N74-14823

AN INTERACTIVE GRAPHICS SYSTEM TO FACILITATE FINITE ELEMENT STRUCTURAL ANALYSIS

BY ROBERT C. BURK AND FRED H. HELD
MCDONNELL DOUGLAS ASTRONAUTICS COMPANY - EAST

SUMMARY

Industry's growing use of finite element structural analysis requires that an increasing portion of the engineer's time be spent in building, checking, executing, and interpreting results of finite element models. The following discussion explains the use of a rapid, inexpensive, graphically oriented system for performing this job. With much of the bookkeeping drudgery removed and the visibility of results enhanced, the inspiration/perspiration ratio of the engineer is significantly improved.

INTRODUCTION

The effectiveness of a finite element analysis depends on the accuracy with which the model represents the actual structure and on the time and money spent to build, solve, and interpret the analysis results. Usually, finite element analysis involves the generation and manipulation of large quantities of data by hand using up valuable engineering time and introducing many opportunities for human errors. A typical analysis would normally require the following steps:

1. Idealize the actual structure into discrete elements.

2. Make sketches of the idealized model and label with node and element numbers for use in referencing element properties, applied loads, reactions, output results, etc.

3. Fill out data sheets.

4. Have data sheets keypunched.

5. Obtain listing of deck and check for any errors (keypunch or coding).

6. Make appropriate corrections.

7. Obtain batch plot of structure to check for incorrect element connectivity or node location.

8. Submit for batch solution.

9. Evaluate results for any errors. Often, several computer runs (with their corresponding costs and turnaround delays) are required before error-free output is obtained.

10. Obtain batch plots of deformed shape.

11. Make freebody sketches of components showing reactions and internal loads. These are used for detailed stress analysis and formal reports of the analysis.

Each of these steps takes time and allows considerable chance for error.

In order to minimize the time spent on each of the steps, to reduce the chance for error, and to enhance the understanding by the engineer, interactive graphics is being harnessed. Reference 1 presents an excellent review of the uses of interactive computer graphics. In the area of structural analysis computer graphic applications, a system developed for two-dimensional modeling and display of results is discussed. This has been implemented on both the UNIVAC 418 - DEC 340 and the IBM 360/50-IBM 2250. Reference 2 presents a system developed by the Jet Propulsion Laboratory using interactive graphics for three-dimensional model checking.

This paper presents a rapid, low cost system which uses interactive graphics in both the preprocessing and postprocessing of finite element data. It can be used with the NASTRAN, ICES STRUDL, and CASD (Computer Aided Structural Design -- an in-house developed and used program) finite element analysis programs to minimize modeling errors, reduce the time required for the design/analysis cycle, and maximize the visibility of results.

A PROMISING SOLUTION

The system being effectively used by the McDonnell Douglas Astronautics Company, Eastern Division, places the engineer at an interactive graphics terminal. He can build, check, edit, solve, and interpret the results of a finite element model static analysis without leaving his chair. Many useful alternatives are controlled by the user at his remote terminal.

The basic hardware employed is an inexpensive, semiportable, Computek interactive graphics terminal with attached hard copier and digitizing tablet (see Figure 1). The terminal is a teletype compatible device with a cathode ray tube (CRT) that can transmit and display both alphanumerics and graphics. A hand held stylus is used to identify point coordinates on the digitizing tablet. These coordinates along with the status of three push buttons and two switches are transmitted to the computer for processing. The stylus position on the tablet is tracked on the CRT, allowing the user to coordinate his input with the displayed image. The hard copier makes 8 1/2 by 11 inch copies of the current image on the screen (in approximately 10 seconds for about 6 cents each) for documentation. The terminal communicates with our XDS Sigma 7 conversational, direct access computer via a standard telephone. Thus, the user may locate his terminal anywhere electrical outlets and a telephone exist.

A finite element model analysis can be broken into three phases: preprocessing (model generation), problem solution, and postprocessing (interpretation of results). The use of our system will be explained in each of these phases and demonstrated with an example problem. The problem involves determination of internal loads and deflections for a swept, multi-cell wing structure subjected to a simplified landing condition.

PREPROCESSING

Preprocessing includes building, checking, and editing a finite element model to prepare it for solution. The preprocessing phase is schematically represented in Figure 2. There are several methods we use to build a finite element model. The most common method is to model in the interactive graphics mode using the finite element modeling (FEM) program. Alternately, the data can be entered in card format directly into an on-line file using a conversational terminal and the computer's editor system. The model data may also be entered onto data sheets, keypunched, then loaded into an on-line file on the computer. In many cases, a small special purpose computer program is used to generate sections of the desired model where extreme accuracy is required or where structural geometry is very repetitious. For all methods, the FEM program is used to display the model for checking and to make any additions or necessary corrections.

