson and Ms. Kari Baird, University of Central Florida, and Dr. Mark Beymer, the director of the summer faculty fellowship program at NASA/KSC.

ABSTRACT

This research report concerns the interaction among the three finite element software packages - SDRC/I-DEAS, MSC/NASTRAN and I/FEM, used at NASA, John F. Kennedy Space Center. The procedures of using more than one of these application software packages to model and analysis a structure design are discussed. Design and stress analysis of a solid rocket booster fixture is illustrated by using four different combinations of the three software packages. Their results are compared and show small yet acceptable differences.

SUMMARY

In this research report the use of the three finite element software packages, SDRC/I-DEAS, MSC/NASTRAN and I/FEM, in model construction and statics analysis is studied. The procedures in using highly interactive and graphics software such as SDRC/I-DEAS and I/FEM to construct a working model are briefly summarized in Section II and illustrated in Figures 1, 2 and 3.

Due to the very general capabilities of structural analysis contained within MSC/NASTRAN, the applicability of using SDRC/I-DEAS (or I/FEM) in preprocessing and postprocessing and MSC/NASTRAN in analysis are discussed in Sections III and IV. Although transferring files between SDRC/I-DEAS and I/FEM is not supported by either software, it is found that it can be done with the aid of MSC/NASTRAN.

The design and statics analysis of a solid rocket booster fixture was studied in Section V. The purpose of studying this problem is twofold: the strength of the fixture so that it won't fail in use can be assured through stress analysis, and the interfacing of more than one software packages can be tested via four different combinations as shown in Table 1. As expected, the different combinations of the three finite element software packages yielded similar results. The effective use of these software packages and the strengths and weaknesses of each software package are discussed in the conclusions.

TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

LIST OF TABLES

I. INTRODUCTION

The finite element method has been established as a powerful and popular numerical procedure for solving many different problems of continua governed by differential equations. The method, in general, can be considered a definite set of seven basic steps: Discretization, Interpolation, Elemental Formulation, Assembly, Constraints, Solution and Computation of derived variables. As a result of the first three steps, 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.

It is possible to write computer code which will create and analyze a finite element model just described. 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.

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 model analysis process. It is clear that the effort in this process can be reduced if a third-party analysis package, with powerful pre- and post-processing capabilities can be used. The purpose of this project is therefore devoted to studying the performance of the interaction among the available finite element software packages at NASA, John F. Kennedy Space Center. In the following sections the procedures combining the powerful pre- and post-processing capabilities of SDRC/I-DEAS and I/FEM with MSC/NASTRAN's analysis capabilities will be summarized and the design of a solid rocket booster fixture will be examined. The last section contains discussions and conclusions.

II. FINITE ELEMENT MODEL DEVELOPMENT

The procedures to creating a finite element model can be very different depending on which application software package is used. However the basic concepts may be the same. The procedures in the model construction process for the available finite element software packages at NASA-KSC: SDRC/I-DEAS (version 4.1), MSC/NASTRAN (version 65) and I/FEM (version 1.3), will be briefly summarized in this section. It will be clear that the I-DEAS and I/FEM software packages are easier to use and save time when compared to NASTRAN.

2.1 SDRC/I-DEAS

I-DEAS (Integrated Design Engineering Analysis Software) developed by Structural Dynamics Research Corporation provides a comprehensive package for mechanical design engineers. Its capabilities are packaged as a set of software modules in which the I-DEAS Supertab modules offer finite element applications. The software provides highly interactive, graphic-oriented and menu-driven modules. Figures 1 and 2 show I-DEAS Supertab modules and Pre/Post processing module [1], respectively.

The geometry and elements of a finite element model can be constructed in the Model Preparation module for simple structures (e.g. truss). This module offers creation, generation, and manipulation of nodes, elements, coordinate systems, physical tables and material tables. For more complicated structures one can first use the Free Mesh Geometry module or the Geometry Definition module to prepare and manipulate the geometry. The element mesh can then be generated automatically by using Free Mesh Generation module or generated semi-automatically by using the Mapped Mesh Generation module. The mesh just created may have different sizes so that high mesh density can be developed in critical regions. The validity of the model is verified in the Model Checking module. The preprocessing phase is finally done by specifying the boundary conditions and loads of the model in the Analysis Cases module.

2.2 MSC/NASTRAN

The software package NASTRAN (NASA Structural Analysis) which was developed by NASA has been expanded to be a large-scale general purpose structural analysis package and is marketed by The MacNeal-Schwendler Corporation since 1972. MSC/NASTRAN is designed to operate in the batch mode. As such, a job submitted resembles a card deck stacked in the following order: NASTRAN Card, Executive Control Deck, Case Control Deck and Bulk Data Deck.

Similar to the preprocessing phase in I-DEAS, the Bulk Data Deck deals with structural modeling. The deck contains all the data necessary to define the geometry, and the constraints and loading conditions. It is the major portion of the input data for MSC/NASTRAN. For a large problem the Bulk Data Deck may consist of several thousand cards [2].

In preparing the Bulk Data Deck engineers usually record the data from the model's sketch. Without any typing mistakes, the time spent in this preparation is approximately twice as long as one would spend by using I-DEAS.

2.3 I/FEM

The Intergraph Finite Element Modeling System (I/FEM) is a computer-aided engineering software package for finite element analysis. The software operates with highly interactive graphics on Intergraph's workstation. The analyst creates the model by first selecting commands from icon-based graphic menus and then by drawing the geometry on the screen in one of the four windows which provides the top view, front view, right view and isometric view (Figure 3).

Like I-DEAS, two methods of building a finite element mesh are available in I/FEM. In using automatic meshing the analyst is restricted to 3-node or 6-node triangles for plane elements or 4-node tetrahedrons for solid elements. With semiautomatic mapped meshing any type of finite element can be used. However the mapped meshing requires more user interactions and therefore is more time consuming than automatic meshing.

One additional attractive feature in I/FEM is the geometry-based modeling [3] which enables users to define boundary conditions (constraints, loads, temperatures, etc.,) on the design geometry so that users do not need to modify the automatically generated mesh to assure accurate placement of the boundary conditions.

