

7 N85-25881

USE OF COSMIC NASTRAN IN A DESIGN DEPARTMENT

Richard C. Coddington and Jose O. Nazario
John Deere Dubuque Works
Dubuque, Iowa 52001

SUMMARY

Application of COSMIC NASTRAN to product development in a design department requires low cost, rapid model construction and interactive evaluation of results. The NASTRAN processor developed by Jose O. Nazario meets these requirements by automating many of the modeling activities.

INTRODUCTION

The proliferation of finite element codes in today's competitive environment complicates the choice, "Which code is best for me?". Generally the user prefers to use one general purpose code that covers most of his or her applications.

NASTRAN is an ideal candidate for several reasons. It is thoroughly tested with thousands of practical applications in diverse situations. It handles large scale problems. It is well documented and well supported.

In the past, lack of automation of model construction, of model checking and of display of results hindered efficient use of NASTRAN. This was especially true in a product engineering environment. Jose Nazario's NASTRAN INTERFACE PROGRAM (NASTI) overcomes these objections by fully automating these needed functions.

PURPOSE

The main purpose of this paper is to highlight the manner in which NASTI automates model preparation, model checking and display of results. The writers will emphasize the features that make NASTI a cost effective tool for structural analysis in a product engineering department.

We will create a simple model of a bucket (Figure 1) to illustrate the many automation features in NASTI. Later, we will include applications of finite element models to three frames. These applications illustrate the level of detail included and the size of models run to date.

FEATURES OF NASTRAN INTERFACE PACKAGE

NASTI contains four main parts.

- * Digitize the model
- * Plot and check the model
- * Create the bulk data and submit job
- * Display the results

On line documentation in each of the main parts assists the user in executing the variety of options available.

Digitize the Model

While in DIGITIZE MODEL the user can create regions using the tablet, the screen or the keyboard (this geometry is three-dimensional), can generate the finite element mesh and can save either the regions, the mesh or both.

Plot and Check the Model

While in PLOT AND CHECK MODEL the user has a myriad of options that automate model construction and data checks.

Create the Bulk Data

CREATE BULK DATA formats the bulk data. It creates the executive and case control statements and automatically generates SPC's and MPC's. Elements currently supported include CBAR, CTRIA, CQUAD, CWEDGE, CTETRA, CTRSHL and CHEXA.

Display the Results

Is not a picture worth a thousand words? NASTI permits the user to plot the results of the analysis. Options include:

- * Plot deformed geometry
- * Plot stress or strain contours
- * Plot stress vectors
- * Plot Von Mises stresses

The more important display features include:

- * Automatic location and display of maximum displacement
- * Automatic location and display of maximum and minimum stresses or strains
- * Interactive zoom and window of plots

DIGITIZING THE MODEL USING NASTI

NASTI provides a four step process to create a finite element mesh. Suggested steps include create regions, generate the mesh, save the mesh and save the regions. Editing and graphical options permit the user to change and view the regions and/or mesh at any convenient intermediate step.

Creating regions

To start the user divides the structure into a collection of four sided regions. The sides of the regions may be straight, curved, or a combination of the two. Figure 2 shows the seven regions we picked to form the side of the bucket.

Regions can be created thru the digitizing tablet, the keyboard, the screen or by copying regions from a file. In many applications to new designs use of the digitizing tablet is most convenient. We will describe mesh creation using the tablet.

The user defines regions created to be in three-dimensional space. The surfaces created may be flat or curved. The user simply inputs three points (x_i, y_i, z_i) , $i=1,2,3$ to define a flat plane. As each region is created, it is automatically connected to neighboring regions to form a "patchwork" that makes up the planar part.

Frequently frame structures consist of intersecting flat plates. In this case, each flat plate is created individually. NASTI automatically ties contiguous plates together along their common edge.

For a curved surface the user creates points along the curved sides of the region. Later he can form the regions using the cursor or keyboard.

Copy feature automates creation of regions

The copy feature in NASTI simplifies construction of regions for both flat and curved surfaces. To construct regions with curved sides, the user simply copies the "key" nodes along the curved boundaries to the desired location and then uses the cursor or keyboard to form regions.

The user may also choose to copy regions to the desired location (See Figure 3) and later delete unwanted regions (regions 8 thru 14). The user then uses the cursors to form the back of the bucket (regions 16 thru 20). Figure 4 shows the final "patchwork" of regions forming one-half of the total bucket. The plane $z = 0$ is a plane of symmetry. Later, the symmetry generator can be used to automatically form the "other half" of the bucket.

Regions, key nodes or both may be copied as desired to form new regions. The flexibility and ease of this feature greatly reduces model construction time.

Generating the finite element mesh

Once the user has his regions it is easy to automatically generate the elements. The user simply selects the generate option to specify the number of elements along the first two sides for each region. He can do this individually for each region or for all regions if they are to be the same. Figure 5 shows the mesh generated for one-half of the bucket.

Interactive documentation aids casual users

On-line documentation guides the user in selecting each option. The experienced user can enter options without this aid whereas the inexperienced user can press the RETURN key for help. The interactive documentation also helps jog the memory of the occasional user who may not recall which option to use next.

Saving the regions and mesh

Once the user is satisfied with the mesh, then he can save both the mesh and the regions for later recall should the design be modified at a later date. He is then ready to go on to plotting and checking the model.

PLOTTING AND CHECKING THE MODEL

The user needs to check the finite element mesh for accuracy. Potential errors cover a broad spectrum of possibilities. Some of these include wrong direction for connectivity, incorrect connectivity, holes and overlaps with the mesh, errors in geometry, wrong material specifications and many others.