When using the FEM program, node geometry is normally digitized directly to the computer utilizing actual scale drawings or layouts of structural cross sections as shown in Figure 3. Optionally, points can be keyed in (to obtain a more accurate location) or specified as vertical or horizontal from previously defined nodes. Bar elements may be indicated between these nodes while in this two-dimensional mode. The wing problem was entered using the parallel rib stations as the entry planes. A typical CRT display of one of these sections, as copied by the attached hard copier, is shown in Figure 4. The type of operation performed is controlled by the menu items on the CRT and the position of the buttons and switches on the digitizing tablet. For example, after the first of the two similar root sections was idealized, the menu item "DUP ALL", along with the appropriate Y station value of the second station, was indicated. This is done by moving the hand-held stylus across the tablet until the cursor on the screen is in the target "O". The stylus is then depressed on the tablet which sends the tablet coordinates of this point to the computer. The computer decodes this into a selection of the appropriate menu item. For the "DUP ALL" item, the computer responds with a prompt on the screen to enter the new plane station. After these data are typed in from the keyboard, all elements in the current plane are duplicated at the newly defined plane. The other rib stations are entered by using the "NEW CUT" option on the menu and are digitized as was the first rib.

After building all cross sections, a three dimensional display is obtained by indicating the "EXIT" item of the "2-D Node Building Mode" menu. Control is then transferred to the "3-D Element Building Mode". The first activity in this mode is the display of the three-dimensional projection of the current model in the last defined orientation with a new menu along the right hand edge of the screen. This display is shown in Figure 5 for the example problem. All remaining connection elements are now added. This is accomplished by indicating the appropriate element type from the menu, and then indicating the corresponding nodes on the model. Representations of the elements (bars, bending bars, shear panels, triangular plates, and quadrilateral plates) are shown immediately on the CRT as they are generated, thus providing graphic assurance of model correctness. Many options are available in this mode and are displayed as a menu along the right hand side of the screen. One such option, "WINDOW", allows magnification of the specified portion of the display for ease in viewing or inputing data in complex areas. This display is obtained by indicating the menu item, then the lower left and upper right corner of the desired view. The screen is erased, the indicated portion of the model is rescaled to fill the screen, and the display redrawn. A companion menu item, "PAN", allows the operator to move the center of the window to a new point on the structure (at the same scale factor) as indicated by the location of the cursor. Thus, he can effectively pan across the structure in discrete steps at a magnified scale. Additional menu items allow the input of reaction and applied load vectors, load magnitudes (for up to six conditions), and symmetry plane (if any). The example problem, with reaction and force vectors at this stage of completion, is shown in Figure 6.

The problem is then stored in a data file in the appropriate format for one of the following three computer codes: NASTRAN, ICES STRUDL, or CASD. A set of standard program control "cards" and default value element properties are automatically inserted into the data file by the program.

On some models, we have saved 80 percent of the time over the old method of submitting tabulated data for keypunching. The example problem took 46 minutes of "clock on the wall time" and expended approximately 7 dollars of computer costs to bring it to this stage of completion.

At this point in the idealization, a listing of the data file is typically obtained on either a time-share alphanumeric terminal or a high-speed batch printer (See Figure 2) and appropriate solution commands and element properties edited from the default values, these same data can be edited for dynamic analysis (e.g., add inertia matrix) and solved for mode shapes and frequencies using the computer code consistent with the data format. Additional preprocessing programs can be run for such functions as adding the appropriate job control language header cards and performing additional data checks. For large, complex models, the geometry can be plotted on a remote plotter to obtain larger copies of the structural idealization. At the end of these operations, the completed model is ready for solution. Portions of the NASTRAN card image data file generated for the example problem are shown in Figure 7.

Having formulated the input data using preprocessing programs, such errors as "key punch errors", data in wrong card columns, incorrect node number reference, etc., are largely eliminated and the probability of a successful solution on the first try is greatly increased.

PROBLEM SOLUTION

Solution can be initiated by a command from the user's remote terminal, as shown in Figure 8. Small problems (less than 650 degrees of freedom), such as the example problem, are usually solved using an abbreviated version of CASD which can be executed in real time on the XDS Sigma 7 computer. Larger problems, and all NASTRAN and ICES STRUDL analyses, require a batch solution on our IBM 360/195-195 computer. In either case, results are normally routed back to an on-line disk file for subsequent batch listing and evaluation using postprocessing programs.

POSTPROCESSING

Postprocessing includes all processes that operate on the solution results. The postprocessing phase of our system is schematically represented in Figure 9. For example, the first postprocessing operation normally executed is the editing of the solution file to find any error statements. This can be performed at any terminal connected to the direct access computer.

The primary postprocessing tool is a program called VUOUT. This program allows results to be graphically displayed at the user's terminal. It permits isometric viewing for static loads of the undeflected shape with, optionally, the deflected shape (with any magnification), buckle shape (NASTRAN only), internal loads for axial bars and shear panels, applied joint loads, and reactions for static problems. These may be displayed with superimposed bar or node numbers, and may be of the entire model, or of only a specified small section. In addition, load sheet (free body) displays of a single bar or panel element with its applied loads may be requested. In this case, appropriate element properties (length, area, thickness, modulus of elasticity, etc.) can be displayed with the loads.