III. ANALYSIS AND USING THE INTERFACE

After the completion of the model construction process in I-DEAS or I/FEM, the analyst can do the analysis on third-party software packages or use their own analysis capabilities which are relatively small when compared to NASTRAN's capabilities. In this section the interface among I-DEAS, NASTRAN and I/FEM will be discussed.

3.1 The Interface Between SDRC/I-DEAS and MSC/NASTRAN

To create an analysis input file for MSC/NASTRAN in I-DEAS, we use the following command sequence which is available in any task: MAnage_fe_model, Write, Nastran_(msc). A file containing the Case Control Deck and the Bulk Data Deck is then generated. With the addition of an Executive Control Deck, which requests a specific solver, and an OUTPUT2 file storing all the data blocks generated via the Case Control Deck, the job can be submitted in NASTRAN. If the data block is not requested via Case Control, the OUTPUT2 functional module will ignore it. As an example, the Executive Control Deck for statics analysis [1] is shown below:

```
$    THE FOLLOWING DATA BLOCKS ARE RECOVERED:
$
$         CSTM     - COORIDNATE SYSTEM TRANSFORMATION MATRICES
$         GPL      - GRID POINT LIST
$         GPDT     - GRID POINT DEFINITION TABLE
$         EPT      - PHYSICAL PROPERTY DEFINITIONS
$         MPT      - MATERIAL PROPERTY DEFINITIONS
$         GEOM2    - ELEMENT DEFINITIONS
$         GEOM3    - LOAD DEFINITIONS
$         GEOM4    - RESTRAINT DEFINITIONS
$         OUGV1    - GRID POINT DISPLACEMENTS
$         OSTR1    - ELEMENTAL STRAIN
$         OES1     - ELEMENTAL STRESS
$         OEF1     - ELEMENT FORCES (STRESS RESULTANTS)
$         ONRGY1   - ELEMENTAL STRAIN ENERGY
$
ID STATICS, EXAMPLE
TIME 10
SOL 24
READ 9
ALTER    210 $
OUTPUT2  CSTM,GPL,GPDT,EPT,MPT//0/11 $
OUTPUT2  GEOM2,GEOM3,GEOM4,,//0/11 $
OUTPUT2  OUGV1,OSTR1,OES1,OEF1,ONRGY1//0/11 $
OUTPUT2  ,,,,//-9/11 $
CEND
```

In the above example the OUTPUT2 file is written to FORTRAN unit 11. Usually a .com file which contains a .dat file assigned to FORTRAN unit 11 (e.g. $ASSIGN OUTPUT.DAT FOR011) must be submitted along with the NASTRAN job so that the OUTPUT2 file can be automatically placed in the .dat file.

Using the I-DEAS Supertab Data Loader module the analyst can recover the analysis results in the NASTRAN OUTPUT2 binary file and create a universal file which in turn be read by the pre/post processing module into a model file. Once a model file has analysis result data in it, the postprocessing phase is ready to be studied.

3.2 The Interface Between I/FEM and MSC/NASTRAN

Translation of an I/FEM design model to an MSC/NASTRAN bulk data deck is done in two steps: first by translating the model to a neutral file (an ASCII file) and then by running the following translator:

/usr/ip32/msc/bin/ifmsc [-l] [-s] <i_fn> <o_fn>

where -l and -s stand for the long format and the short format for the bulk data deck,respectively. The <i_fn> is the name of the neutral file and <o_fn> is the name of the resultant bulk data deck.

To run the job the analyst must add the MSC/NASTRAN executive control deck and case control deck. The request of an MSC punched output file is necessary so that the results can be transferred back to I/FEM. For running a statics problem, the executive control deck and case control deck may have the following form:

```
ID STATICS, EXAMPLE
TIME 10
SOL 24
CEND
DISPLACEMENT (PUNCH) = ALL
STRESS (PUNCH) = ALL
```

Because the punched files are written to FORTRAN unit 7, the NASTRAN job should be submitted along with a .com file which assigns a .pch file to FORTRAN unit 7 (e.g. $ASSIGN OUTPUT.PCH FOR007) so that all the output data is stored in the .pch file.

To load the MSC .pch file into I/FEM model, first run the following loader:

/usr/ip32/msc/bin/ilmsc <i_fn> <o_fn>

where <i_fn> is the name of the MSC punch file and <o_fn> is the name of the generic file which can then be loaded from the I/FEM environment.

3.3 The Interface Between SDRC/I-DEAS and I/FEM

Direct interaction between SDRC/I-DEAS and I/FEM is not supported by either software package. However, it can be accomplished with the aid of NASTRAN. For a model designed in I-DEAS the corresponding MSC bulk data deck can be obtained as described in Section 3.1. If the analysis is going to be done in I/FEM, one first brings the bulk data file to the workstation and then runs the following translator to create a neutral file:

/usr/ip32/msc/bin/ibmsc <i_fn> <o_fn>

where <i_fn> is the name of the MSC file and <o_fn> is the name of the neutral file. Then, the design model can be recovered in I/FEM by translating the neutral file from the I/FEM environment .

For a model designed in I/FEM which is going to be analyzed in I-DEAS, the following procedure applies:

(1) Create an MSC bulk data file from I/FEM as stated in Section 3.2.
(2) Add MSC executive control deck as described in Section 3.1 omitting the following card (which corresponds to the output data, such as displacements, strain, stress, etc):

OUTPUT2 OUGV1,OSTR1,OES1,OEF1,ONRGY1//0/11 $

(3) Run the job with an alter card RF24D32 (used to read the design model without running analysis) and with a .com file storing the OUTPUT2 files.
(4) Retrieve the model in I-DEAS as described in Section 3.1.

The interfacing procedures discussed so far could also support other two combinations among the three software packages: design model in I-DEAS (I/FEM), analysis in NASTRAN, postprocessing in I/FEM (I-DEAS).

IV. THE POSTPROCESSING PHASE IN FINITE ELEMENT ANALYSIS

Highly interactive graphics capabilities in I-DEAS and I/FEM provide advantages in postprocessing. The analyst can display, manipulate and manage analysis results from the graphics environment. The results are then stored in the working design file. Some of the features of the two software packages are listed below:

1) Plot of deformed shape based on loading conditions.
2) Line contours of stress, displacement, strain and moment, which are all based on the results on grid points.
3) Criterion display (used in I-DEAS) or color-coded elements (used in I/FEM) for stress, strain and moment, which are all based on elements.

