THREE DIMENSIONAL STRESS ANALYSIS OF A HELICOPTER MAIN ROTOR HUB USING CYCLIC SYMMETRY

By:

Richard L. Rotelli, Jr.

Jordan, Apostal, Ritter Associates, Inc.
Administration Building 7
Davisville, Rhode Island 02854
(401) 884-3014 (401) 294-4589

Presented At The

1983 MSC/NASTRAN User's Conference
Pasadena, California
March 24-25, 1983

ABSTRACT

A three dimensional stress analysis of the new Main Rotor Hub design for the SH-2F Helicopter was performed using the cyclic symmetry feature of MSC/NASTRAN. The FEMGEN interactive graphics mesh generator was used to create the one-eighth symmetric finite element model. The structural response of the rotor hub to several different loading conditions, as predicted by MSC/NASTRAN, was displayed graphically by the FEMVIEW interactive results viewing program. Presented in this way, the results of the NASTRAN analysis had a positive impact on the new design of the Main Rotor Hub.

ACKNOWLEDGEMENT

The work was performed for the KAMAN Aerospace Corporation, Bloomfield, Connecticut and funded under NAVAIR Contract N0019-82-G-D107. The author worked closely with Robert Mayerjak of KAMAN Aerospace and wishes to gratefully acknowledge his guidance in the preparation of the model and its loading so that a faithful simulation of the rotor hub-blade assembly could be achieved.

I. INTRODUCTION

NASTRAN was used to determine stresses and deformations in a three-dimensional structure of complex geometry. The work was accomplished using several computer codes at geographically separated sites. This paper describes the user's solutions to practical problems concerning the economical generation of the model, the execution of the analysis, the logistics of data transfer, and the organization of post-processed results for efficient interpretation.

The purpose of this paper is not to present some new development in the field of finite element technology. The paper's intent is to demonstrate how MSC/NASTRAN, in conjunction with a well-organized system of graphic pre- and post-processing, may be used to efficiently complete a large analysis project.

The structure that was analyzed is a new rotor hub for the SH-2F helicopter. Figure 1.1 shows the rotor hub's position on the helicopter; Figure 1.2, its general shape. The hub connects the rotor blades to the helicopter. It is the main path for the loads which support the helicopter in flight. In addition, it is subjected to significant centrifugal forces produced by the rotating blades. Its importance to the safety of the helicopter justifies extensive laboratory fatigue testing to demonstrate its integrity. The purpose of the NASTRAN analysis described herein was to assist in the selection of dimensions for test hubs.

A finite element model of a one-eighth symmetric segment of the rotor hub was created using the FEMGEN interactive graphics mesh generation program [1]. The model consisted of 402 second-order solid elements (20-node bricks and 15-node wedges) to idealize the hub itself, and 44 beam and rod elements used to model the Lead/Lag pin to which the rotor blades are attached. The details of the finite element mesh generation are presented in the next section of this paper.

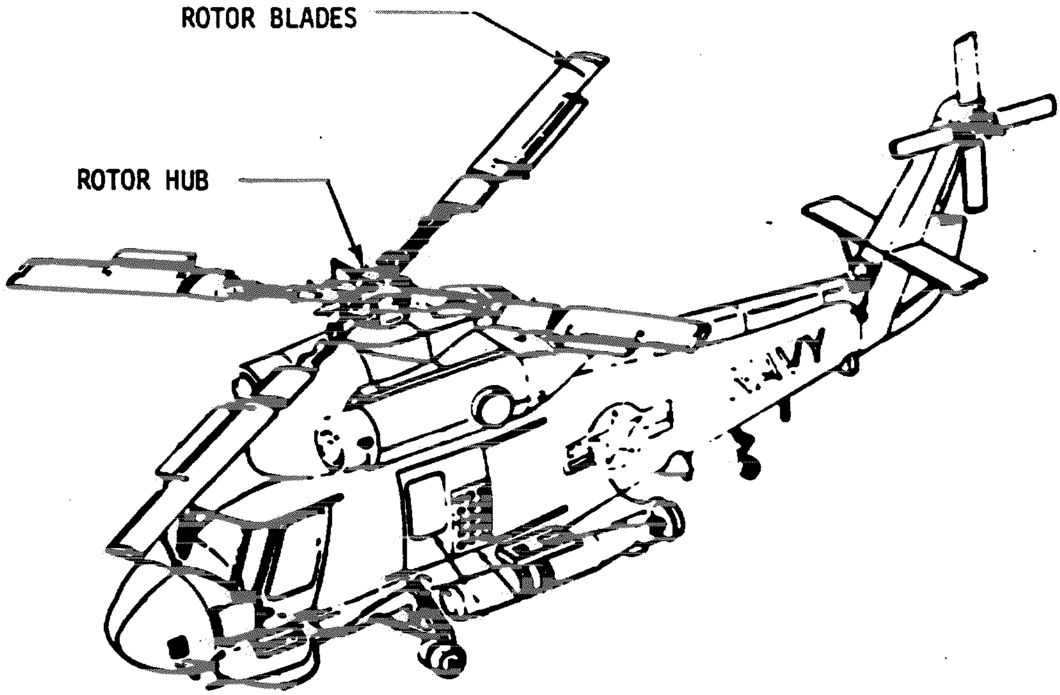


FIGURE 1.1: THE SH-2F HELICOPTER

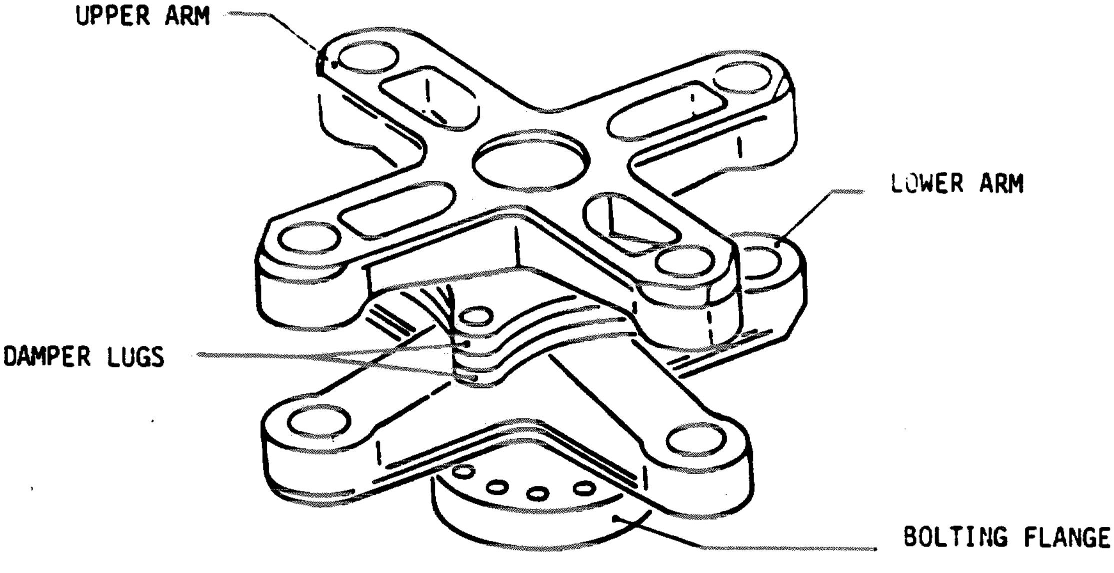


FIGURE 1.2: BASIC ROTOR HUB CONFIGURATION

The FEMGEN mesh generation was done on a VAX 11/780, a computer which is particularly well suited to this type of interactive processing.

The dihedral cyclic symmetry feature of MSC/NASTRAN [2] was used to predict the structural response of the hub to the following seven different loading conditions:

1. C.F. Load (Centrifical Force)
2. Drag Load (Torque)
3. Lift Load
4. Hub Moment Load (Vibratory)
5. Damper Lug Load
6. A Limit Load Combination
7. A Fatigue Test Steady Combination

Since a relatively small number of nodes lie in the boundary planes of the primary half-segment, static condensation [3] was performed on the basic segment before it was transformed into cyclic space to create the other half-segments required for analysis of the full rotor hub. It was felt that the use of static condensation could substantially reduce the total solution time required.

The NASTRAN execution was done on a CRAY-1 computer. The CRAY computer is especially economical for the large batch executions that were required.

To most efficiently present the great volume of numerical results, the stress and displacement data produced by NASTRAN was post-processed by the FEMVIEW interactive graphics results viewing program [4]. The FEMVIEW post-processing was done on the VAX 11/780 computer.

The remainder of this paper presents the details of the three major steps in the analysis procedure; the mesh generation with FEMGEN, the NASTRAN Cyclic Symmetry execution, and the results processing performed via FEMVIEW.

II. THE ANALYSIS PROCEDURE

The details of the analysis procedure are summarized in the flow diagram of Figure 2.1. The three major steps were FEMGEN mesh generation, the NASTRAN execution, and the post-processing done with FEMVIEW. Each of these steps is discussed in the remainder of this section.

MODEL CREATION
VAX 11/780

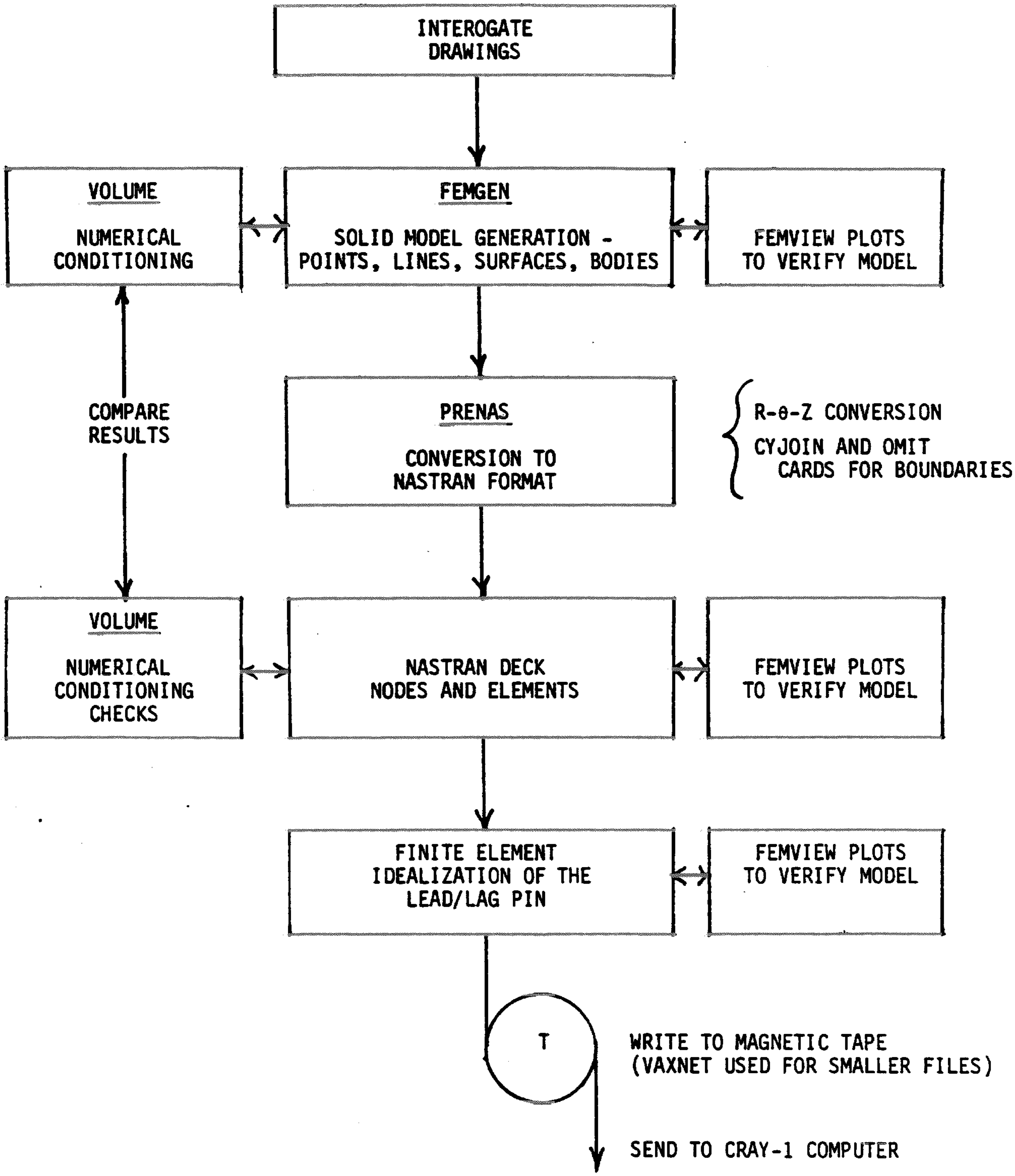


FIGURE 2.1: THE ANALYSIS PROCEDURE (PAGE 1 OF 3)