In addition to checking the model, this part of NASTI permits the user to efficiently alter the geometry of the part to reflect a design change. Constraints on material, manufacturing process, part fitup, cost of the part, use of existing parts and many other reasons may dictate the need for a design change. For whatever reason, this change must be easy and fast to accomplish. Otherwise, results of the analysis may arrive too late to aid in final design decisions.

NASTI has four main parts to facilitate model checking and modification. These parts include preset options, graphic options, edit options and input of loading and boundary conditions. Once again, as the user selects an option, interactive help is available to assist the user in proper use of the option.

Stringing together commands saves time

Many of the available options are not necessary but are extremely handy in automating model checking and changes. It is also very helpful to be able to string options together. This capability saves considerable time for the experienced user while not hindering the casual user. The casual user simply enters one option at a time. With experience the user gains confidence and starts to string options together. The following two examples illustrate the idea.

EXAMPLE 1

R A 30 150 0 O N T N G N P

The meaning of this string of commands is as follows:

COMMAND	MEANING
R A 30 150 0	Rotate the view using angles 30,150,0 degrees
O N	Turn off the outline (Let's user see elements)
T N	Turn the element tag off
G N	Turn grid number tag off
P	Plot the structure

The options selected by the user remain active until turned off. Thus the user need not repeat the options each time. Figure 6 shows the outline of the part prior to execution of the options shown. Figure 7 shows the result. The user can magnify the view, tag the elements and plot the result with the following string of options:

EXAMPLE 2

Z Y T Y P

COMMAND	MEANING
Z Y	Turn the zoom option on
T Y	Tag the elements
P	Plot the results

Figure 8 shows the part prior to execution of these options. Figure 9 shows the result.

Part Identification with structure

Frequently the total structure consists of many distinct parts. It is convenient to identify the individual parts to facilitate the checking process. It's usually easier to check each part individually than collectively. In addition, it is convenient to display the results as contour plots for individual parts of the total structure. Thus identification of individual parts by physical property type serves a dual purpose.

The writers changed the material specification for the leading edge of the back of the bucket from 4 to 5. This change permits flexibility to specify a stronger material for the cutting edge of the bucket.

Checking the Model for Cracks and/or Connectivity

Figure 10 shows the outline of the bucket. The top right portion represents the side of the bucket and the bottom left portion the back of the bucket. The outline on the bottom right portion of the bucket is its cutting edge.

The line between the back and side of the bucket could indicate either different materials, a crack exists between the back and the side, double elements (one on top of another) exist, elements are disconnected or elements are missing. Let's determine which of these alternatives apply.

Select the edit option thru the cursors by entering U C .
These options include:

- * Change element connectivity
- * List grid points and element properties
- * Form or delete elements
- * Align grid points along a line or in a plane
- * Measure distances between grid points
- * Copy or interpolate grid points
- * Show relative element connectivity

Use the V option to show differences (if they exist) in the connect'vity of each element. A plus sign in each element indicates that no elements are missing. By turning the outline option off and the element tag option on, the user can verify that no double elements exist. Next the user can plot the elements associated with material 2 by using the preset option for material groups in the main menu. The result is a plot of the side of the bucket only. This result indicates that differences in material cause the line shown between the side and the back of the blade.

Automating Design Change to include a hole in the bucket

Suppose the designer decides to include a circular hole in the back of the bucket after completion of the model. This modification is easily done by deleting elements in the area where the hole is to go, by using a local coordinate system to create grid points on the edge of the hole and by forming elements using the cursors. Figures 11 thru 16 show the sequence of options to rapidly modify the initial design to include this circular hole. The steps include:

Action	Figure(s)
* Delete elements in region where hole is to occur	11
* Use copy option to create nodes on radius of hole about a local x-axis	12
* Use the align option to place the nodes in the surface of the blade	13
* Use the interpolate and copy options to create additional nodes for mesh near the hole	14
* Fill in elements around the hole and display the back of the bucket	15
* Rotate the view to check that all elements in the back of the bucket are in the surface of this part	16

Automated Mesh Generation

After the analyst completes checking the model he is ready to generate the other half of the bucket. To do so the user merely specifies the plane of symmetry, the grid point increment (for numbering the new nodes) and the element increment (for numbering the new elements). For flexibility the user can save grid points, elements or both. Further, the user can choose to translate the "master set" of elements and/or grid points if desired.

Interactive application of Constraints and Loads

Once the model geometry is complete (See Figure 17) the user is ready to apply loads and to restrain the structure where it attaches to other components or to the ground. The user first selects the material number that identifies the part where he will apply constraints. Next he rotates this part to a convenient view. He can either automatically generate constraints in user specified directions or manually apply them using the cursors.

Figure 18 shows the fore-aft constraints on the back of the bucket. They might represent the the restraining action of a hydraulic cylinder on the back of the bucket.

Application of point loads is easily and quickly done by specifying the part, picking a good view, stating the force components and by using the cursors to show where they apply. The final step is to check the accuracy of the information by plotting the parts where the constraints and loads occur and selecting the display option for this information.

Figure 18 displays the pry load acting on the cutting edge of the bucket.

CREATION OF THE BULK DATA

NASTI fully automates the creation of the bulk data prior to job submission. Even on a large problem (over 10 000 degrees of freedom), the time to create the bulk data is under five minutes.

Removal of Singularities and Selecting Material Properties

The first step in creating the bulk data in NASTI is to

apply the single and multipoint constraints to remove the singularities from the stiffness matrix. Nazario developed an algorithm to automate the calculation and application of these constraints. The actual calculation is transparent to the user and normally takes less than one minute to complete.

The remaining steps to complete the bulk data include the following specifications:

- * Number of load cases
- * Thickness for each material
- * Material properties

After this input is complete, NASTI informs the user that the bulk data is complete and lists totals for the number of elements and grid points.