One may check the plots of stress (or strain) to determine whether the element mesh of the design model should be refined. Refining mesh may be necessary if the contour lines are not smooth enough.

V. AN EXAMPLE OF APPLICATION - DESIGN OF SOLID ROCKET BOOSTER FIXTURE

The purpose of studying this problem is twofold. We want to assure the fixture has enough strength so that it will not fail when used, and we want to verify the procedures described in the previous sections. The results based on using different combinations of the three software packages will be compared.

The fixture is made of aluminum alloy. The components of the fixture consist of a circular plate of 0.379 inches thick and 155 inches in diameter, a cylindrical plate of the same thickness welded under the circular plate, and three lifters which are welded on the outside surface of the cylindrical plate and located 120 degrees apart. To connect the fixture with a solid rocket booster by means of bolts, four sets of small holes are drilled in a circle of 148 inches diameter on the circular plate. Each set are located 90 degree apart and contains eight holes. A crane can then apply load through the three lifters to move the 1200 pound solid rocket booster.

In studying the response of the fixture in statics analysis the boundary conditions are considered to be fixed on the locations where the solid rocket booster is connected. The loads are modeled to be concentrated, upward and equally applied to each lifter. In other words, each lifter carries 400 lbs. To illustrate the procedures stated in Sections II and III, one can analyze the problem just described by means of four different combinations of the three software packages as listed in the following table:

Table 1 - Different Combinations of The Three Software Packages

	Preprocessing	Analysis	Postprocessing
(i)	I/FEM	I/FEM	I/FEM
(i i)	I/FEM	MSC/NASTRAN	I/FEM
(i i i)	SDRC/I-DEAS	SDRC/I-DEAS	SDRC/I-DEAS
(i v)	SDRC/I-DEAS	MSC/NASTRAN	SDRC/I-DEAS

The geometry and element mesh of the fixture developed in I/FEM is shown in Figure 4. It contains 640 nodes and 723 elements of linear triangles and quadrilaterals. The element mesh is designed so that the areas near the constraints and loads have higher mesh density. In order to obtain a meaningful comparison of the results by using different software combinations, the working model created in the preprocessing phase should be the same. Instead of developing of the same element mesh in I-DEAS, analysts can obtain the same working model created in I/FEM by following the procedures stated in Section 3.3. Using combination (iii) from Table 1 above, the result of deformation is shown in Figure 5 (the dash lines represent the undeformed shape). The deformation was scaled by 100 so that the difference between the two shapes can be shown. The CPU times spent doing the analysis phase were 56 minutes in I/FEM, 6 hours 39 minutes in MSC/NASTRAN and 1 hour 36 minutes in SDRC/I-DEAS (the latter two software packages run on the VAX 11/780).

The results based on combination (iv) are shown in Figures 6 and 7. Figure 6 depicts the contours for the displacement field. The maximum displacement is shown to occur at the lifters. The contouring plot of the principle stress field is shown in Figure 7. Both figures show symmetric distribution of the variables about one plane (yz-plane). This should be the case because the geometry, boundary conditions, loads, material and physical properties are all symmetric with respect to this plane. As a matter of fact, the same results can be obtained by studying half of the fixture together with an appropriate boundary condition along the x axis.

The contouring plots of the results based on the other analysis combinations show similar curves for displacement fields but somewhat different curves for principle stress fields. For example, the results based on the combination (iii) are shown in Figures 8 and 9. The following table shows the critical values for the whole structure based on the different solver combinations.

Table 2 - Comparison for The Max. Displacement and Nodal Principle Stress

Analysis Combination	Maximum Displacement	Maximum Principle Stress
(i)	0.157"	5717 psi
(ii)	0.166"	6094 psi
(iii)	0.156"	5891 psi
(iv)	0.166"	2307 psi

It is seen that combination (iv) yields much different maximum principle stress results. The discrepancy may be explained as follows. The output of stresses in MSC/NASTRAN are the stresses related to Gauss points which are the centroid of the elements if linear elements are used. On the other hand, the output of the stresses when using I-DEAS or I/FEM are the stresses computed at each node, which can then be used in contouring plots. Although the grid point stresses in NASTRAN can be assigned to write to a punch file and then be brought to I/FEM, the translation of grid point stress from NASTRAN to I-DEAS is not supported by I-DEAS. However, when the analysis results of MSC/NASTRAN are translated back to I-DEAS for postprocessing, the stresses at each node are accessible. Obviously, the interpolation and extrapolation, from the stresses at the centroids of the elements to the nodal stresses, has been executed. The stresses are therefore different from the one obtained by the actual calculation of the stresses at the nodes. In fact, the stresses related to elements rather than nodes can be obtained by the "criterion" command in I-DEAS and by selecting the "color-coded element" icon in I/FEM. The elemental stresses obtained from MSC/NASTRAN can be retrieved by using these commands. Therefore it may be more reliable to compare the principle stresses at the centroids of elements, as shown in the following table:

Table 3 - Comparison for The Max. Elemental Principle Stress

Analysis Combination	Maximum Principle Stress
(i)	3203 psi
(ii)	3414 psi
(iii)	3225 psi
(iv)	3414 psi

The table shows that the accuracy of the maximum displacement can be obtained for the maximum principle stress if they are based on the centroids of elements. Indeed stresses usually give the most accurate values on the Gauss points [4]. To overlook the stresses at the nodes may be misleading when arriving at conclusions.

All the stresses shown on table 3 are one order lower than the yield stress of the aluminum alloy. Hence the fixture should have enough strength for lifting the solid rocket booster.

VI. CONCLUSIONS

Investigations of model design and statics analysis on three finite element software packages - SDRC/I-DEAS, MSC/NASTRAN and I/FEM were performed. The interfaces between these software packages were tested via the design and analysis of a solid rocket booster fixture. It was found that with the aid of MSC/NASTRAN a model can be developed in I/FEM and translated into SDRC/I-DEAS and vice versa.