NASTRAN EXECUTION

CRAY-1

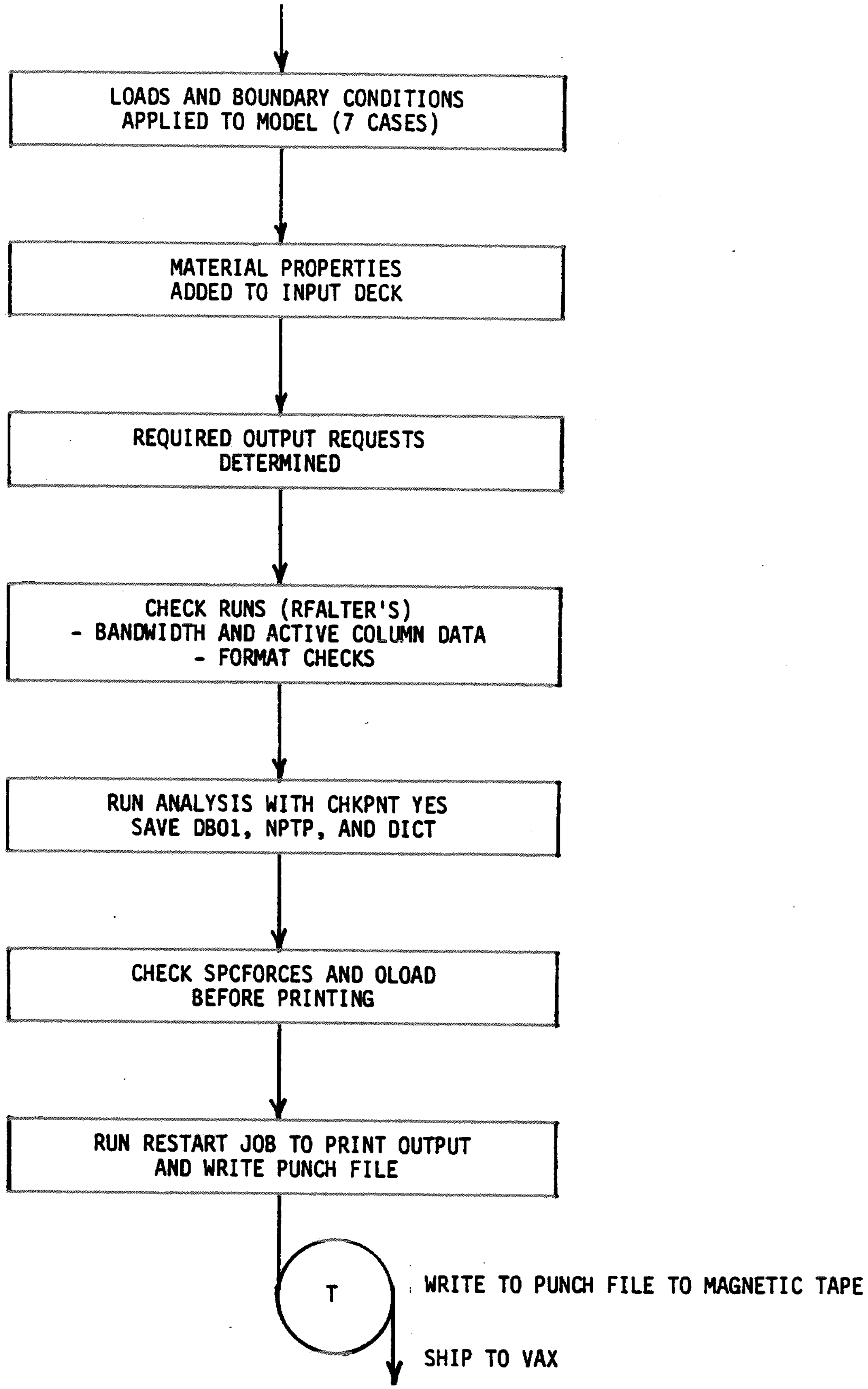


FIGURE 2.1: THE ANALYSIS PROCEDURE (PAGE 2 OF 3)

POST-PROCESSING

VAX 11/780

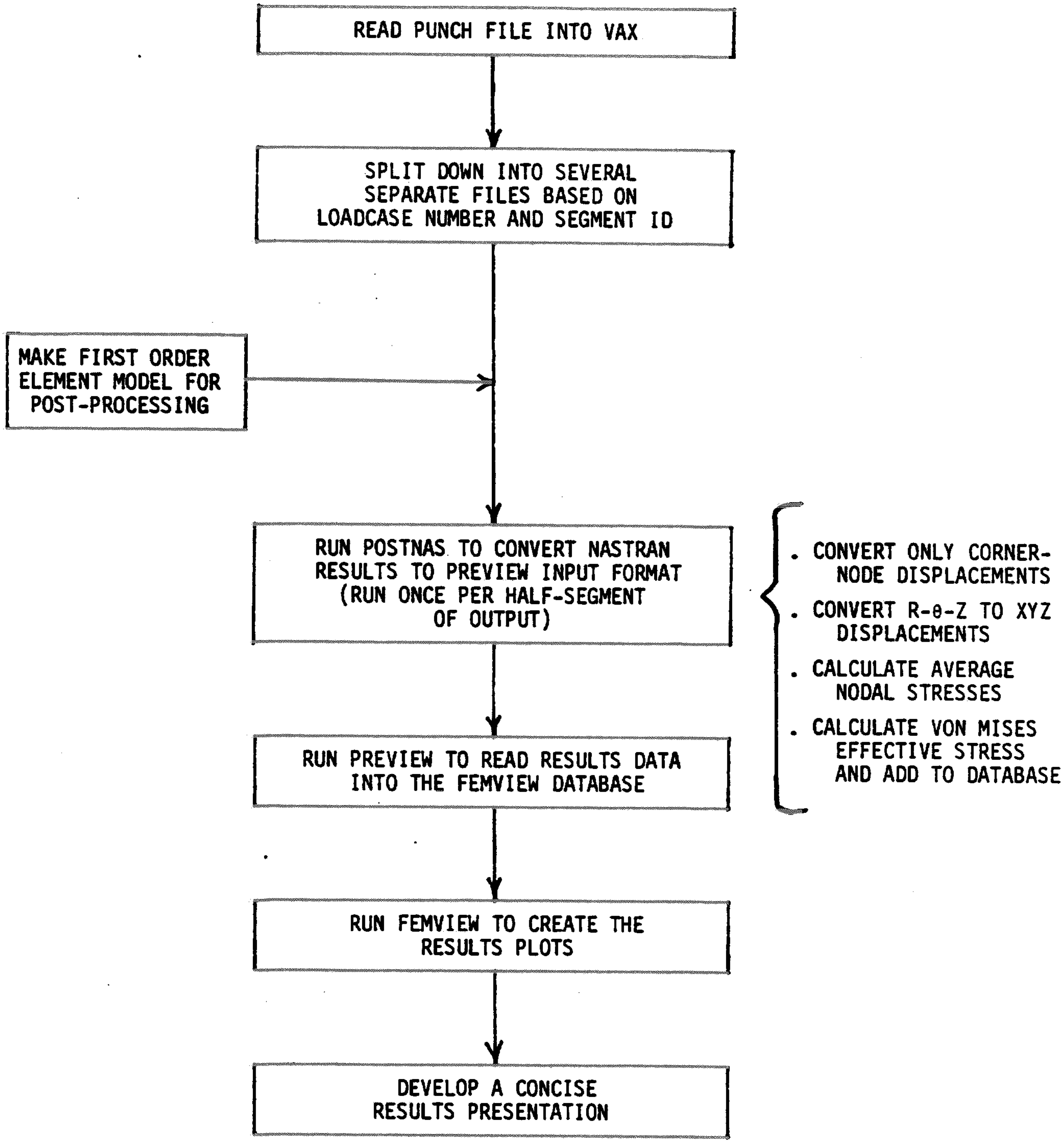


FIGURE 2.1: THE ANALYSIS PROCEDURE (PAGE 3 OF 3)

A. MODEL GENERATION

Using the FEMGEN mesh generator, a finite element model of a one-eighth segment of the rotor hub was created for use with the cyclic symmetry option of NASTRAN.

The model of the main rotor hub itself was composed entirely of second-order isoparametric solid elements. (Twenty-node bricks and fifteen-node wedges.) A total of 402 solid elements were used. This total may be broken down as follows:

- o 345 twenty-node bricks
- o 57 fifteen-node wedges

During mesh generation, the model was converted into the FEMVIEW database in order to use the FEMVIEW full hidden line plot capability. (Full hidden line plots are not available in FEMGEN.) Hidden line plots are especially useful in verifying a complex three-dimensional model. The FEMVIEW plots of Figures 2.2 through 2.4 show the solid element model of the rotor hub from different angles. The different parts of the rotor hub are identified on these figures with the names by which they will be called in this paper.

The mesh refinement of the rotor hub model was selected so that the curved sides of the elements of the finite element model matched the configuration of the actual rotor hub nearly exactly. (FEMGEN's ability to make use of analytical shapes was employed to insure that certain surfaces in the upper arm area actually formed portions of spheres.)

Even with this degree of geometric correctness, the mesh is not so refined that it will be able to predict the peak stresses caused by very localized notch effects. To obtain the very localized notch effects, the