The program first processes the solution results line by line and writes the data into a compacted on-line file for ready access by the display subroutines. This file is only generated during the first viewing of the results. Subsequent reviewing of the data bypasses this step. The display technique requires only a minimal amount of data in core at any one time and hence allows the results of an essentially unlimited size model to be displayed. The type of display is controlled by the menu selections shown in Figure 10. Actual hard copies of the results of the example problem are shown in Figures 11 through 16. Hard copies provide a permanent record for futher study, stress analysis, and use in reports. The graphical displays highlight any errors and enhance the engineer's understanding of the problem, allowing a more creative use of the finite element technique.

CONCLUSION

The application of interactive graphics to finite element structural analysis is one of the most exciting developments since the introduction of high speed computers. Significant engineering time can be saved and the usability of results enhanced. It allows the engineer to concentrate on the creative aspects of his job, freeing him from tedious mechanical tasks of input coding and output data reduction. It is no substitute for good engineering, but rather provides another tool for unlocking the complexities of intricate structures. Interactive graphic displays of the problem and its solution enhance understanding of structural interactions.

The coupling of our preprocessing and postprocessing programs to work with analysis programs, such as NASTRAN, has given us an inexpensive tool to quickly analyze complex structures. This system has played an important part in our static and dynamic structural analyses of several projects including our NASA Space Shuttle effort, the Navy Harpoon missile, and the NASA Skylab, as well as several advanced design efforts. It has proven to be an effective tool to eliminate many errors associated with finite element model generation and interpretation of the analysis results.

ACKNOWLEDGEMENTS

The authors wish to acknowledge the many contributors to the structural analysis system presented herein. The basic system requirements were defined by the Strength and Structural Dynamics Departments of McDonnell Douglas Astronautics Company - East (MDAC-E). Actual software development was performed primarily by the authors, Dr. D.C. Tsai, and P. A. Giarritano. The CASD analysis program was developed by the Advanced Procedures Group within Structural Engineering at the Douglas Aircraft Company and incorporated into our XDS Sigma 7 computer by MDAC-E.

REFERENCES

1. Prince, M. David, Lockheed-Georgia Company, "Interactive Graphics for Computer-Aided Design," Addison-Wesley Publishing Co., 1971.

2. Katow, M. Smoot, Cooper, Barry M., "NASTRAN Data Generation and Management Using Interactive Graphics," paper contained in "NASTRAN User's Experiences," NASA TMX-2637, 1972.