Although MSC/NASTRAN provides a very broad range of capabilities in structural analysis, this study shows that for the analysis of the SRB fixture model MSC/NASTRAN consumed large CPU time. Analysts should consider using SDRC/I-DEAS or I/FEM in the analysis as long as it can solve the problem at hand.

Disagreement on nodal stress contours was found between the results from SDRC/I-DEAS and from MSC/NASTRAN. The discrepancy may be due to different computation schemes. However, it is known that the stresses give the most accuracy on the Gauss points. The comparison of these critical elemental stresses does give differences, yet they are within an a

Based on the currently released versions of the software packages, SDRC/I-DEAS has a wider range than I/FEM as to the types of problem it can solve. Also, when compared to I/FEM, SDRC/I-DEAS can create MSC/NASTRAN files for a wider array of problems. For example, a problem like laminated analysis of fiber-oriented composite structure can be solved in SDRC/I-DEAS but I/FEM lacks the capability to solve a problem of this type. However, I/FEM provides a "geometry-based finite element modeling" feature which is not provided in SDRC/I-DEAS. Even in the next version of SDRC/I-DEAS (version 5), only loads (or generalized forces) can be applied on geometry before the element mesh is crated.

REFERENCES

1. Supertab Pre- and Post- Processing Engineering Analysis User's Guide, I-DEAS Level 4, SDRC, 1988

2. MSC/NASTRAN Handbook for Linear Analysis, MSC/NASTRAN Version 64, The MacNeal-Schwendler Corporation, August, 1985.

3. Intergraph/Finite Element Modeling (I/FEM) Operation Training Guide, Intergraph, December, 1988.

4. Cook, R.D., Malkus, D.S. and Plesha, M.E., Concepts and Applications of Finite Element Analysis, 3rd Edition, John Wiley & Sons, 1989.