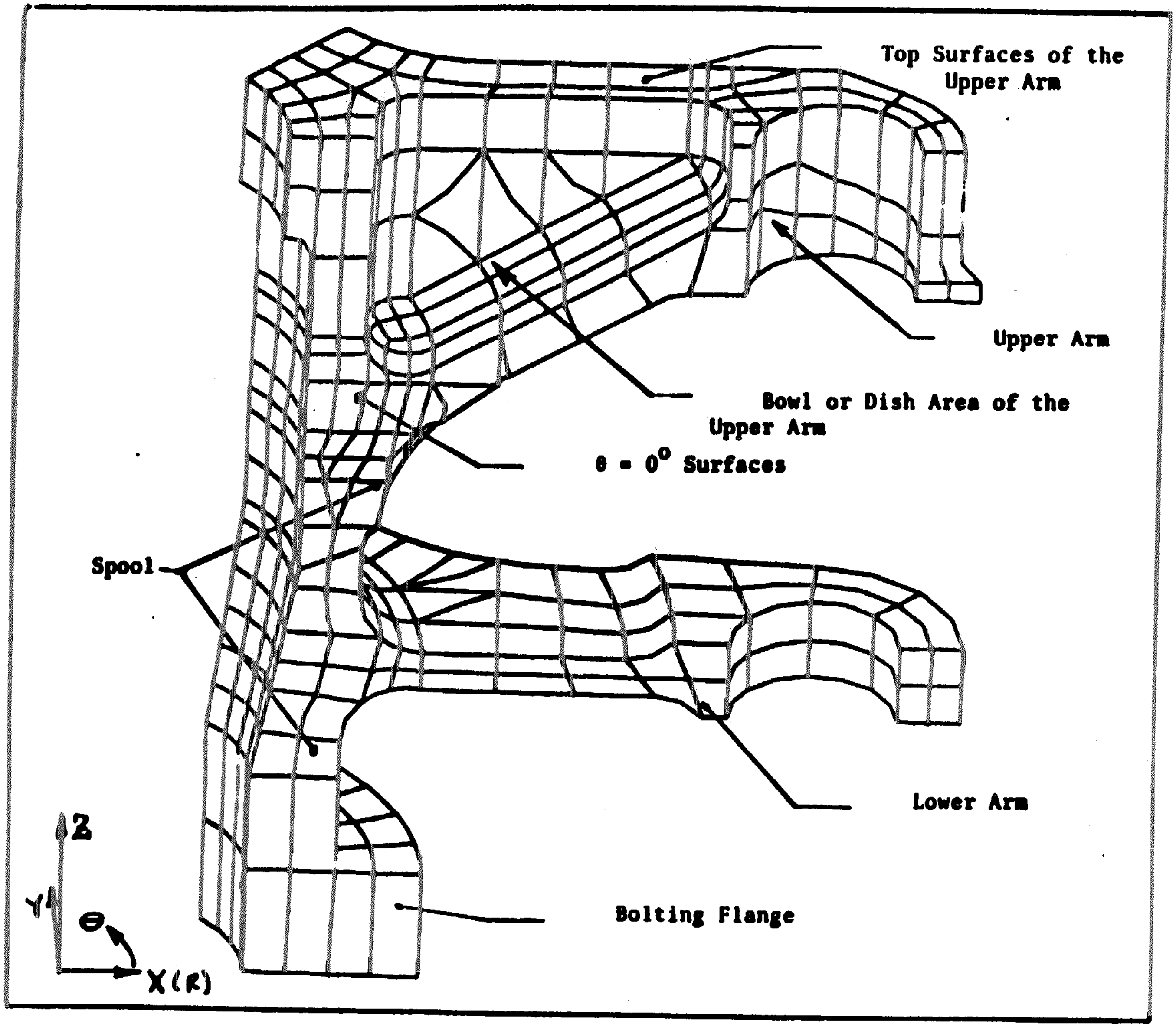


FIGURE 2.2: THE SOLID FINITE ELEMENT MODEL

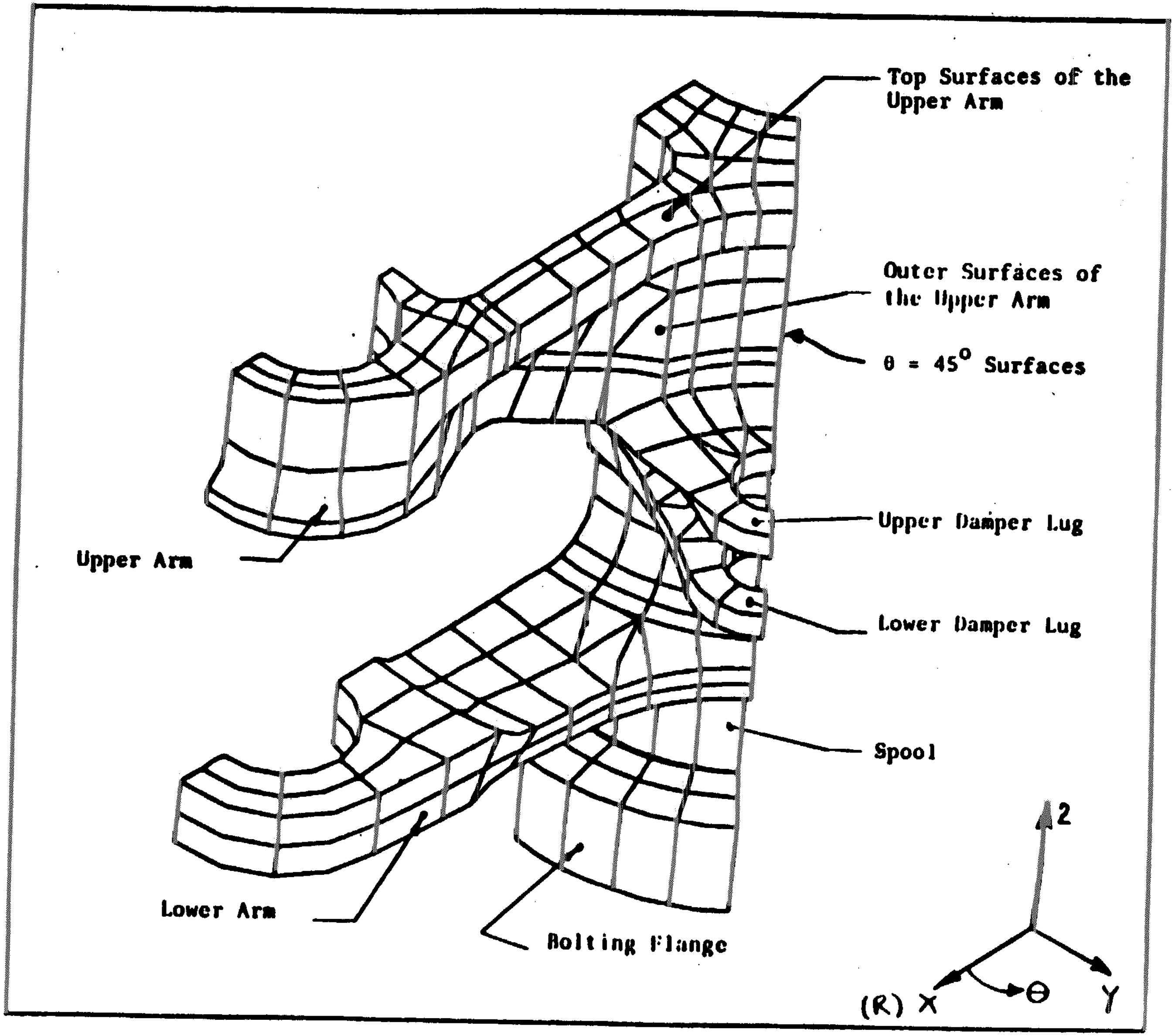


FIGURE 2.3: THE SOLID FINITE ELEMENT MODEL

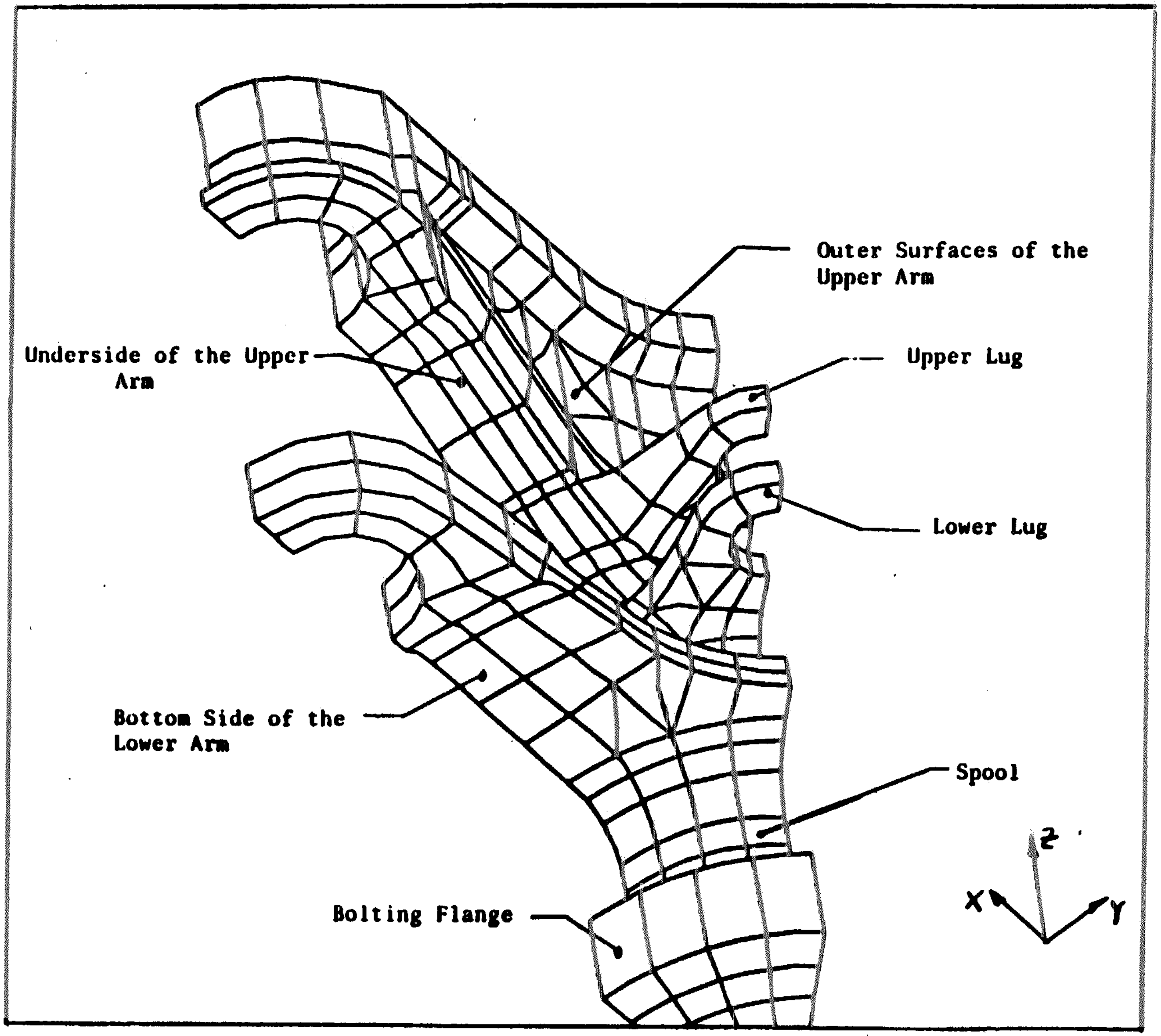


FIGURE 2.4: THE SOLID FINITE ELEMENT MODEL

mesh would have to be extremely well refined. For a cyclic symmetry application with as many loadcases as were analyzed, to increase the mesh refinement enough to pick up notch stresses would require so much time and computer resources as to be economically unfeasible.

A program was developed and linked to the FEMGEN system which performed numerical conditioning checks on the model as it was being created. The program (called VOLUME) performed checks (based on the determinant of the Jacobian) to be sure that every integration point in every element produced a positive contribution to the element volume. The program was also used to calculate the total mesh volume and the global coordinates of the model's center of volume. The total mesh volume calculated for the solid model of the rotor hub was later matched by the NASTRAN calculation.

Once the solid element model of the one-eighth segment had been created in FEMGEN, it was run through a conversion program called PRENAS which converts the nodes and elements into NASTRAN input file format. (The nodes become GRID cards, and the elements become CHEXA or CPENTA cards.) The PRENAS program reads from the FEMGEN database via the FEMGEN User Subroutines. The program was also modified to create the CYJOIN cards needed for side 1 and side 2, and the OMIT Cards used to implement the static condensation.

In order to insure that the conversion from FEMGEN to NASTRAN format was achieved correctly, the VOLUME program was modified to read from a NASTRAN bulk data deck. (The volume and center-of-volume values calculated for the NASTRAN deck should exactly match those determined for the FEMGEN model.) As a further check on the conversion to NASTRAN format, an interface program was written to put the NASTRAN data directly into the FEMVIEW database. The FEMVIEW model plots before and after conversion were then compared.

The rotor blades are attached to the hub by four lead/lag pins which lie parallel to the z-axis and pass through cooperating holes in the upper and lower arms. The pins, the blades, and the hub are clamped together in a manner which imposes continuity of deflection and rotation between the arms and the pins at the holes. Such continuity was provided by the lead/lag pin model shown in Figure 2.5. This model is an assemblage of 21 axial force members (CROD's) and 23 bending elements (CBAR's). (Note that bars and rods lying in planes of symmetry will actually be duplicated during cyclic symmetry operations and as such their properties must be halved.)

When the NASTRAN finite element model was completed, it was written to a magnetic tape and shipped to the CRAY computer installation.

Note On Data Communications:

At the beginning of this analysis project, a communications software package was installed at both data services to allow direct data transfer between the two computers via normal telephone lines. Although the system performed very cleanly and correctly, the transmission rate was much too slow to make the system useful for a file of more than approximately 1000 lines. (A 1000 line test case took nearly two hours to be completely sent from the VAX to the CRAY.) The system was used several times however, to send smaller files containing minor model corrections and small utility programs.