Submitting the Job for Batch Processing

NASTI fully automates the clerical work involved in preparing the executive control, the case control and file manipulation. Input includes:

- * Job Name and Project Number
- * Destination for Output
- * Time estimate for solution
- * Job Priority
- * Name for Output Data Set
- * Checks for creation of checkpoint tape
- * Checks to see if Restart Dictionary is to be used
- * Number of subcases
- * OK to submit job for batch execution

If the user wants to use non-standard options, then he elects not to submit for execution. He can then interactively edit the executive control, the case control or any feature in the bulk data to suit his needs.

DISPLAY OF RESULTS

Rapid data reduction is essential to effective use of finite element technology in today's modern design department. The design engineer must get the reduced data at a time where he can use it to make timely design decisions.

NASTI shrinks the time for data reduction by automatically sorting the output for maximum principal stresses, minimum principal stresses, maximum shearing stresses and maximum Von Mises stresses. The user can exploit this data to quickly zero in on regions of high stress and by using the interactive plotting features to display these regions.

Displacement Plots give Clue to Overall Structural Response

Figure 20 shows the deformation of the bucket due to pry loads on the cutting edge. Relative movement of the sides and back of the bucket give a qualitative "feel" of response of the bucket to pry loading. The printed maximum displacement quantifies this movement of material.

Contour Plots of Stress and Strain

Eighteen standard contour plots are available to display the distribution of stress or strain in the structure. The options include:

- * Principal Stress Contours
- * Principal Strain Contours
- * Shearing Stress Contours
- * Shearing Strain Contours
- * Von Mises Stress Contours
- * Von Mises Strain Contours

The usual procedure is to select the material to plot, rotate to a convenient view and pick the desired contour plots. Rotate, window and zoom features permit the user to focus on areas with large stress or strain gradients.

Figure 21 shows the distribution of strain on the front surface of the back side of the bucket. Figure 22 shows the detailed maximum principal strain distribution where the hydraulic cylinder attaches to the back of the bucket.

PRODUCT EVALUATION USING COSMIC NASTRAN

Engineers at the Dubuque Works use COSMIC NASTRAN to evaluate structural response of new and current products. The application to the design of frames, booms, buckets, castings and weldments has been particularly fruitful.

The level of complexity and model size used for the simulations depends on the particular application, the time frame for the analysis and the cost limits for the evaluation. The three examples included below illustrate the larger models used for evaluation of frames on our motor graders, our four wheel drive loaders and our two wheel drive loader-backhoes.

Figure 23 shows a schematic of the motor grader main frame. The model of this frame contains quadrilateral plate elements, triangular plate elements and bar elements. It has about 5000 elements, 4700 grid points and 24 000 degrees of freedom. As the design progressed the analyst used this model to try various plate thicknesses, reinforcements, and geometry changes to "tune the design" for improved performance.

Figure 24 shows a schematic of the engine frame on a four wheel drive loader. The model of this frame contains quadrilateral plate elements, triangular plate elements, solid elements and bar elements. It has about 7200 elements, 5300 grid points and 20 000 degrees of freedom. The analyst used this model to conduct "numerical tests" of the structure to evaluate its performance for different types of loading.

Figure 25 shows a schematic of the main frame of a two wheel drive loader-backhoe. The model of this frame contains quadrilateral plate elements, triangular plate elements, solid elements and bar elements. It has about 18 000 elements, 10 000 grid points and 50 000 degrees of freedom. The analyst used this model to evaluate design options for the mast and for its interaction with the main channel of the frame.

CONCLUSIONS

The NASTRAN INTERFACE PROGRAM (NASTI) fully automates construction, checking and data reduction in finite element models using COSMIC NASTRAN for structural analysis. This automation is essential to the fast paced environment that exists in new product design.

Design changes "on the fly" mandate the need for rapid and accurate data to support decisions on the proposed changes. The NASTI program meets this need by shrinking the time to "test" more proposed changes. The significant impact on our business is a better performing product at lower cost.