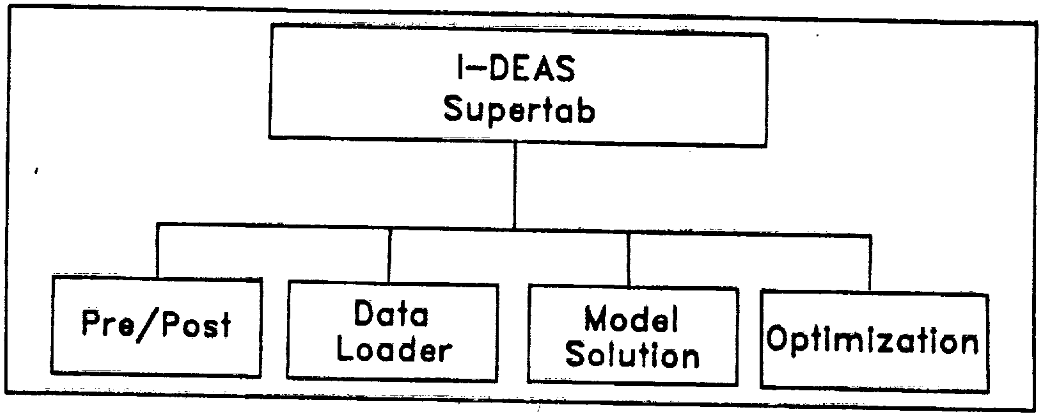

Figure 1. I-DEAS Supertab Modules

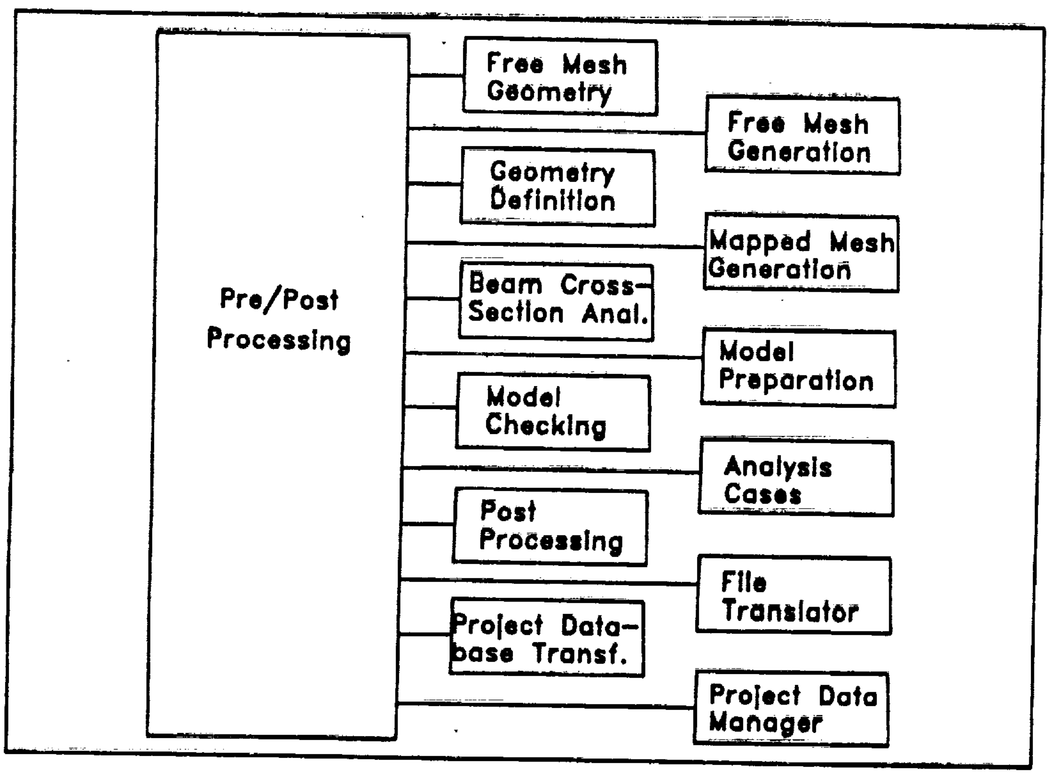

Figure 2. I-DEAS Pre/Post Processing Module

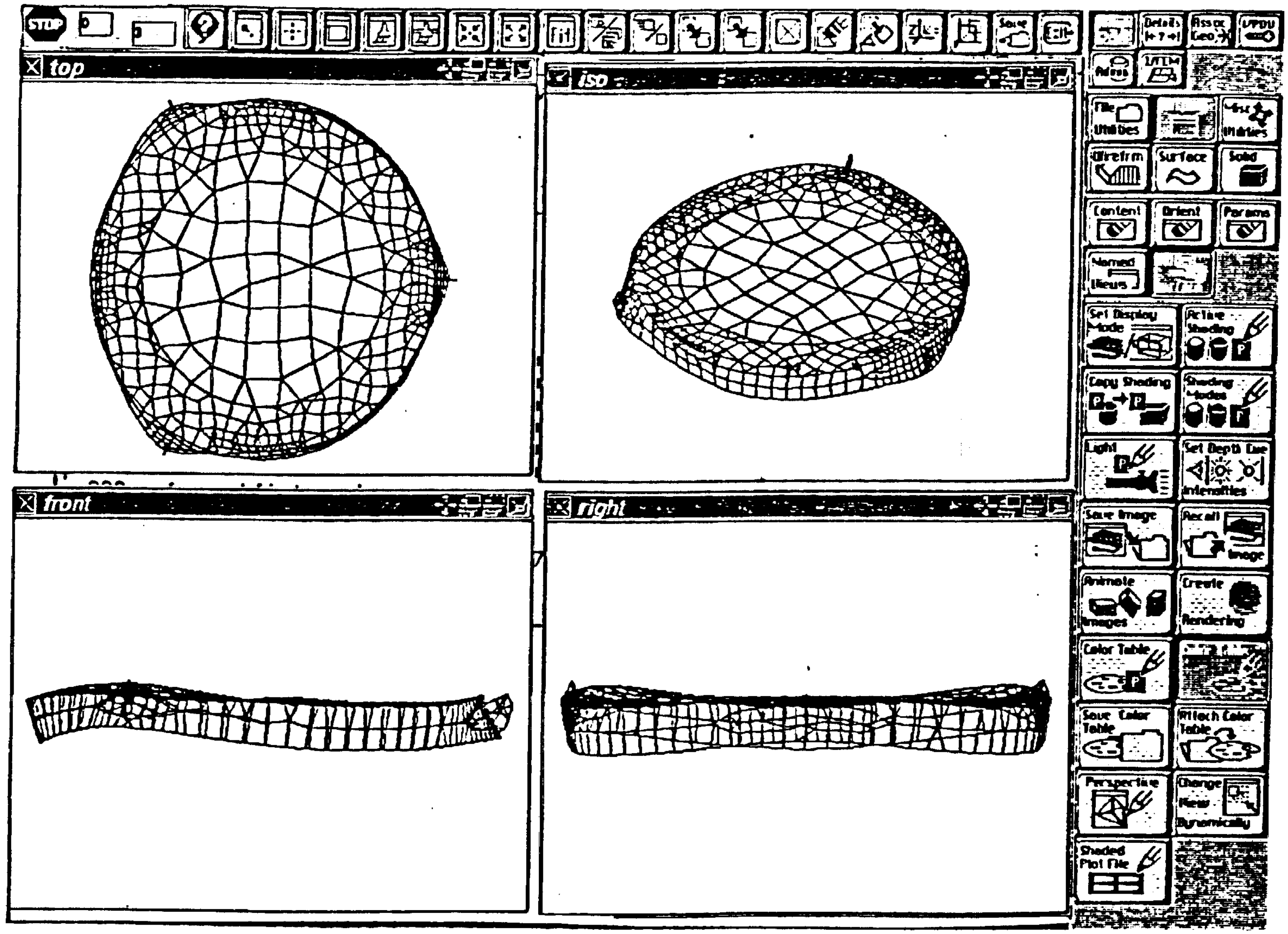

Figure 3. Graphics Menu and Working Windows of I/FEM