A much quicker communication system is available to achieve this type of data transmission, but would require the installation of some fairly expensive hardware on the VAX. Since magnetic tape operations on both the service bureaus are very simple, and it takes less than 24 hours to send the tape via air courier, the expense of the communications hardware is simply not justified.

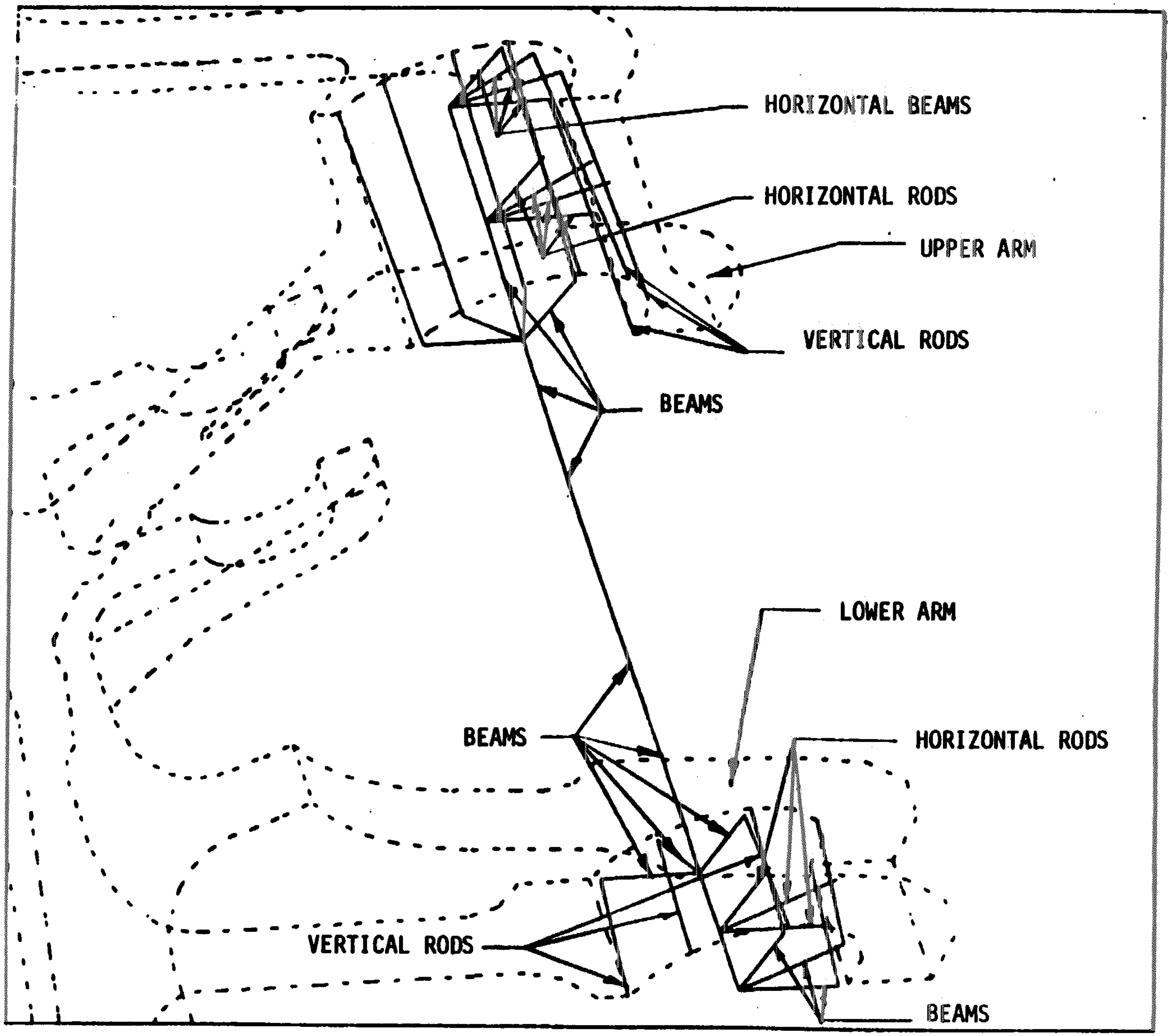


FIGURE 2.5: THE FINITE ELEMENT IDEALIZATION OF THE LEAD/LAG PIN

B. NASTRAN EXECUTION

Once the model had been generated and properly converted to NASTRAN input format, the next step was to specify the loads on the structure and to identify the locations and types of displacement constraints present. The task of correctly specifying the loads for a cyclic symmetry analysis is considerably more complicated than for a normal structural anaysis. This is due to the inherent complexity of performing an analysis to predict the elastic response of the entire structure even though only a small segment of the structure is actually modelled.

When applying a static load to the structure in a cyclic symmetry analysis not only is it necessary to specify the node (or element) to which the load is applied, but it is also necessary to identify which segment of the structure is to be loaded. It is therefore important to understand the segment numbering system used by NASTRAN.

For the dihedral* option of cyclic symmetry which was used for this analysis, the modelled portion becomes the 1R segment. (Segment 1, right half). This is sometimes called the primary or basic segment. NASTRAN will automatically create eight such half-segments to complete the full rotor hub model, and they will be labelled as shown in Figure 2.6. Notice that all the "left" half-segments are simply mirror images of the corresponding right half-segments. As a consequence of being mirror images, the left half-segments have left-handed coordinate systems. Specifically, for our cylindrical coordinate system (R-θ-Z), the R and Z coordinate directions remain unchanged, but the θ direction is reversed. So when viewed down the Z axis, a load in the positive θ direction on an "R" segment would cause a counter-clockwise deformation, while a load in the positive θ direction on an "L" segment would cause a clockwise motion. This aspect of the left segments must be considered when applying loads or interpretting results.

---

*Dihedral Symmetry means that the basic segment must first be mirrored, then the basic segment and its mirror image become the rotationally symmetric segment.

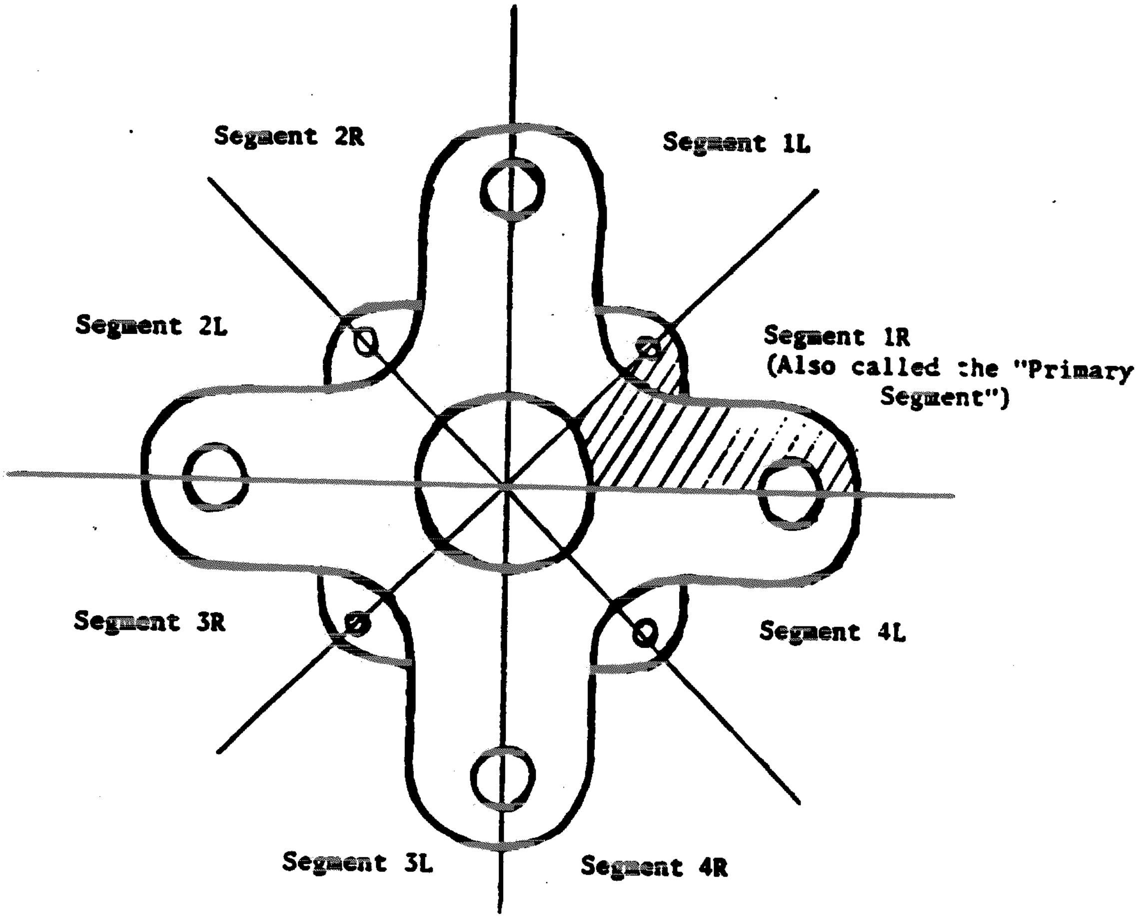


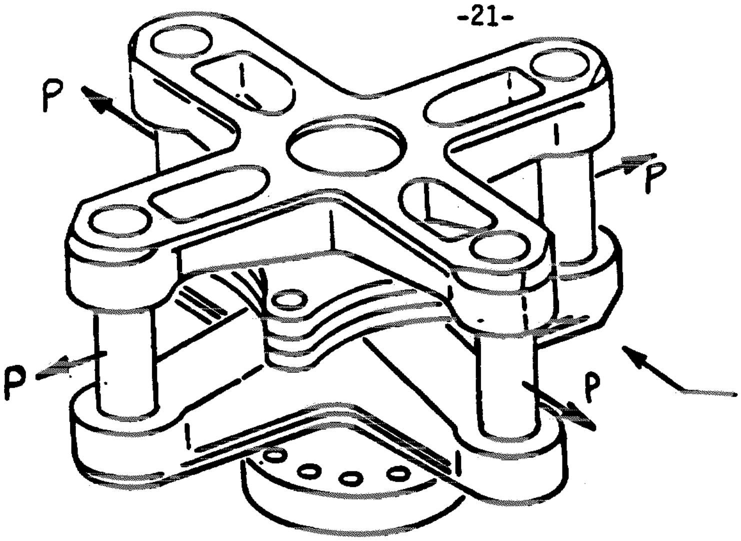
FIGURE 2.6: NASTRAN SEGMENT NAMING CONVENTION

The analysis of the rotor hub was performed for seven different loading conditions. Five of these were individual loadcases and two were combinations involving the first five. The five individual loadcases are depicted in the sketches of Figures 2.7 through 2.11. The sixth loadcase was a factored superposition of the C.F., Drag, Lift, and Hub Moment Loadcases and was called the Limit Load Combination. The seventh loadcase, called the Fatigue Test Steady Combination, was a superposition of multiples of the C.F., Drag, and Lift Loadcases. Since the loaded elements of the lead/lag pin model lie in the plane of reflective symmetry, the total load applied in any segment equals one half of the load per arm. (The load on the 1R segment goes at the same physical location as the load applied to the 4L segment, etc.)

The only nodal constraints applied to the rotor hub model were at the base of the bolting flange. Since a cylindrical displacement coordinate system was chosen to facilitate the use of cyclic symmetry in NASTRAN, the nodal constraints (or SPC's) must also refer to this cylindrical system. The plot of Figure 2.12 shows the node numbers at the base of the bolting flange and identifies the displacement constraints applied.

The Z displacement of nodes 2799 through 2835 was constrained to zero. The remaining nodes at the base of the flange were left free to displace in the Z direction. Such a combination of z constraints simulates the contact surface to mating hardware. Nodes 2813 through 2821 were also constrained against any tangential displacement ($\Delta\theta$) to provide approximate reactions for the drag load case.