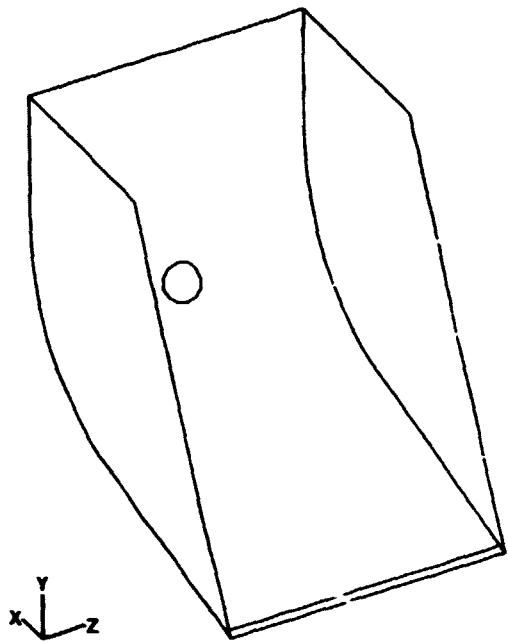


FIGURE 1 BUCKET

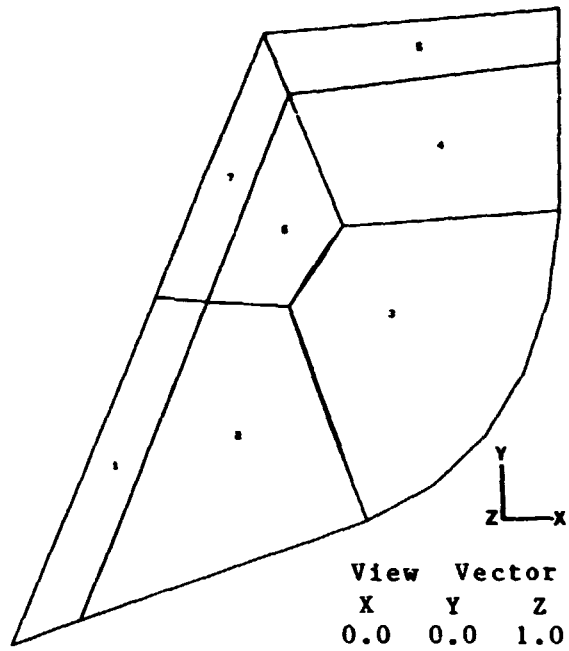


FIGURE 2 REGIONS

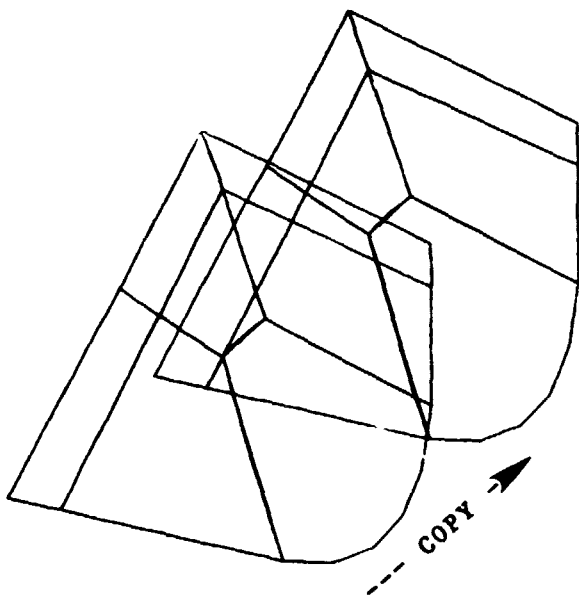


FIGURE 3 REGION COPY

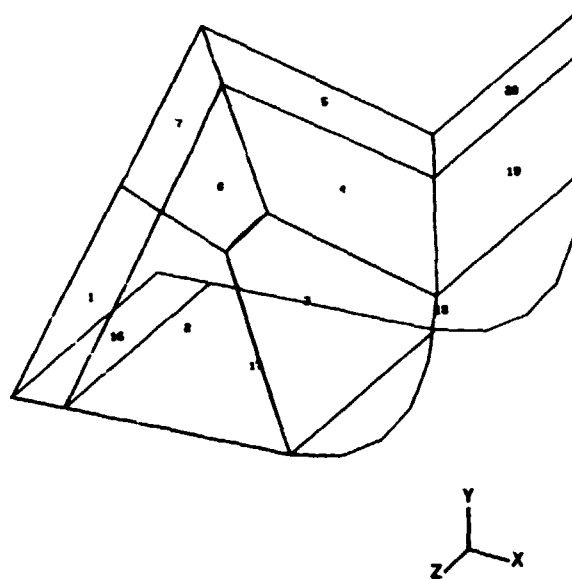


FIGURE 4 REGIONS FOR ONE-HALF OF THE BUCKET

ORIGINAL PAGE IS
OF POOR QUALITY.

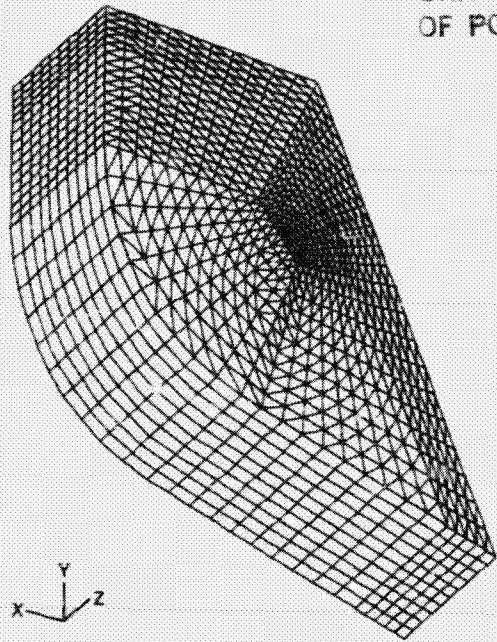
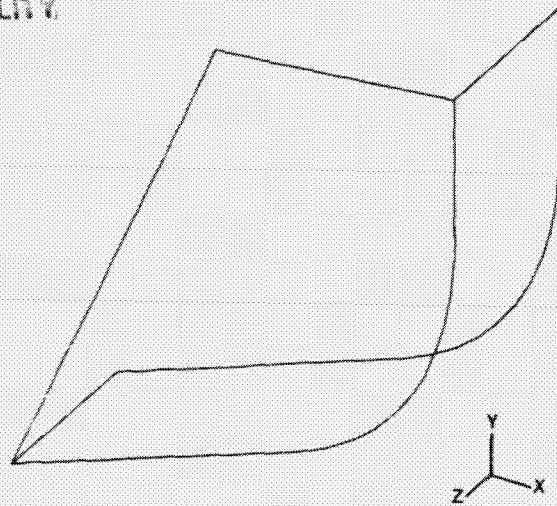
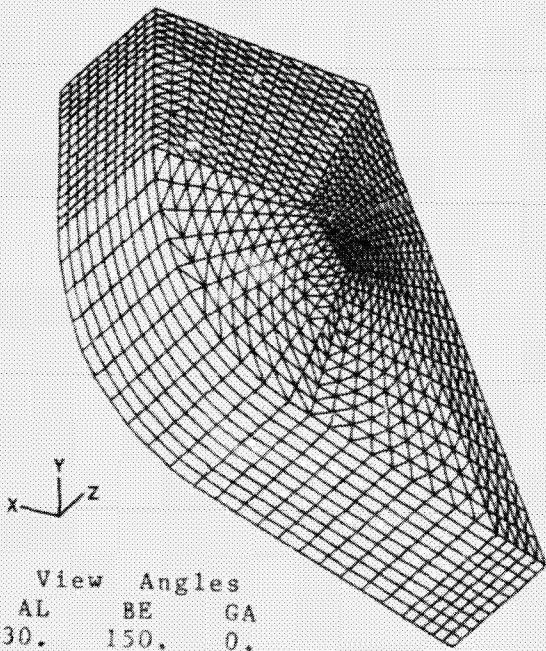