ORIGINAL PAGE IS
OF POOR QUALITY

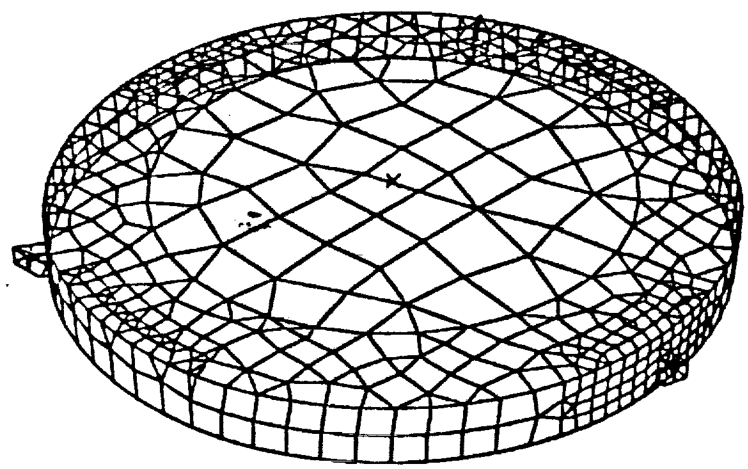

Figure 4. Design of Solid Rocket Booster Fixture in I/FEM

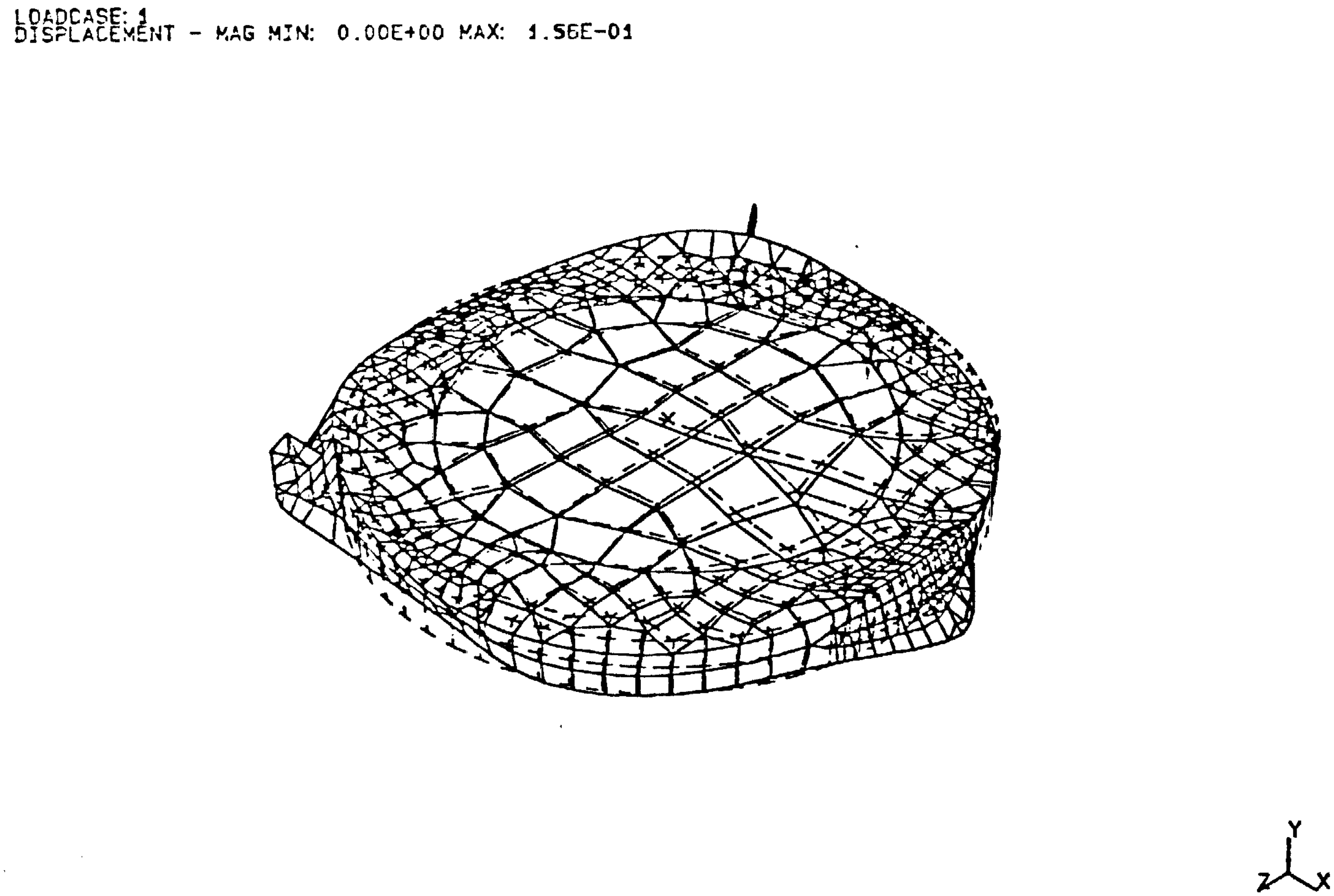

Figure 5. Deformed and Undeformed Geometries of SRB Fixture - Based on the Analysis in I-DEAS