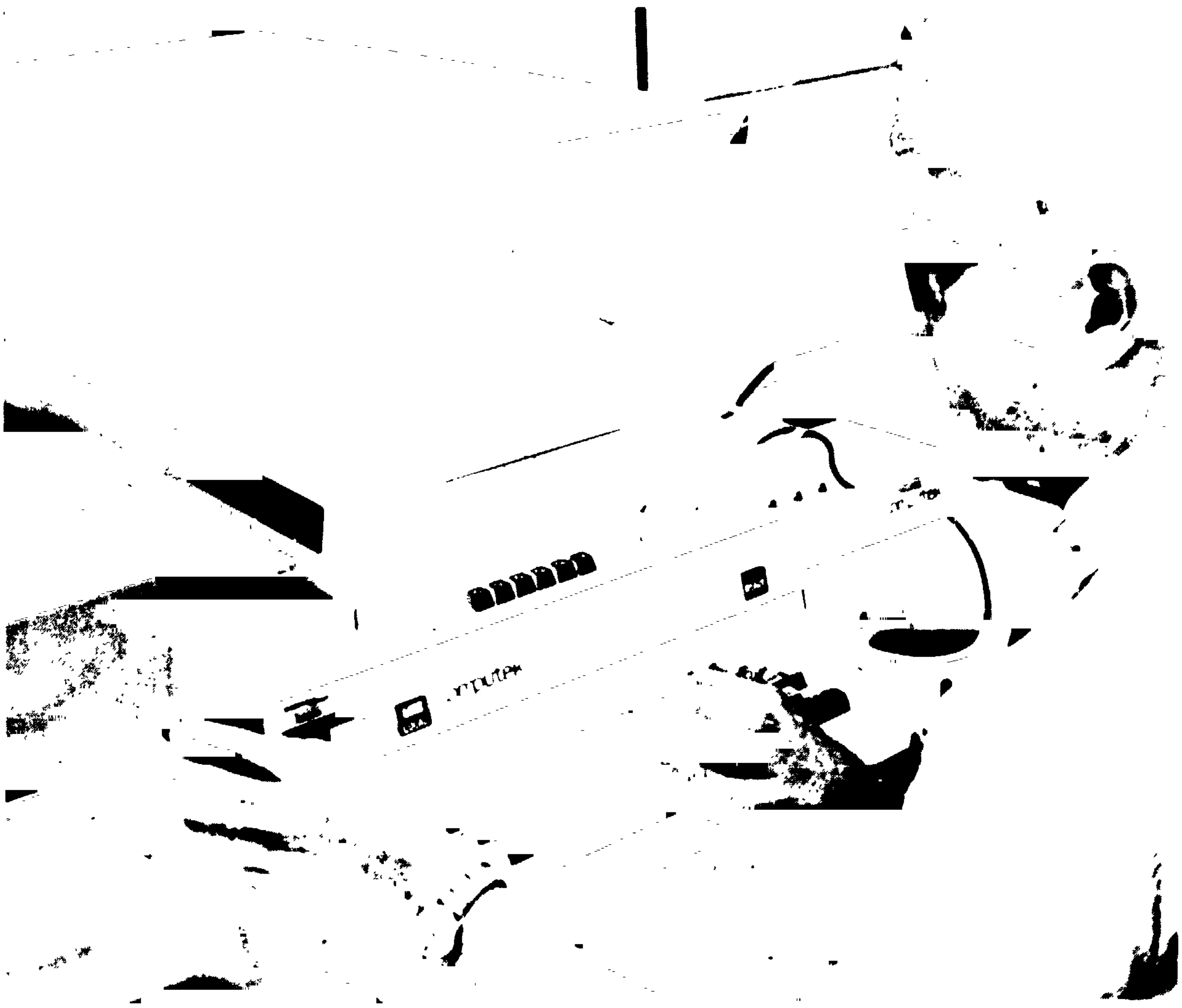


FIGURE 1.–
COMPUTEK INTERACTIVE GRAPHICS TERMINAL.

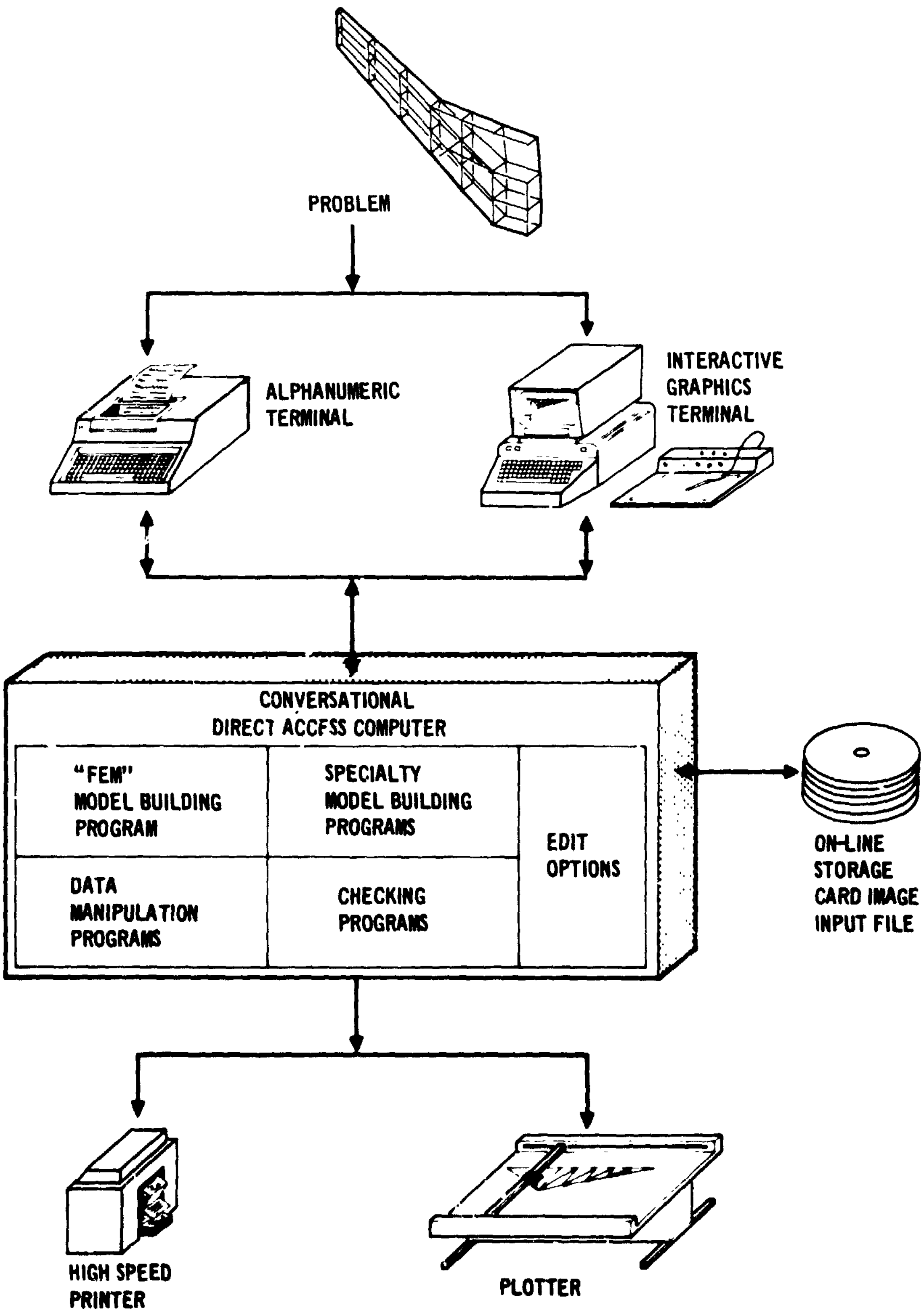


FIGURE 2. PREPROCESSING – BUILDING AND CHECKING THE FINITE ELEMENT MODEL.