FIGURE 5 FINITE ELEMENT MESH
FOR ONE-HALF OF BUCKET



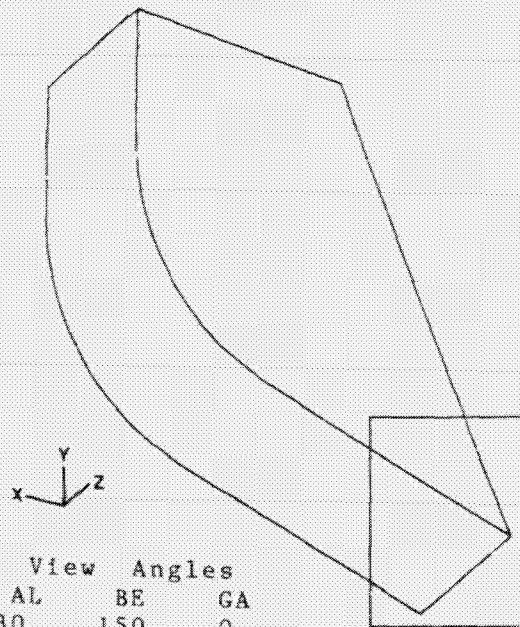
View Angles		
AL	BE	GA
30.	-30.	0.

FIGURE 6 OUTLINE OF BUCKET



View Angles		
AL	BE	GA
30.	150.	0.

FIGURE 7 ROTATED VIEW
SHOWING ELEMENTS



View Angles		
AL	BE	GA
30.	150.	0.