Degree of freedom 6 (rotation about the Z axis) was also constrained to zero at one node of the pin model to prevent its rigid body rotation.



ALL LEAD/LAG PINS LOADED
RADIALLY OUTWARD

FIGURE 2.7: C.F. LOAD

ALL LEAD/LAG PINS LOADED
TO PRODUCE CLOCKWISE TORQUE

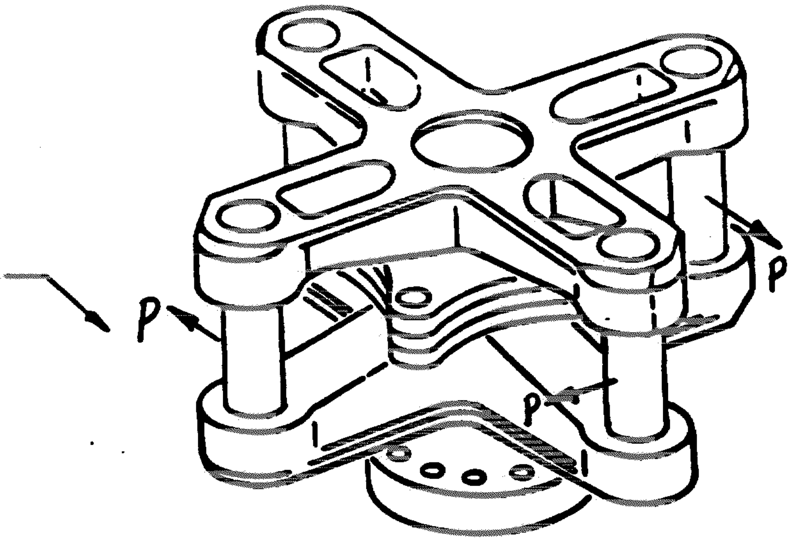
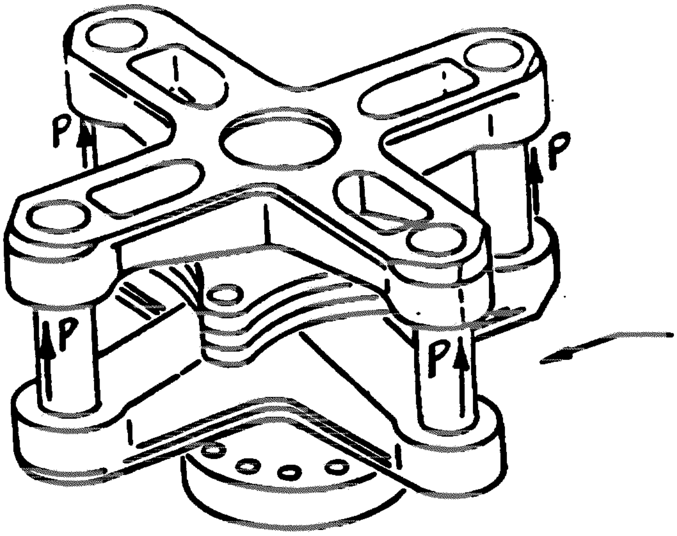
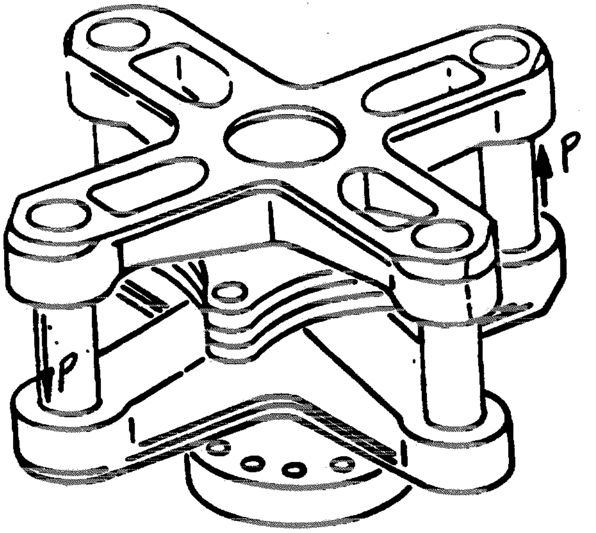


FIGURE 2.8: DRAG LOAD



ALL LEAD/LAG PINS LOADED UPWARD

FIGURE 2.9: LIFT LOAD



- o LEAD/LAG PINS LOADED UPWARD IN SEGMENTS 1R AND 4L
- o LEAD/LAG PINS LOADED DOWNWARD IN SEGMENTS 2L AND 3R

(REMAINING TWO LEAD/LAG PINS UNLOADED)

FIGURE 2.10: HUB MOMENT LOAD

ALL DAMPER LUGS LOADED RADIALLY OUTWARD. LOAD APPLIED DIRECTLY TO LUG ELEMENTS

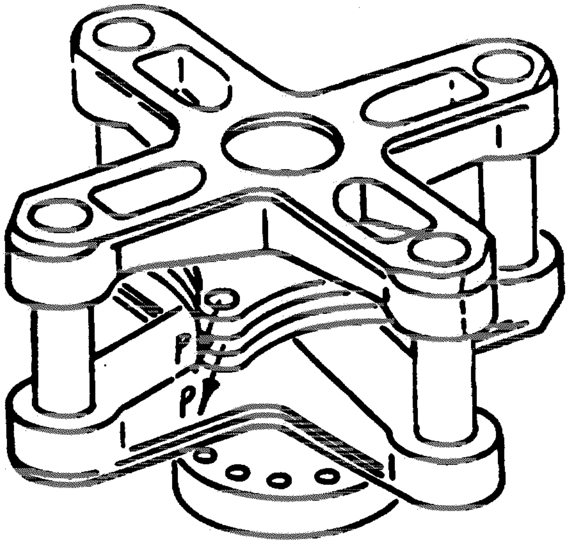


FIGURE 2.11: DAMPER LUG LOAD

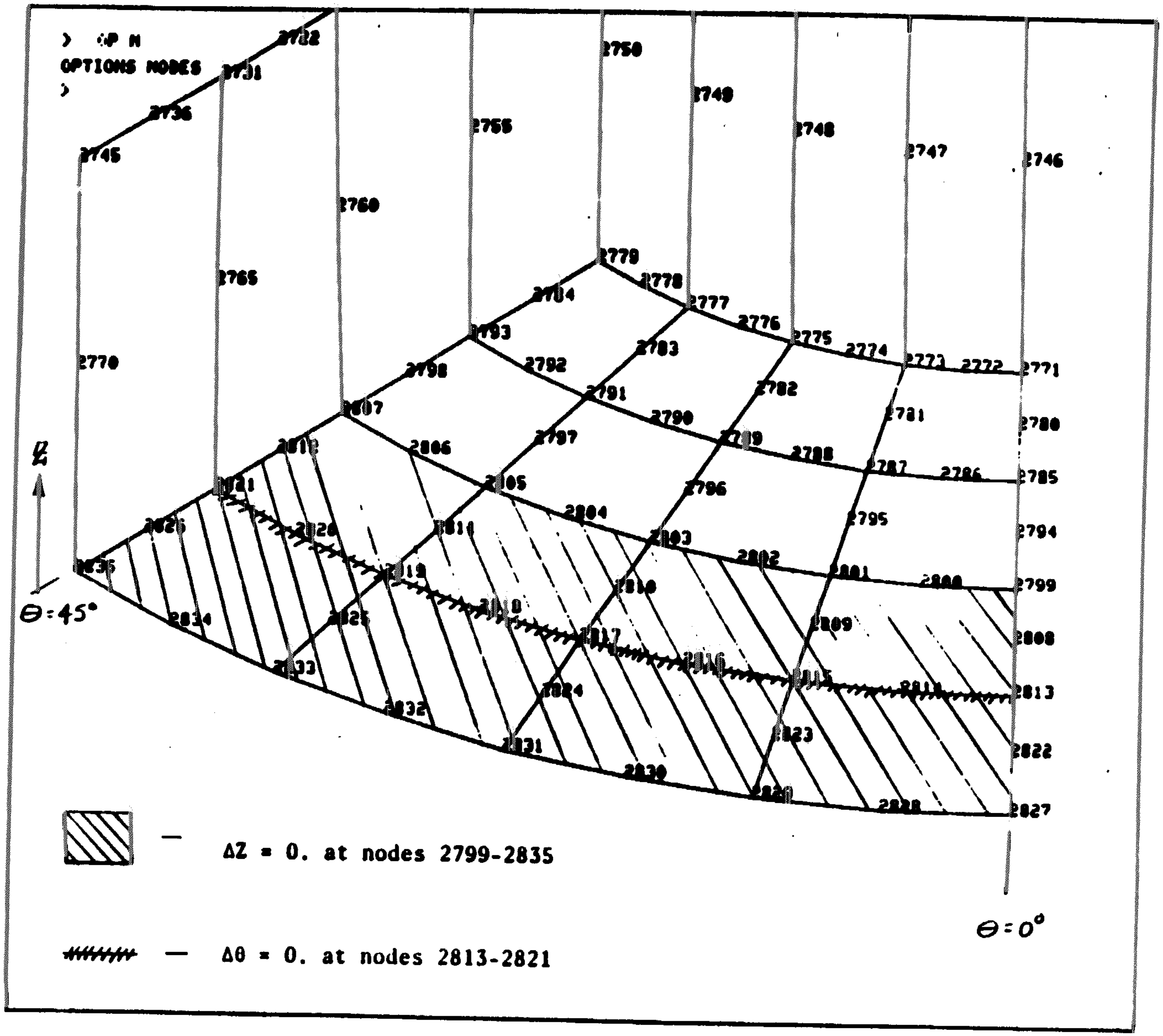


FIGURE 2.12: NODAL CONSTRAINTS AT THE BASE OF THE BOLTING FLANGE

Because of the size and expense of the rotor hub analysis, it was necessary to take several precautions to insure that the analysis would have to be run only once. (There were approximately 7800 degrees of freedom in the one-eighth primary segment.) When the NASTRAN deck had been completely assembled, several check runs (RFALTER's RF24D32 and RF24D74) were performed to verify the format of the input data, and to obtain the bandwidth and active column information from which an estimate may be made of the total computational expense.

The cyclic symmetry analysis performed on the rotor hub provides the opportunity to obtain the stresses and displacements in any of the eight half-segments which comprise the full hub. Due to the large amount of output data which each half-segment produces, it is imperative that output is requested from only those segments for which it is truly required. This also cuts down on the computation required, since back-substitution is performed only on the segments for which output is requested.

Since the symmetry conditions vary from loadcase to loadcase, the output required to characterize the structural response to each loadcase differs accordingly. The segments of output required for each loadcase are summarized as follows:

CF Loadcase:	Since the CF load is fully eighth symmetric, behavior will be identical in every segment. Therefoutput was requested only for the 1R segment.
Drag Loadcase:	The drag load is rotationally quarter symmetric, meaning that all "R" segments will have same results, and all "L" segments will behave the same. Therefore, output was obtained only for the 1R and 1L segments.

Lift Loadcase:	The lift load is eighth symmetric, so only the output from the 1R segment was requested.
Hub Moment Loadcase:	The hub moment is truly only half symmetric, but it may be observed that the output values in the 1R segment will be equal in magnitude and opposite in direction to the values in segment 2L, so no output was requested from the 2L segment. An analogous relationship exists between the 1L and 2R segments, but since these segments represent the unloaded arm, the output values for segments will be lesser in magnitude, and therefore neither the 1L nor the 2L segments were included in the output requests. Output for the hub moment loadcase was requested for the 1R segment only.
Damper Lug Loadcase:	As with the CF and Lift load cases, the damper lug load is eighth symmetric, and as such only the 1R segments results were required.
Limit Load Combination:	Since this combination involved the CF, Drag, Lift, and Hub Moment Loadcases, the maximum output values could occur anywhere in the half symmetric portion. Specifically, it was necessary to obtain output from the 1R, 1L, 2R, and 2L segments.
Fatigue Test Steady Combination:	This loadcase was a combination of the CF, Drag, and Lift loads only, and as such all "R" segments will respond identically and all "L" segments will behave the same. For this reason, output was requested for the 1R and 1L segments only.