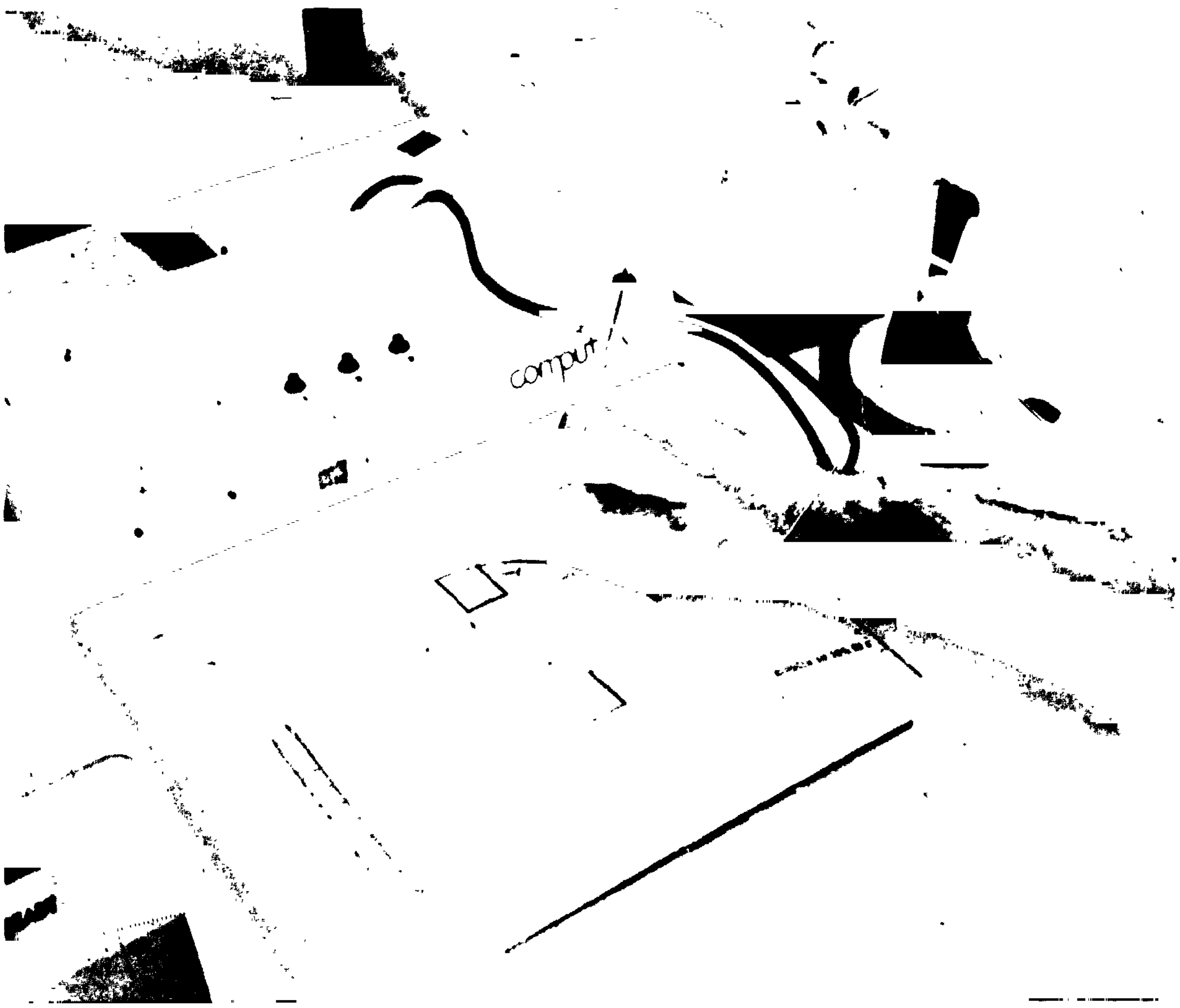


FIGURE 3.– NODE GEOMETRY IS ENTERED USING SCALE DRAWINGS.