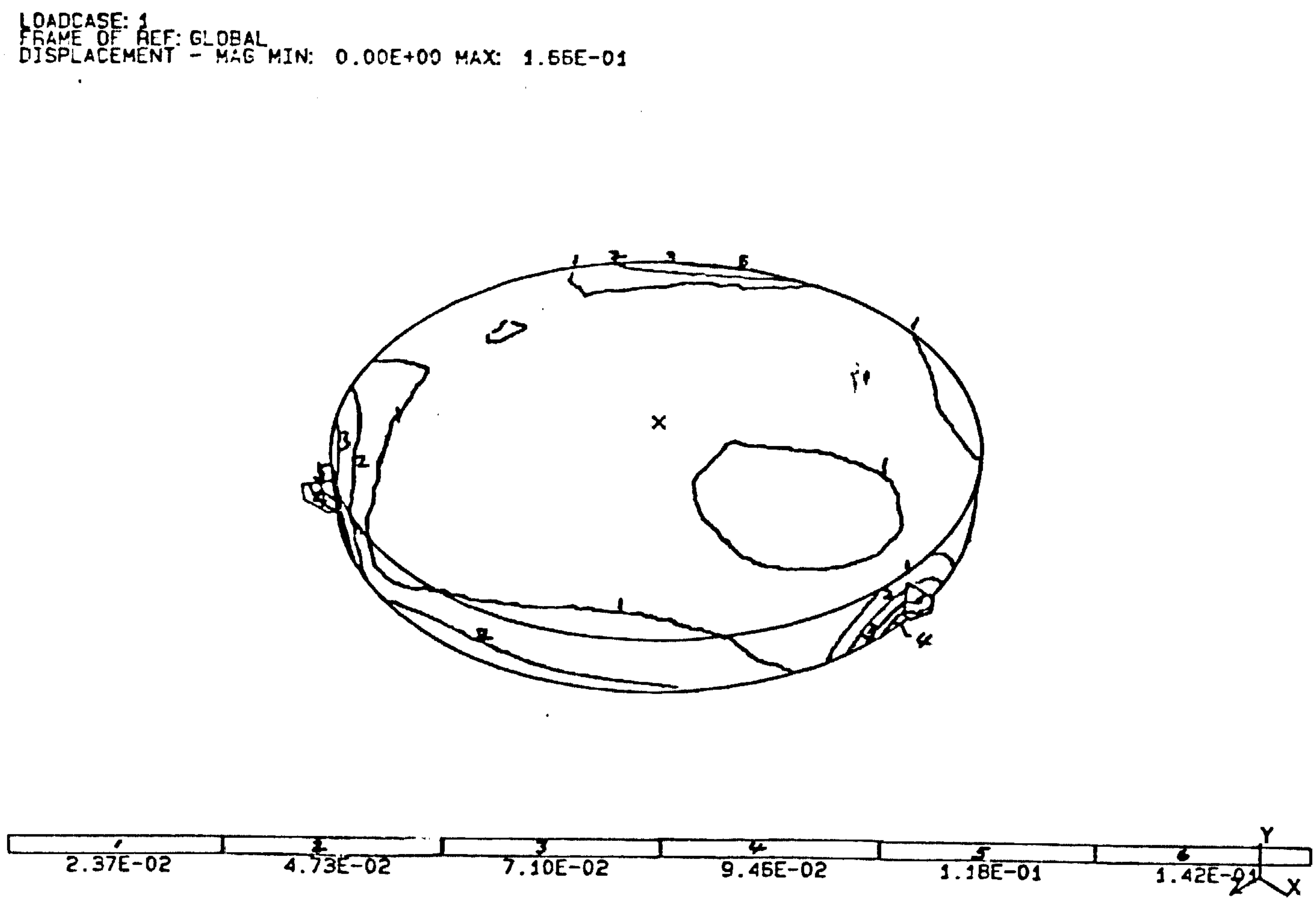

Figure 6. Nodal Displacement Contour - Based on the Analysis in NASTRAN

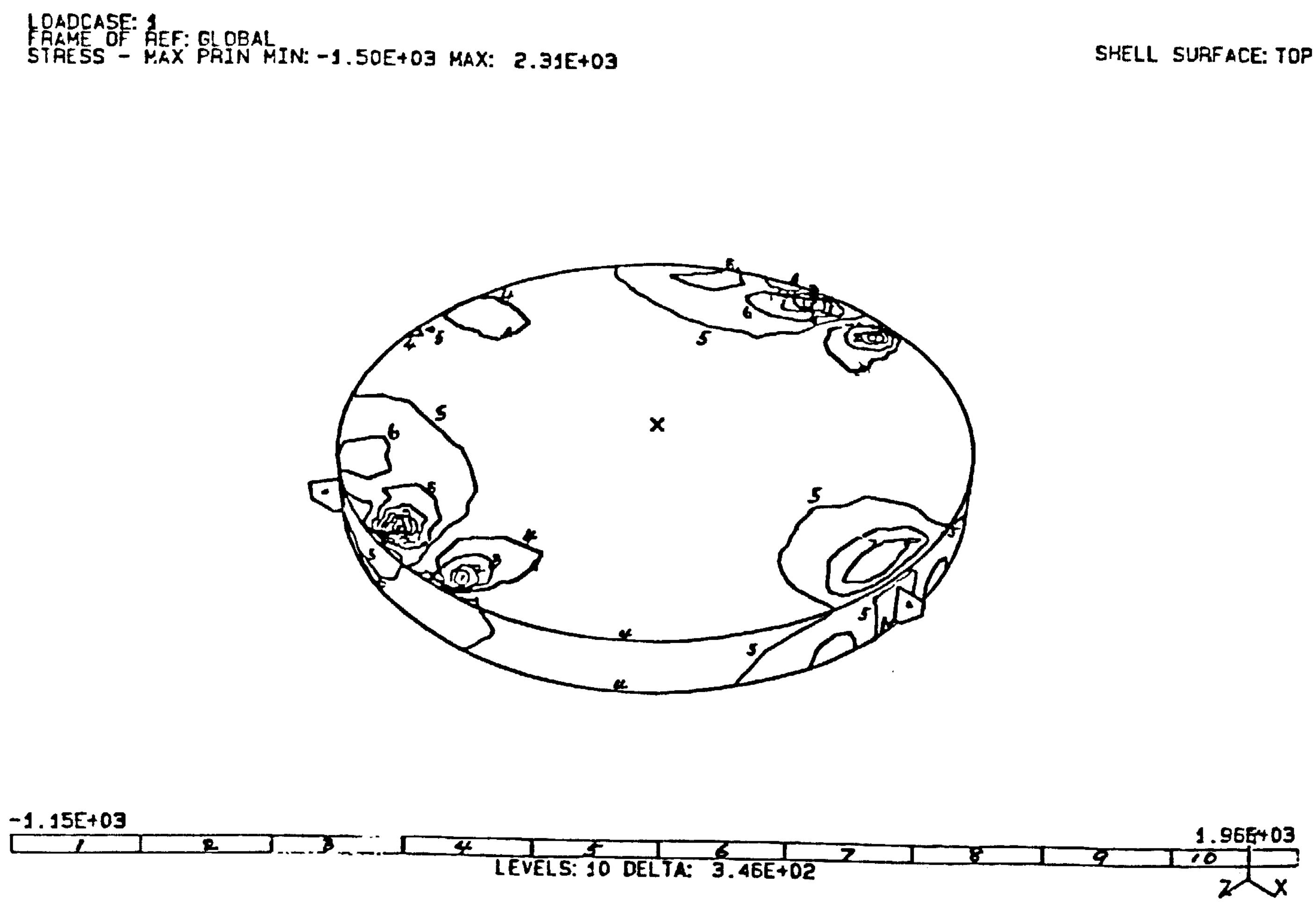

Figure 7. Nodal Stress Contour - Based on the Analysis in NASTRAN