The output requested for the seven physical loadcases amounts to twelve half-segments worth of results. (1 for the CF Load, 2 for the Drag Load, 1 for the Lift Load, 1 for the Hub Moment Load, 1 for the Damper Lug Load, 4 for the Limit Load Combination, and 2 for the Fatigue Test Steady Combination.) All of the output requested was obtained in two forms - as printed output and as a results file (or "PUNCH" file) containing all the stresses and displacements. The results file is an essential requirement, since it is the only link to the FEMVIEW program to be used later to facilitate interpretation of the analysis results.

Once the NASTRAN deck had been assembled and thoroughly interogated, the following execution steps were performed:

1. Run the NASTRAN analysis for all seven loadcases. (This was done at the lower, weekend rates.) The analysis created a checkpoint/restart file, a checkpoint dictionary file, and a database file. (The files were so large that several system parameters on the CRAY had to be reset.) Only SPCFORCES were printed during the actual analysis run.

2. Once it was determined that the SPCFORCES obtained from the analysis were consistent with an equilibrium condition, a restart job was submitted to print all the stresses and displacements for all loadcases. This output file (approximately 5000 pages) was printed on a high-speed line printer at the CRAY installation. Printing at the service bureau turned out to be less expensive and faster than trying to print on the JAR line printer. The output was printed on two part paper, decollated by machine, and then shipped via Air Express to the JAR office.

The restart job also created the results PUNCH file to be sent to the VAX for post-processing via the FEMVIEW Program. The results file created in this way contained approximately 30 million characters (approximately 364,000 lines).

The NASTRAN execution was done in two steps for two reasons. First, the output file would be large and expensive to print. Thus, it was desirable to establish that a correct execution had been achieved before printing. Secondly, it is good practice to CHKPNT so large an analysis, and the Checkpoint dictionary file is written to the same logical unit as the system punch file (FT07). This second reason indicates that a PUNCH file cannot be easily created during an execution with CHKPNT YES.

Before the PUNCH file containing the analysis results could be sent to the VAX it first had to be converted from internal CRAY format and written to an unlabelled, ASCII, blocked tape with fixed-length 80 byte records. It is very important to make all these conversions so that the VAX can read the tape successfully. This tape was sent via air express to the VAX and was received the following day.

C. RESULTS PROCESSING AND PRESENTATION

In order to evaluate the structural response of the hub, the FEMVIEW interactive results viewing program was employed. FEMVIEW allows the creation of stress contour plots, displaced shape plots, and other graphical representations of the structural behavior predicted by the NASTRAN analysis. Use of the FEMVIEW program drastically reduces the time required for results interpretation. (The NASTRAN analysis produced approximately 5000 pages of printed output.)

In order to use FEMVIEW to process the analysis results the data from the CRAY computer used for the NASTRAN analysis had to be transferred to the VAX computer where FEMVIEW is available. The results data then had to be manipulated to create an appropriate FEMVIEW database. The task of getting the NASTRAN results into the FEMVIEW database involved these following steps:

1. The magnetic tape containing the NASTRAN PUNCH file with the analysis results was read into the VAX.

2. A program was written to split the one large punch file (approximately 30 million characters) into several smaller files.

3. Since NASTRAN produces stress results only at the corner nodes of second-order solid elements, the finite element model was converted to a model containing only first-order elements (eight-node bricks and six-node wedges). This first-order model was used only for post-processing in FEMVIEW. (See Figure 2.13.)

4. A program was written, called POSTNAS, to read from the smaller punch files and create a data file in a format readable by the PREVIEW program. (PREVIEW is the program which actually puts information into the FEMVIEW database.) In

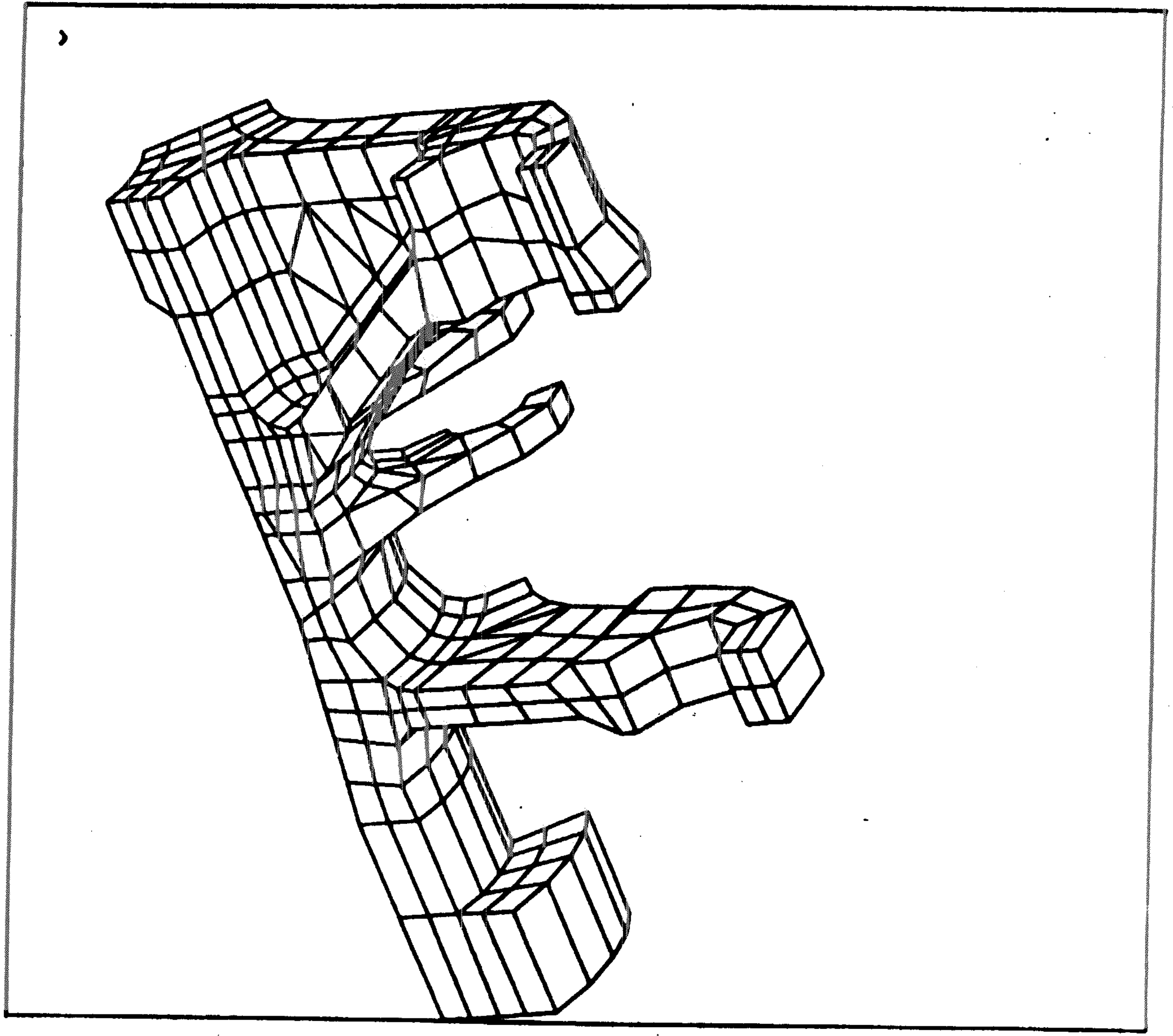


FIGURE 2.13: FIRST-ORDER ELEMENT MODEL USED FOR POST-PROCESSING

addition to converting the data to PREVIEW input format, the POSTNAS program also had to perform several intermediate steps, some of which are listed below:

- The program had to translate the nodal displacements from the R-θ-Z coordinate system necessary for NASTRAN cyclic symmetry to the X-Y-Z system which FEMVIEW requires.

- Only corner-node displacements could be written to the PREVIEW file since the finite element model used for FEMVIEW processing is made up of corner-node elements only.

- The program had to calculate average nodal stresses. NASTRAN outputs stresses at node points on an element by element basis. There will, therefore, be as many values for a particular stress component at a node as there are elements connected to that node. Since FEMVIEW can accept only one unique value at each node, an average nodal stress had to be calculated from the NASTRAN data. (This was done by adding all the stress values at a node and dividing the sum by the total number of values.)

- Since the Von Mises effective stress will be the attribute most commonly plotted, the program used the average nodal stress tensor at each node to calculate the Von Mises stress and this value was added to the FEMVIEW data-base. The Von Mises effective stress is an invariant of the stress tensor defined as the following algebraic combination of the stress components [5]:

$$\sigma_e = \frac{1}{2}\sqrt{(\sigma_{xx}-\sigma_{yy})^2+(\sigma_{yy}-\sigma_{zz})^2+(\sigma_{zz}-\sigma_{xx})^2+6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz}^2)}$$

The Von Mises effective stress, σ_e, is very useful since, in terms of the principal stresses (σ_1, σ_2, σ_3):

$$\sigma_e = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}}$$

When a node on an unloaded edge of the model is considered, only one principal stress is non-zero, and it lies along the edge. In this specific case:

$$\sigma_e = \sigma_{principal}$$

Processing the results of a cyclic symmetry analysis with a program such as FEMVIEW presents the analyst with an interesting problem: How to graphically post-process results data produced by NASTRAN for parts of the structure which are not physically modelled. When the dihedral option of cyclic symmetry is used, the matter is further complicated, since some results are produced for "L" segments, which are mirror images of the physically modelled segment.

For the rotor hub analysis, there were only seven physical loadcases analyzed, but several of the loadcases produced unique results for more than one segment. For instance, the limit load combination case produced

FEMVIEW		PHYSICAL LOADCASE DESCRIPTION	NASTRAN	
LOADCASE NUMBER	LOADCASE NAME		SUBCASE NUMBER	SEGMENT ID
1	CF1R	C.F. LOAD	1	1R
2	DRAG1R	DRAG LOAD	2	1R
3	DRAG1L		2	1L
4	LIFT1R	LIFT LOAD	3	1R
5	HUBM1R	HUB MOMENT LOAD	4	1R
6	DAMP1R	DAMPER LUG LOAD	5	1R
7	LLCM1R	LIMIT LOAD COMBINATION	6	1R
8	LLCM1L		6	1L
9	LLCM2R		6	2R
10	LLCM2L		6	2L
11	FTSC1R	FATIGUE TEST STEADY COMBINATION	7	1R
12	FTSC1L		7	1L

TABLE 2.1: LOAD CASE DESCRIPTORS

output for a full half of the Rotor Hub (4 segments). (Indeed, this is the entire purpose of performing a cyclic symmetry analysis.) In order to process the results on the single segment which was actually modelled, each segment of output was treated by FEMVIEW as a separate loadcase. With the NASTRAN cyclic symmetry results organized in this way, the FEMVIEW database contained twelve separate loadcases. The relation between the FEMVIEW loadcases and the NASTRAN loadcases is summarized in Table 2.1.

The results from all these loadcases (stress contours, etc.) were plotted on the same finite element model of a one-eighth segment of the hub. The finite element model is of a right half segment, so when viewing results of left half segments, one must keep in mind that he is viewing a mirror image of the model. Viewing the results of "L" segments this way is not particularly difficult, and was deemed far simpler than creating a complete mirror-image FEMVIEW model simply for post-processing. In certain respects it is actually more effective to display "L" results on an "R" segment, since this allows the viewer to think in terms of the usual right-handed coordinate system.

Once the FEMVIEW database had been created with all the loadcases included, the task of producing graphic displays of the results began. More than 400 results plots were prepared to assist in the interpretation of the stresses and deformations in the rotor hub.

The analyst is confronted with a significant challenge in the presentation of such a large amount of results data. Indeed, it is pointless to produce great amounts of analytical results, if these results are not presented to the customer in a clear, concise manner which will enable him to make well-informed engineering decisions.