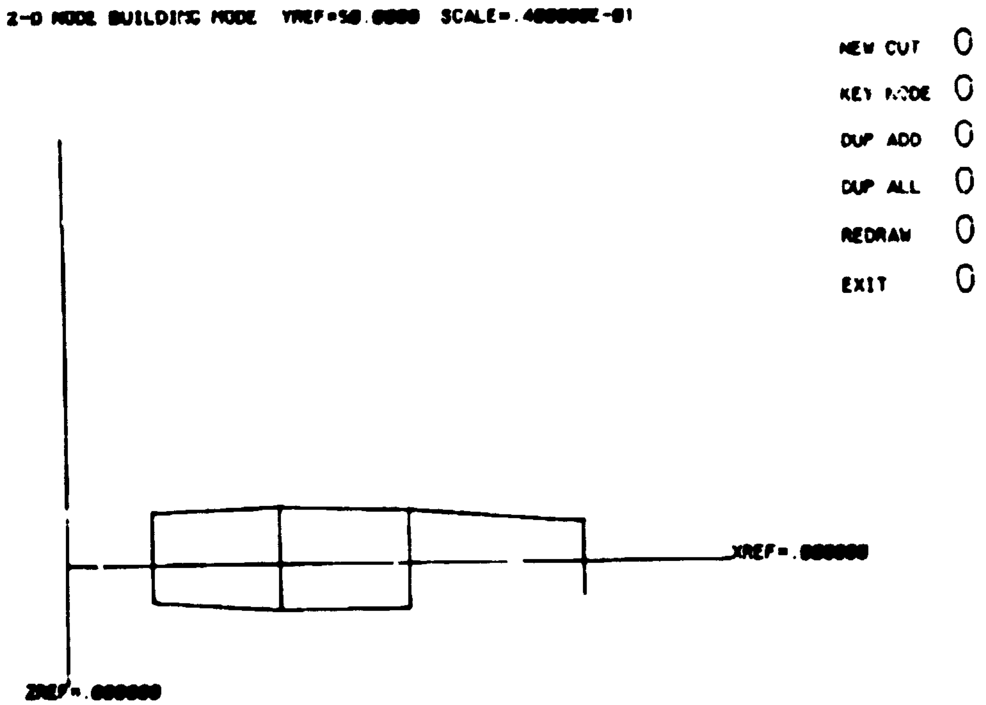


FIGURE 4.– "FEM" DISPLAY OF WING ROOT COMPLETED CROSS SECTION.

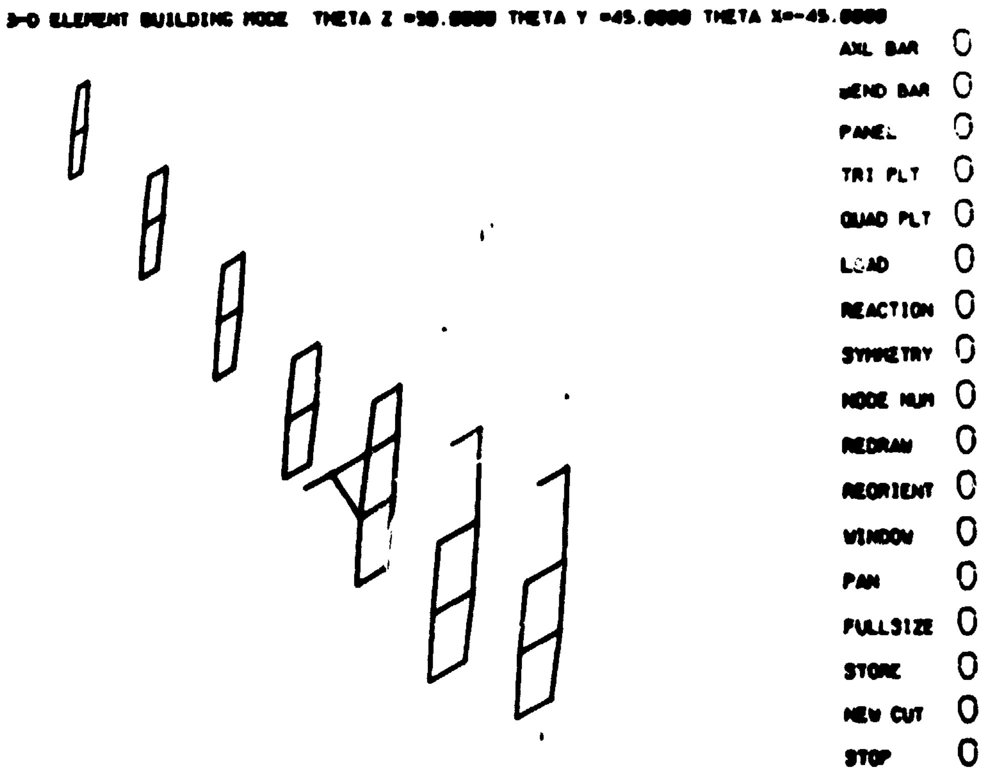


FIGURE 5.– "FEM" ISOMETRIC DISPLAY OF COMPLETED CROSS SECTIONS.