FIGURE 8 ZOOM AND TAG
ELEMENTS

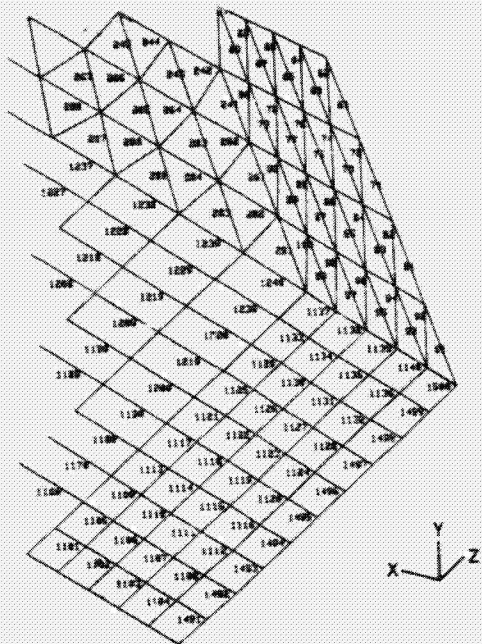


FIGURE 9
ELEMENT IDENTIFICATION

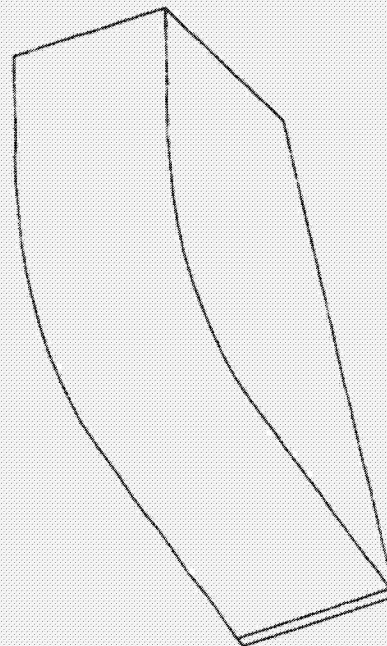


FIGURE 10 BUCKET OUTLINE

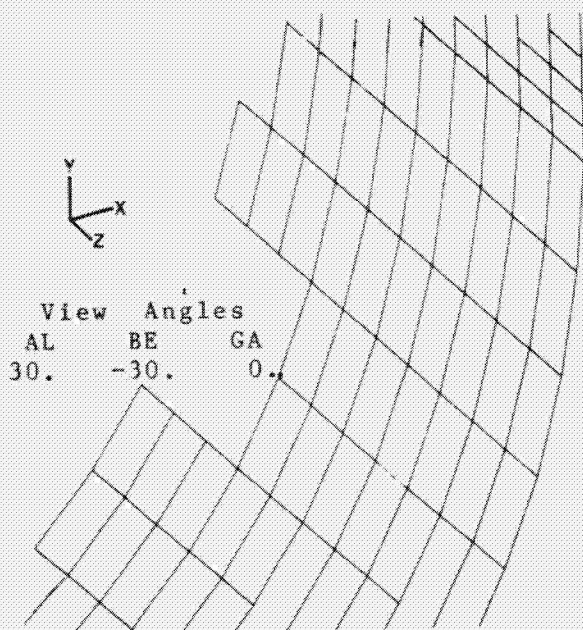


FIGURE 11 DELETION OF ELEMENTS
FOR CIRCULAR HOLE

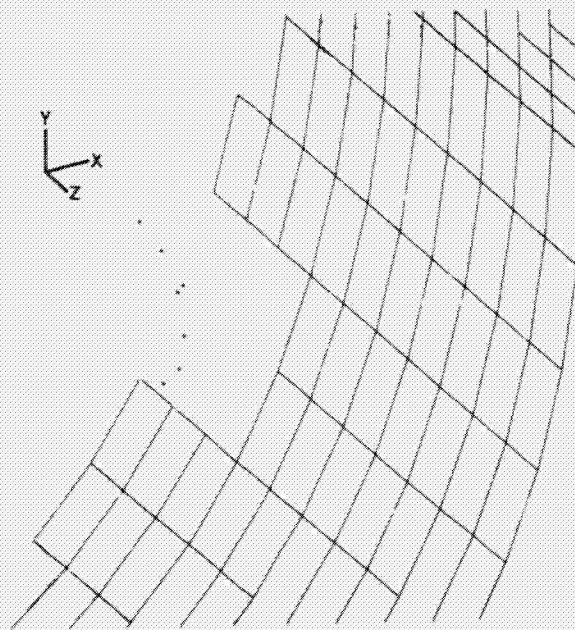


FIGURE 12 GENERATE GRID POINTS
USING COPY FEATURE

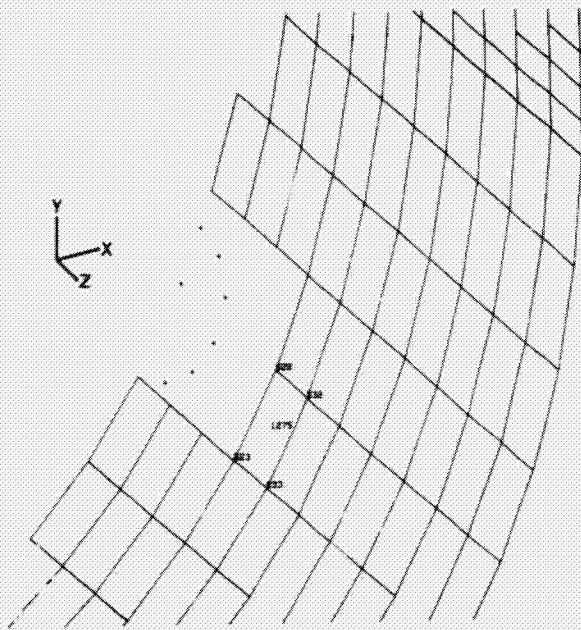


FIGURE 13 USE OF ALIGN OPTION

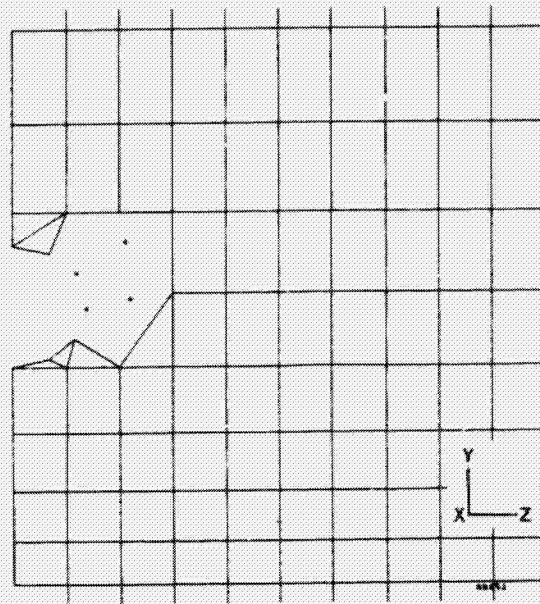