In order to present the results data in an effective manner, a special system for numbering the result figures was established. Each results figure produced by FEMVIEW was given a unique two-part number which identifies the type of information to be found on that plot. The results figure numbers were of the form:

LN - PID

where:

LN is the FEMVIEW Loadcase number, which identifies the physical loadcase and the segment of the hub involved.

PID is the Plot ID, which identifies the specific region of the model and the result attribute being examined.

The relationship between the FEMVIEW loadcase numbers and the physical loadcases in the NASTRAN analysis was previously summarized in Table 2.1. The plots with a FEMVIEW loadcase number of zero correspond to the node and element numbers in the region indicated by the Plot ID.

This numbering system proved useful in that all figures with the same Plot ID were plots of the same result attribute in the same region of the model - only the FEMVIEW loadcase was different. In this way it was simple to compare results from one loadcase to another. (For instance, let's say the analyst noticed, from Figure 5-31, a large value of σ_{XX} caused by the hub moment load on the top surface of the lower arm. If he wished to find the values of σ_{XX} in those same locations caused by the C.F. Load, he would look at Figure 1-31. If he needed to know the node and element numbers in that area, he would look at Figure 0-31.)

The Plot ID numbers ranged from 1 to 76, although most of the loadcases had far less than the possible 76 plots. With this numbering system, it was not necessary to type lengthy titles on each of the 420 plots included in the final analysis report. The figure number itself, along with the explanations of each of the seventy-six basic plots, was enough to completely describe any plot.

The results plots were presented in their entirety, in an appendix of the analysis report. Some typical plots are included in this paper as Figures 2.14 through 2.18.

Along with the many FEMVIEW results plots, the final report to the customer included a table summarizing the Von Mises effective stress values at some key locations of the rotor hub. The results data was tabulated for the 1R segment of the Hub Moment Loadcase and for all requested output segments of the Limit Load Combination and the Fatigue Test Steady Combination.

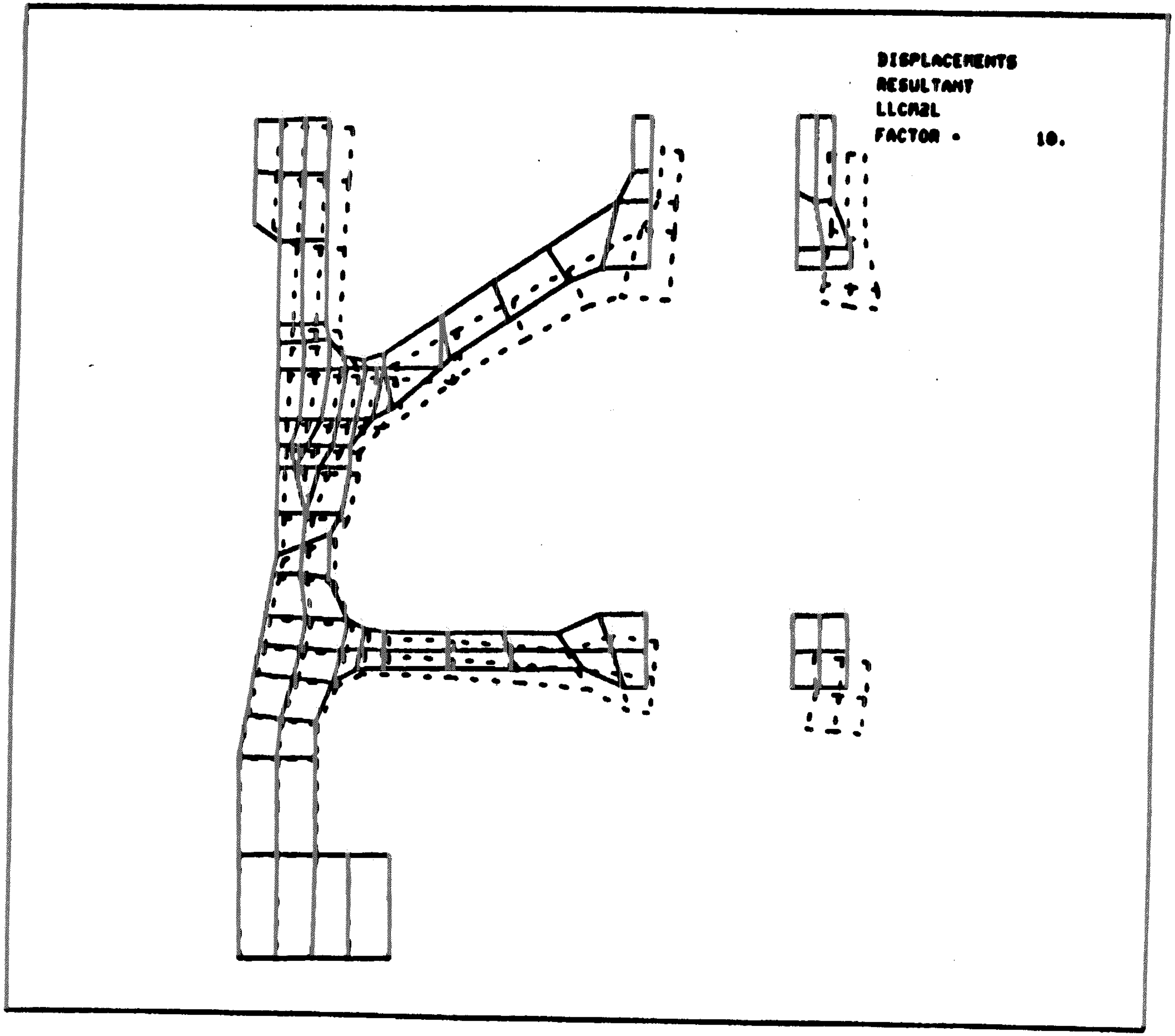
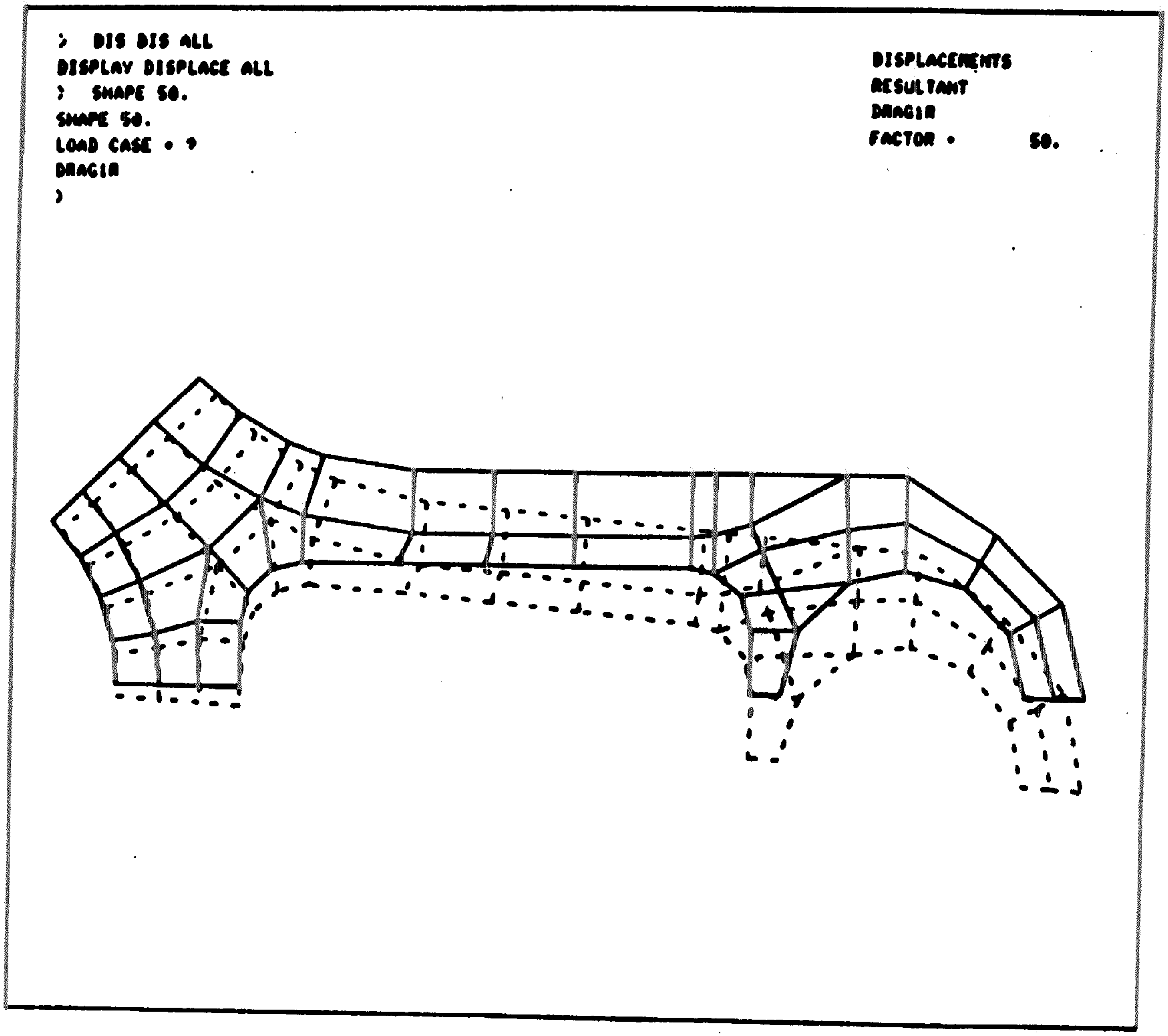


FIGURE 2.14: DISPLACED SHAPE, $\theta=0^\circ$ PLANE.
LIMIT LOAD COMBINATION, SEGMENT 2L.
(REPORT FIGURE 10-12)



```
>  DIS DIS ALL
DISPLAY DISPLACE ALL
>  SHAPE 50.
SHAPE 50.
LOAD CASE = 9
DRAG1R
>
```

```
DISPLACEMENTS
RESULTANT
DRAG1R
FACTOR =        50.
```

FIGURE 2.15: DISPLACED SHAPE, X-Y PLANE.
DRAG LOAD, SEGMENT 1R.
(REPORT FIGURE 2-75)