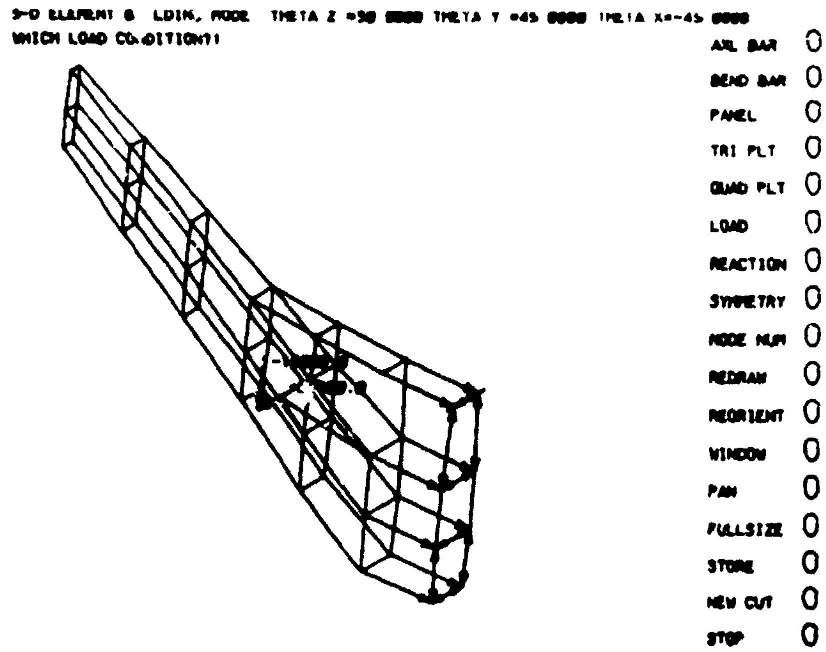


FIGURE 6.– "FEM" DISPLAY OF COMPLETED MODEL WITH FORCE VECTORS.

```
ID   FEM NASTRAN
APP   DISPLACEMENT
SOL 1,0
TIME  2
CEND
 TITLE = FEM,NASTRAN
 LOAD = 600
 SPC = 700
 DISPLACEMENT = ALL
 ELFORCE = ALL
 STRESS = ALL
 SPCFORCES = ALL
BEGIN BULK
CROD          1     430       1       2
CROD          2     430       2       3
CROD          3     430       3       4

CROD         98     430      21      31
CROD         99     430      27      14
CBAR        100     440      27      22      21                       2
CBAR        101     440      27      26      21                       2
CBAR        102     440      22      13      21                       2
CSHEAR      103     420       4      12      11       3
CSHEAR      104     420      12      20      19      11
CSHEAR      105     420       3      11      10       2

CSHEAR      163     420       1       8       7       2
CSHEAR      164     420       2       7       6       3
FORCE       600      26              -15000.  .00000  .00000-1.00000
FORCE       600      26               27300.-1.00000  .00000  .00000
GRID          1               14.84   50.00   24.83
GRID          2               16.42   50.00   61.98
GRID          3               15.24   50.00   92.37

GRID         52               21.30  350.00  173.52
GRID         53              -50.57  150.00  155.27
MAT1        300   1.0+7                 0.3
PBAR        440     300          .1      2.      3.      4.
PQUAD2      410     300         0.1
PROD        430     300         0.1     1.0
PSHEAR      420     300         .02
PTRIA2      400     300         0.1
SPC         700       4           2      .0
SPC         700       5           2      .0

SPC         700       1           1      .0
SPC         700       1           3      .0
ENDDATA
```

FIGURE 7.– CARD IMAGE DATA FOR EXAMPLE PROBLEM.