FIGURE 14
INTERPOLATE AND COPY
TO CREATE NEW GRID POINTS

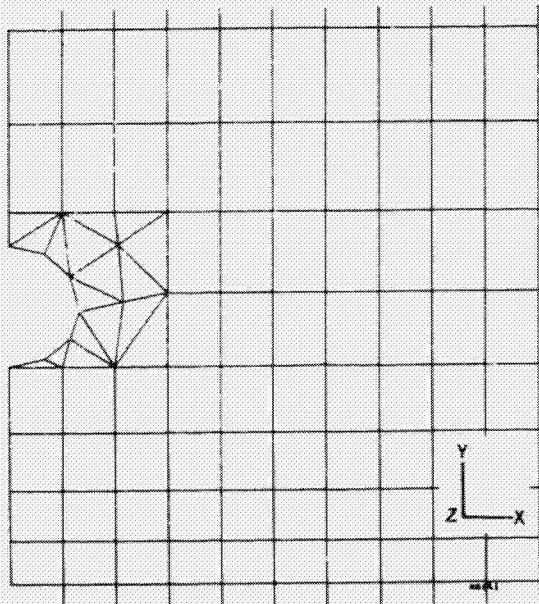


FIGURE 15
MESH AROUND CIRCULAR HOLE

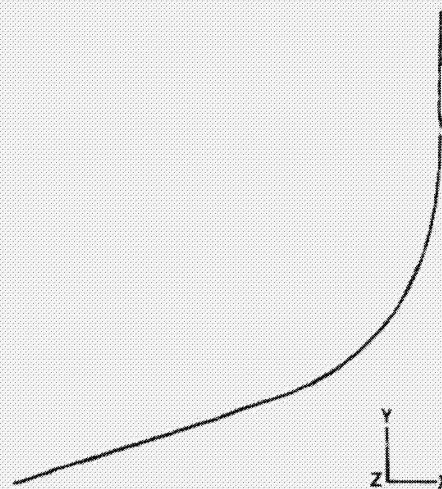


FIGURE 16
SIDE VIEW OF BACK
OF BUCKET

ORIGINAL PAGE IS
OF POOR QUALITY

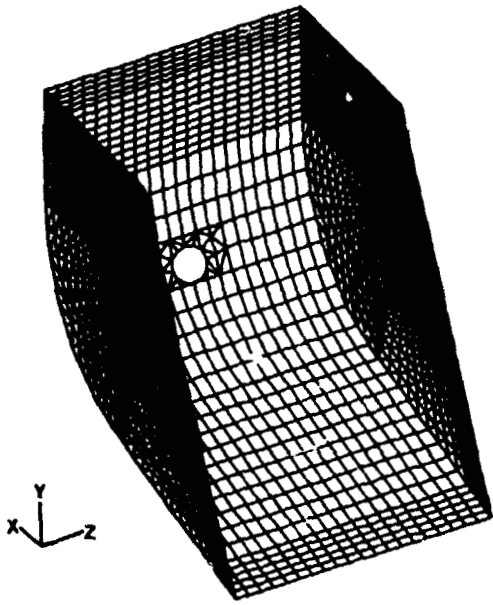


FIGURE 17 MODEL OF BUCKET

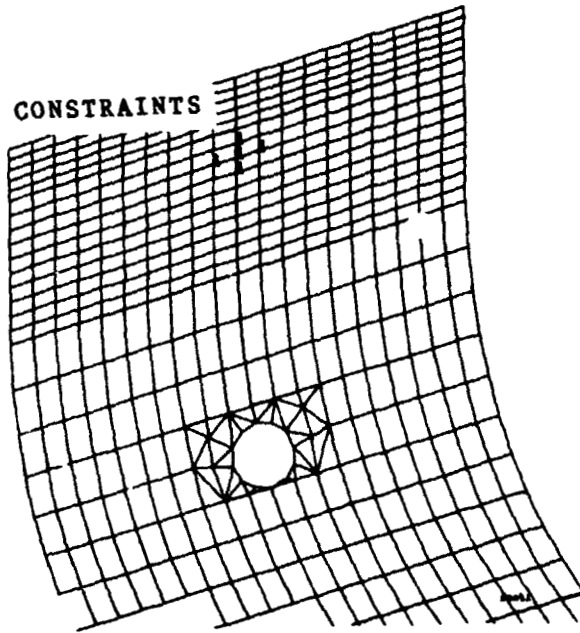


FIGURE 18 FORE-AFT CONSTRAINTS

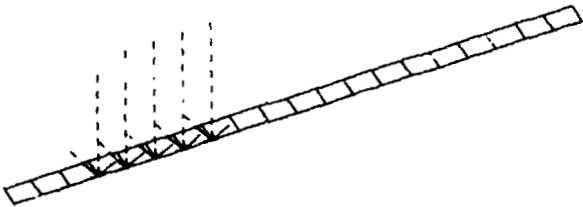
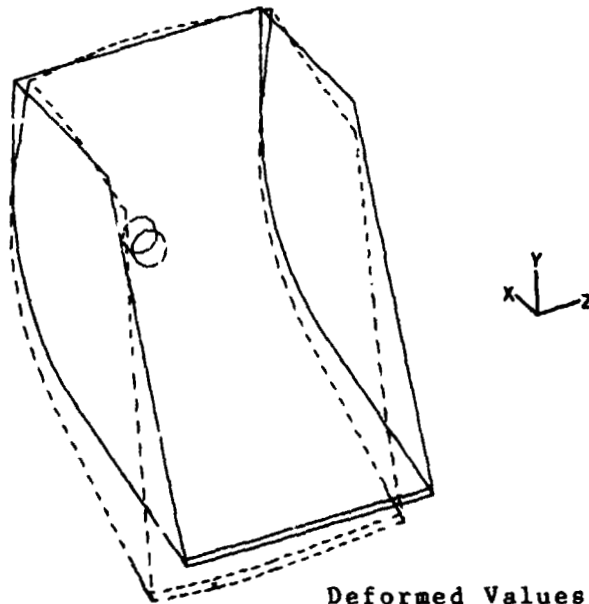