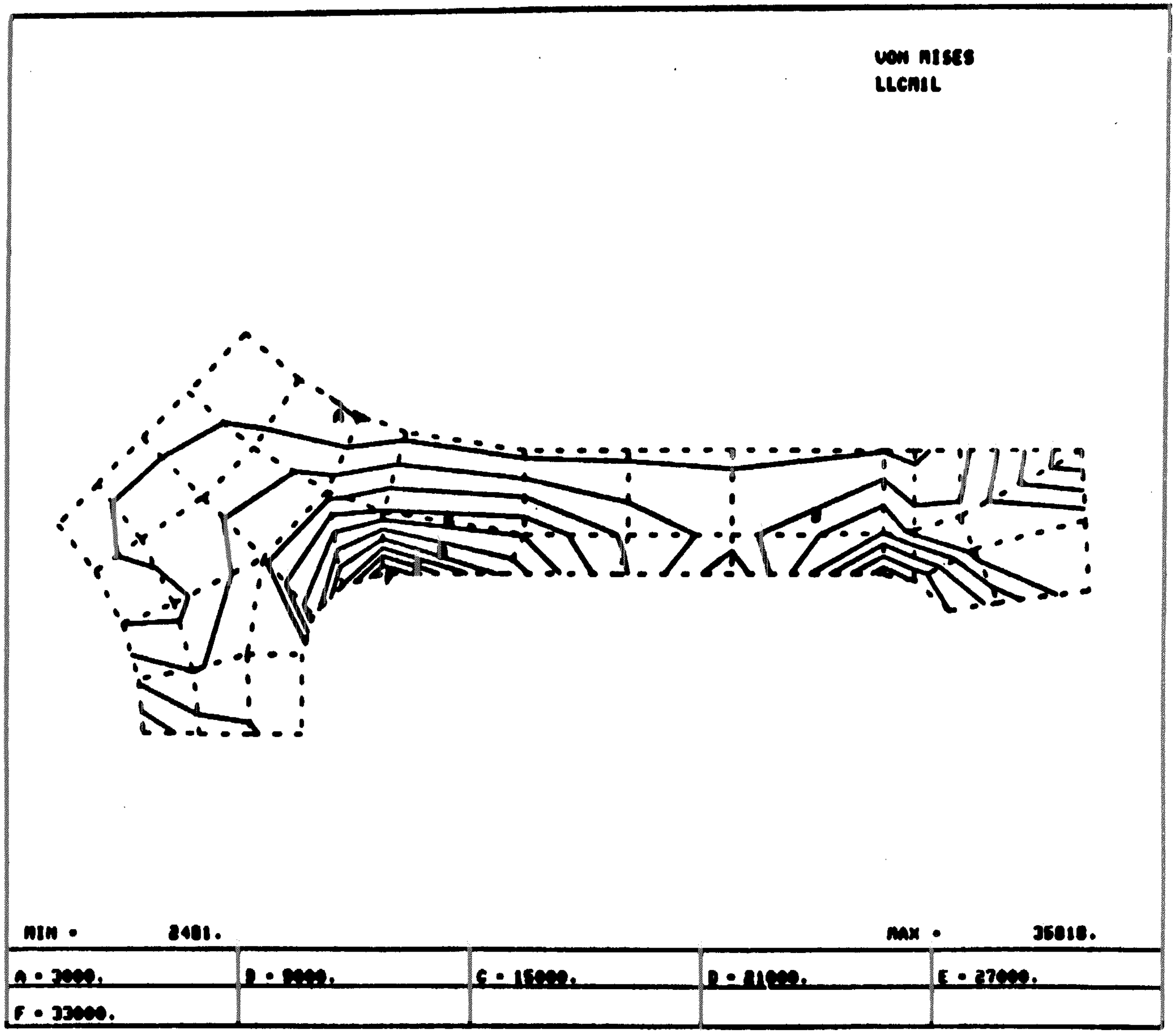


FIGURE 2.16: VON MISES CONTOURS ON TOP SURFACE OF UPPER ARM.
LIMIT LOAD COMBINATION, SEGMENT 1L.
(REPORT FIGURE 8-3)

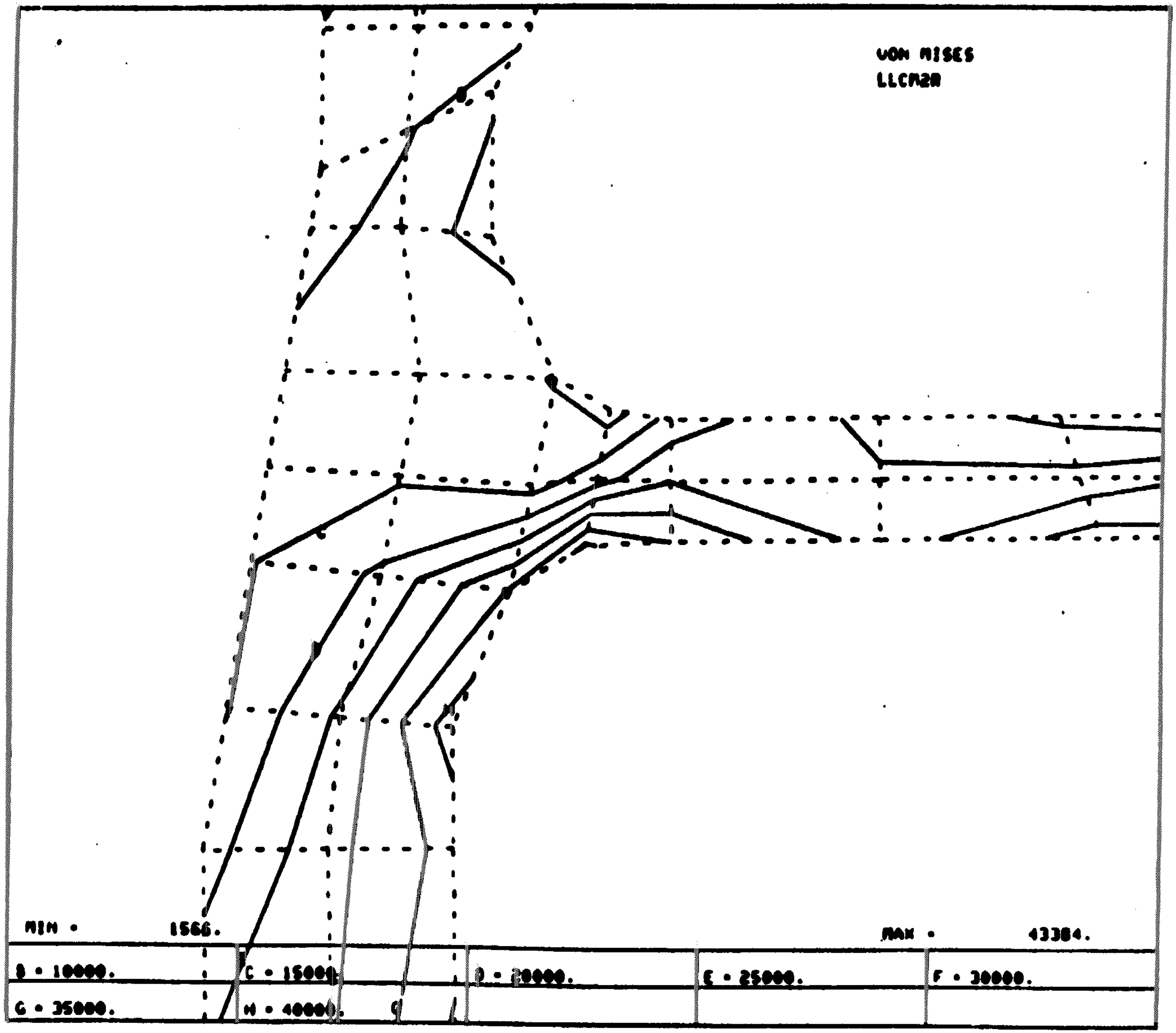


FIGURE 2.17: VON MISES CONTOURS ON THE $\theta=0^{0}$ SURFACES NEAR INTERSECTION WITH LOWER ARM. LIMIT LOAD COMBINATION, SEGMENT 2R. (REPORT FIGURE 9-14)

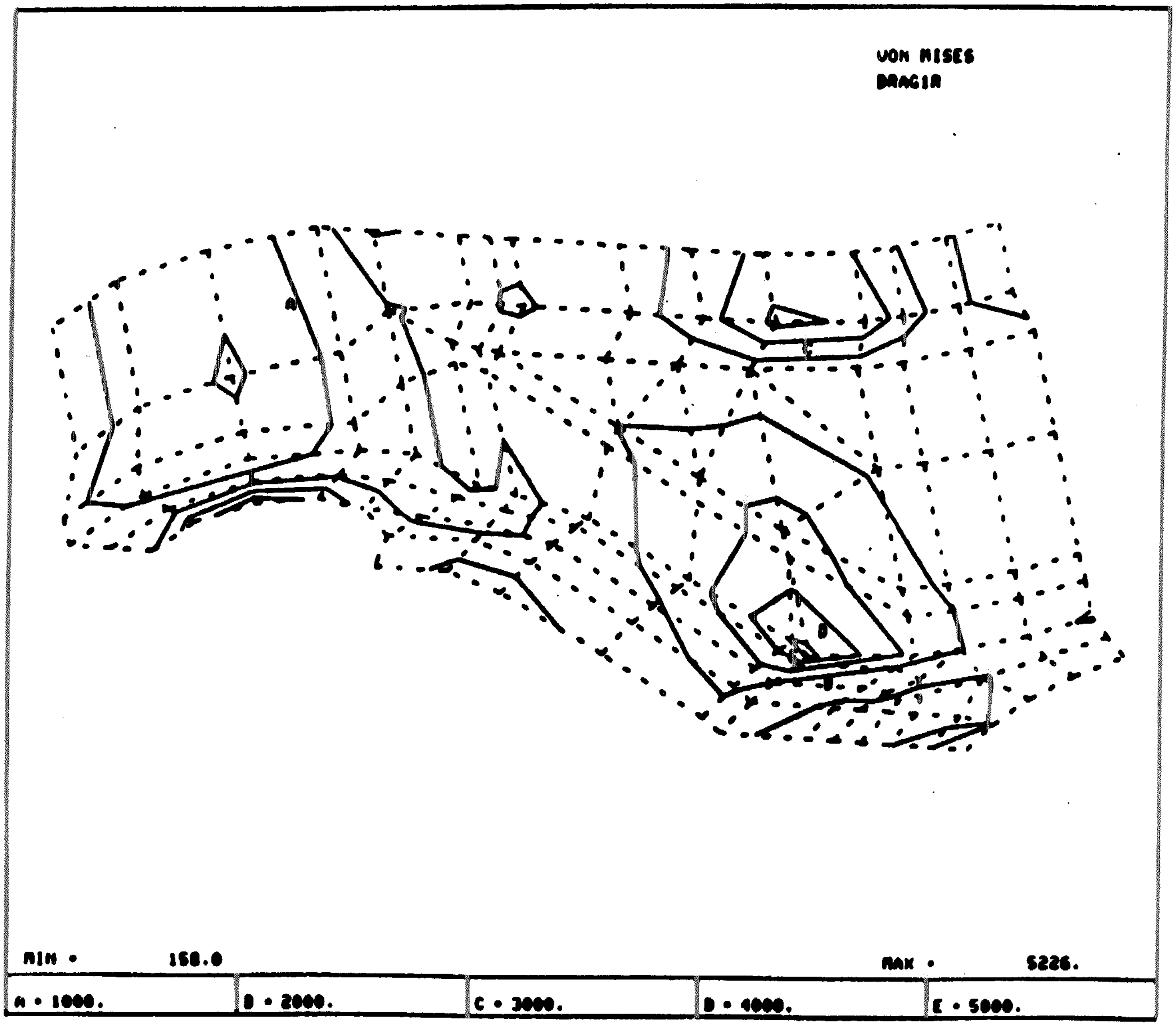


FIGURE 2.18: VON MISES CONTOURS ON THE OUTER SURFACES OF THE UPPER ARM. DRAG LOAD, SEGMENT 1R. (REPORT FIGURE 2-47)

III. SUMMARY

A three dimensional finite element analysis of the SH-2F Main Rotor Hub was performed by JAR Associates, Inc.. The FEMGEN interactive graphics mesh generation program was used effectively to create the one-eighth symmetric finite element model of the rotor hub required for the analysis. The overall structural response of the hub to several different loading conditions (vibratory and steady) was successfully predicted using the dihedral cyclic symmetry option of MSC/NASTRAN. Static Condensation was performed on the primary segment of the hub to reduce the total computational expense. The results were presented graphically as an extensive group of plots created by the FEMVIEW interactive results viewing program. The FEMVIEW plots were an integral part of an effective results presentation delivered to the customer.

With the structural behavior presented in this form, the customer was able to make efficient use of the analytical results, and as such the finite element analysis had a significant and positive impact on the new design of the main rotor hub.

REFERENCES

1. Johansson, Thomas, FEMGEN - A General Finite Element Mesh Generator, User Manual, 1980.

2a. MacNeal, Richard H., The NASTRAN Theoretical Manual, Scientific and Technical Information Office, NASA, 1972.

2b. McCormick, Caleb W., Editor, MSC/NASTRAN Version 62 User's Manual, MacNeal - Schwendler Corporation, 1982.

2c. Joseph, Jerrard A., Editor, MSC/NASTRAN Application Manual, MacNeal - Schwendler Corporation, 1982.

2d. Schaeffer, Harry G., MSC/NASTRAN Primer: Static and Normal Modes Analysis, Wallace Press, Inc., 1982.

3. Guyan, R.J., "Reduction of Stiffness and Mass Matrices", AIAA Journal, Volume 3, No. 2, pp. 526, 1965.

4. Warde, Simon, FEMVIEW Version 2.0 User Manual, Simon Warde Associates, Ltd., 1981.

5. Cook, Robert, D., Concepts and Applications of Finite Element Analysis, John Wiley and Sons, Inc., 1978, pp. 293.