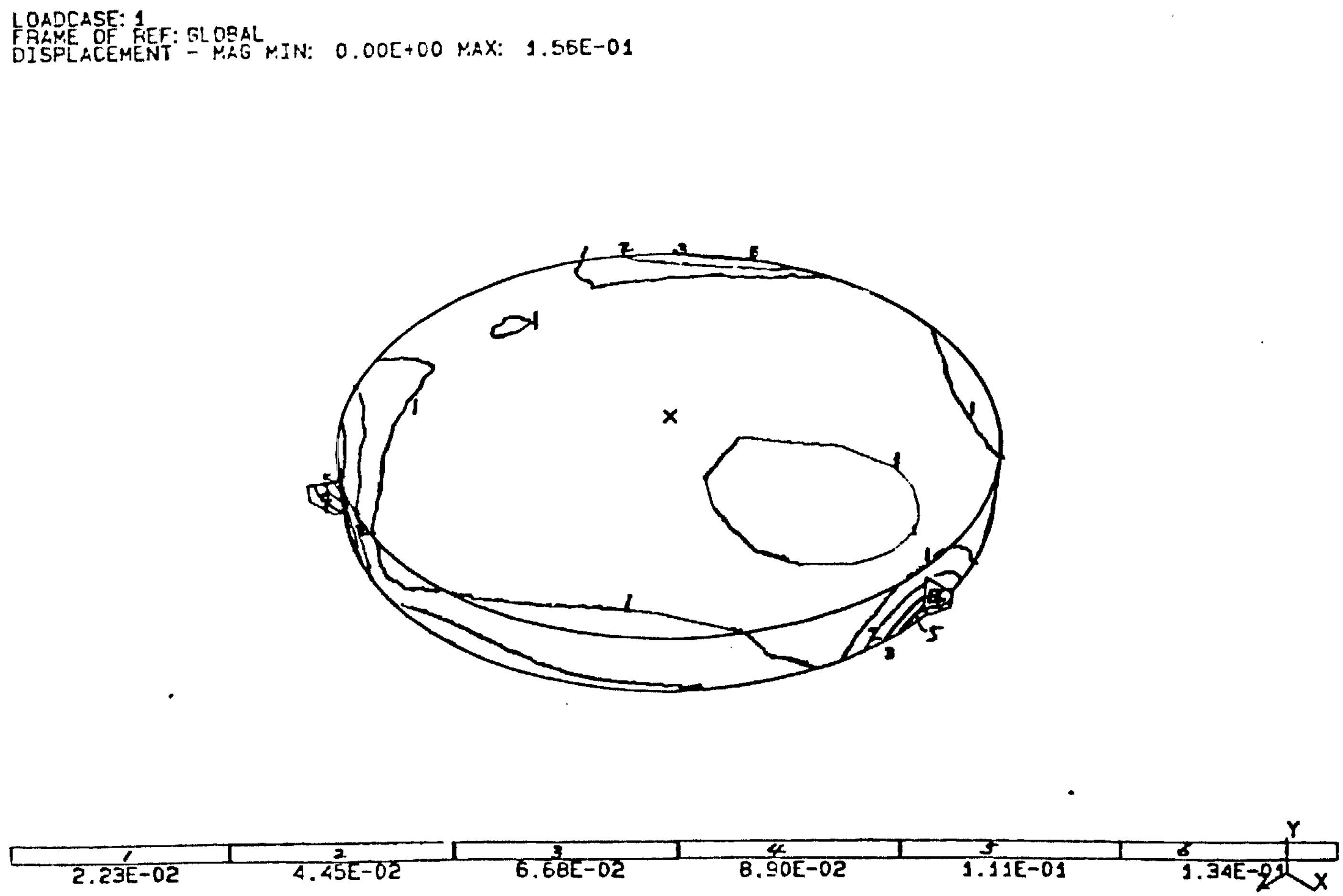

Figure 8. Nodal Displacement Contour - Based on the Analysis in I-DEAS

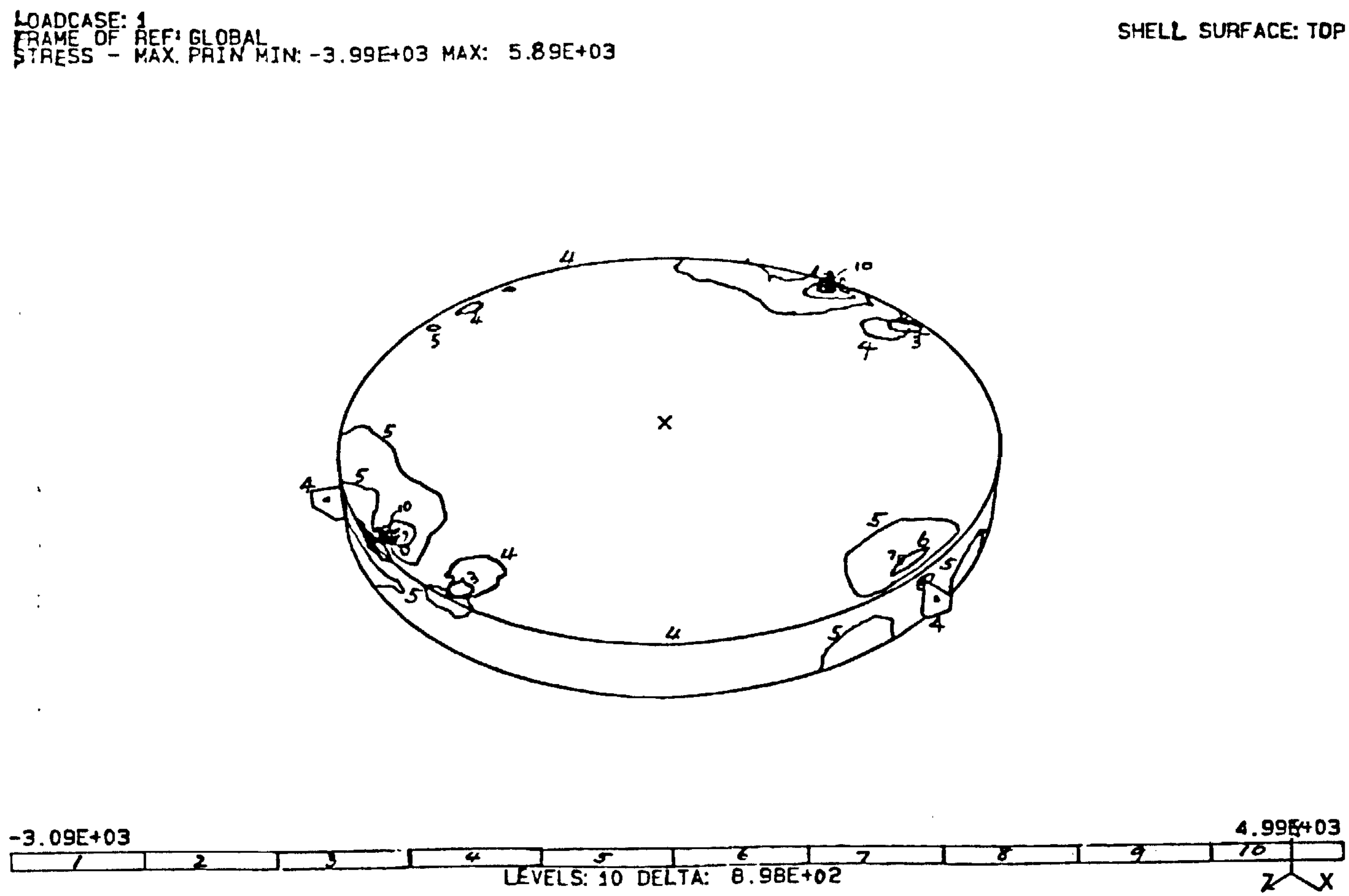

Figure 9. Nodal Stress Contour - Based on the Analysis in I-DEAS