FIGURE 19
PRY LOAD ON CUTTING EDGE
OF BUCKET



Deformed Values:
Max-Resultant Disp = 5.95
... At Gridpoint = 2622

FIGURE 20 DEFORMATION OF BUCKET

ORIGINAL PAGE IS
OF POOR QUALITY

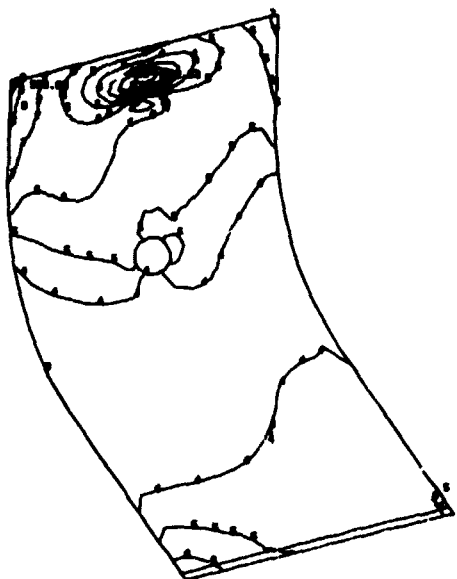


FIGURE 21 DISTRIBUTION OF STRAIN
IN BACK PLATE

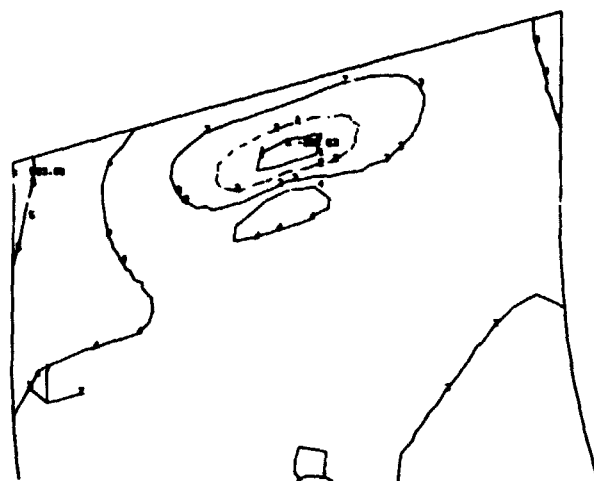


FIGURE 22
STRAIN DISTRIBUTION
NEAR HYDRAULIC CYLINDER

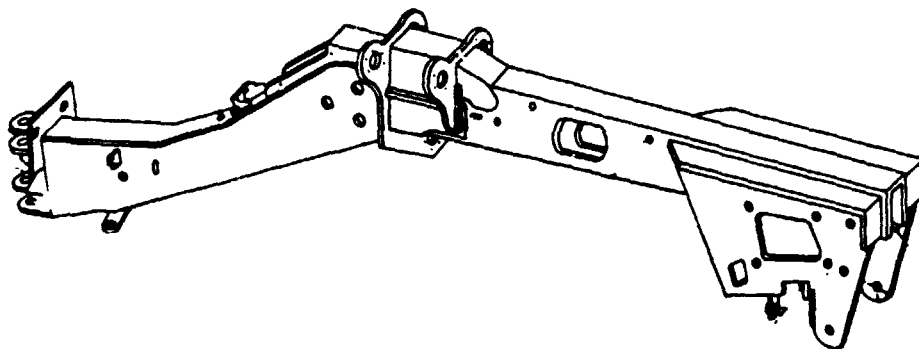


FIGURE 23 SCHEMATIC OF MOTOR GRADER
MAIN FRAME

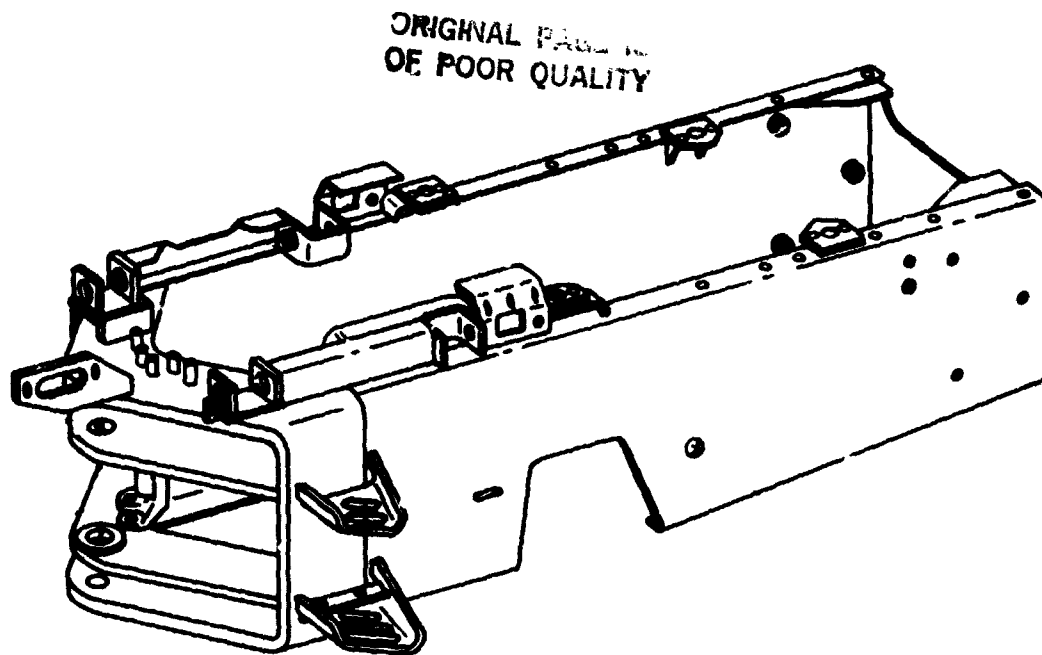


FIGURE 24 SCHEMATIC OF ENGINE FRAME
ON FOUR WHEEL DRIVE LOADER

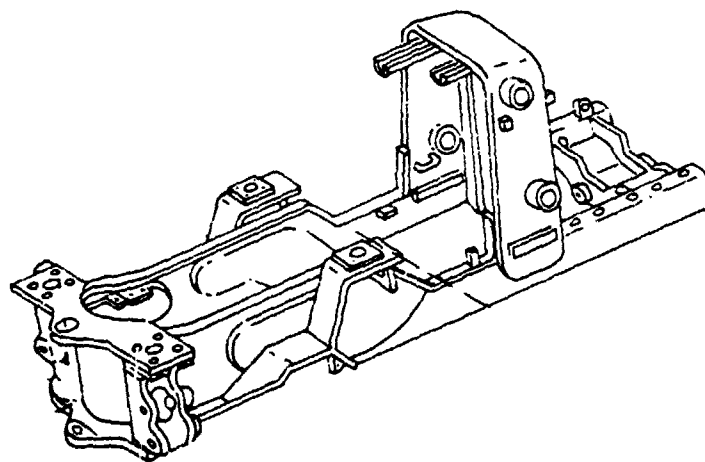


FIGURE 25 SCHEMATIC OF FRAME ON
TWO WHEEL DRIVE LOADER-BACKHOE