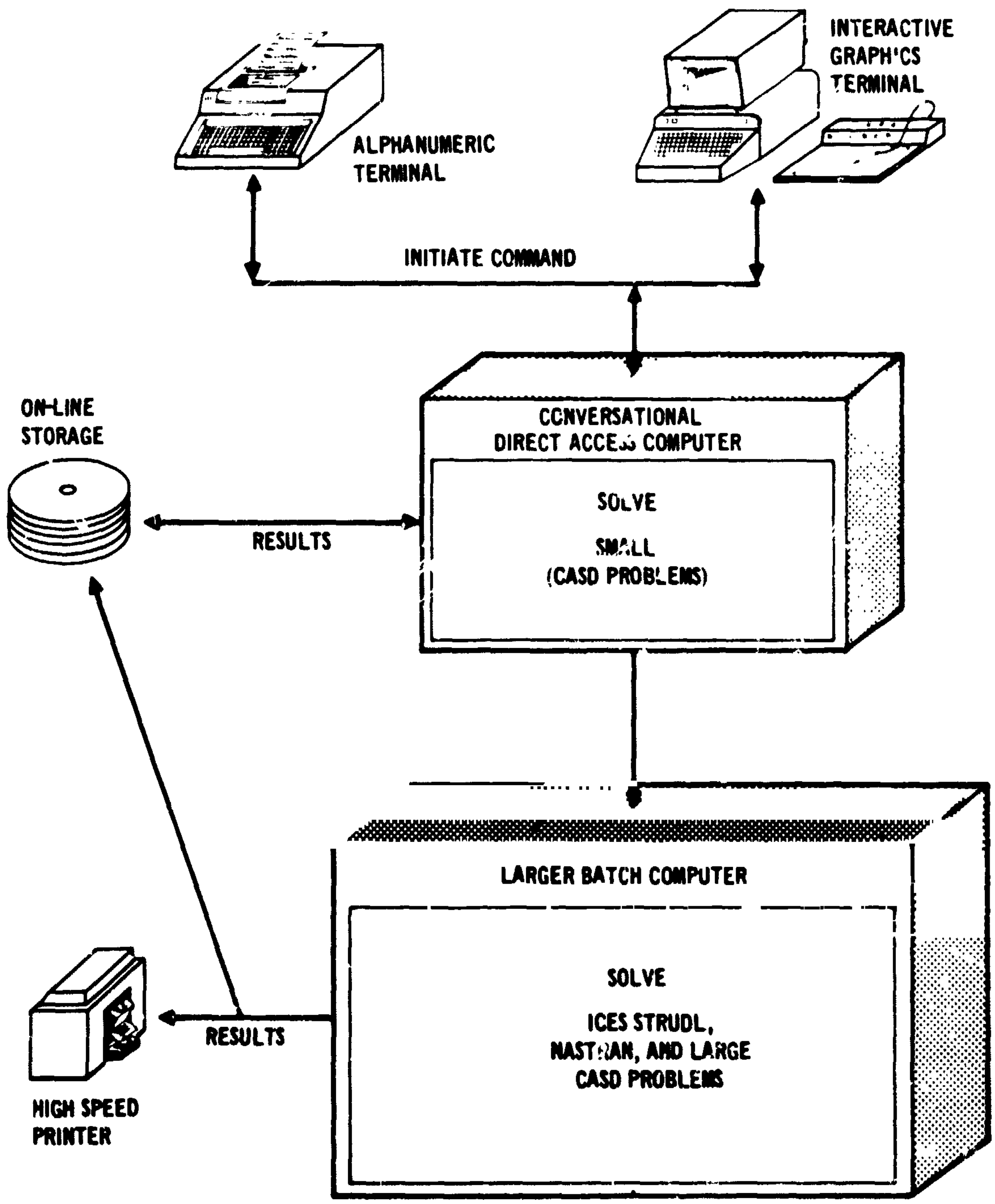


FIGURE 8. SOLUTION: SMALL PROBLEMS IN REAL TIME, LARGE PROBLEMS IN BATCH

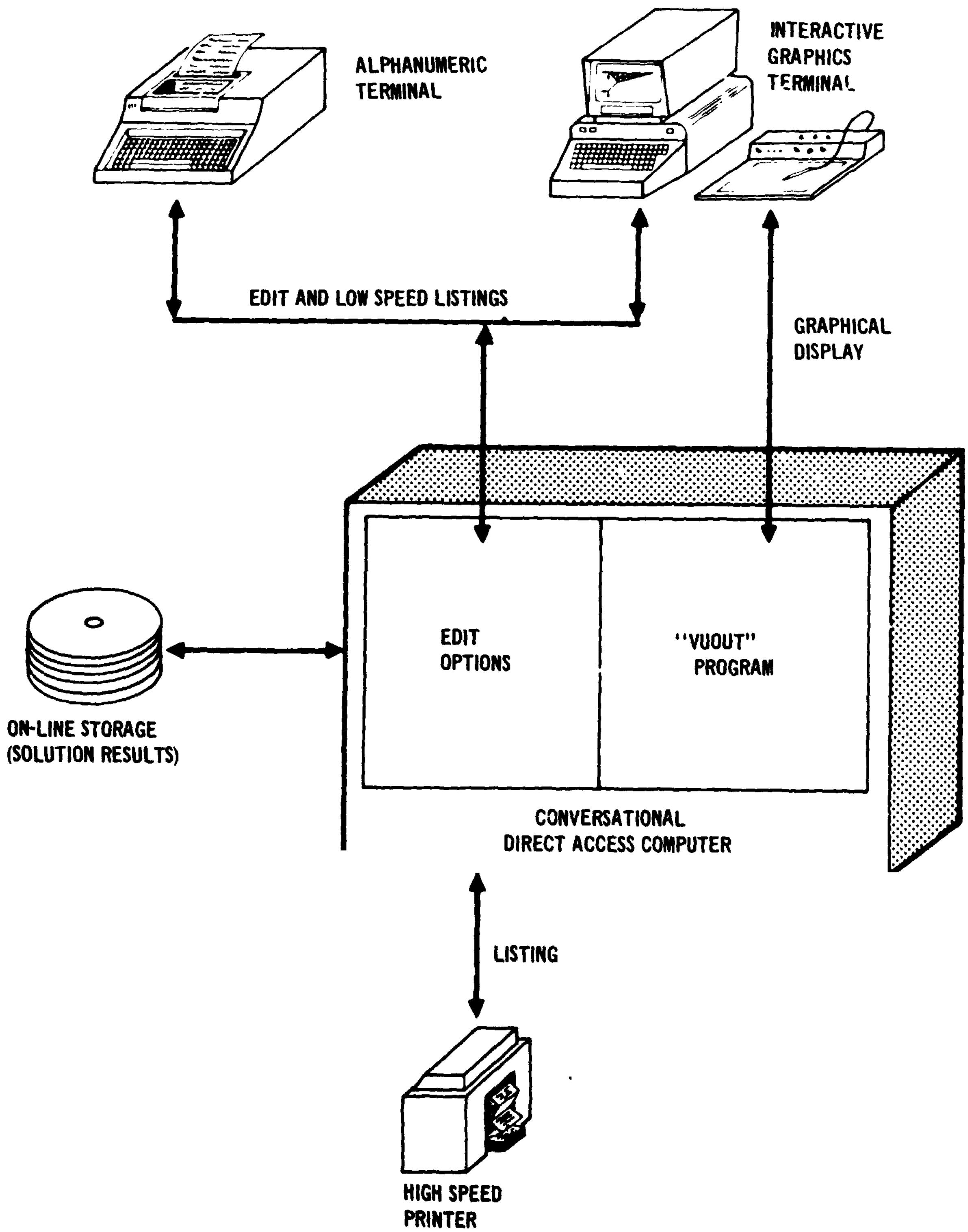


FIGURE 9. POSTPROCESSING—INTERPRETATION OF SOLUTION RESULTS.

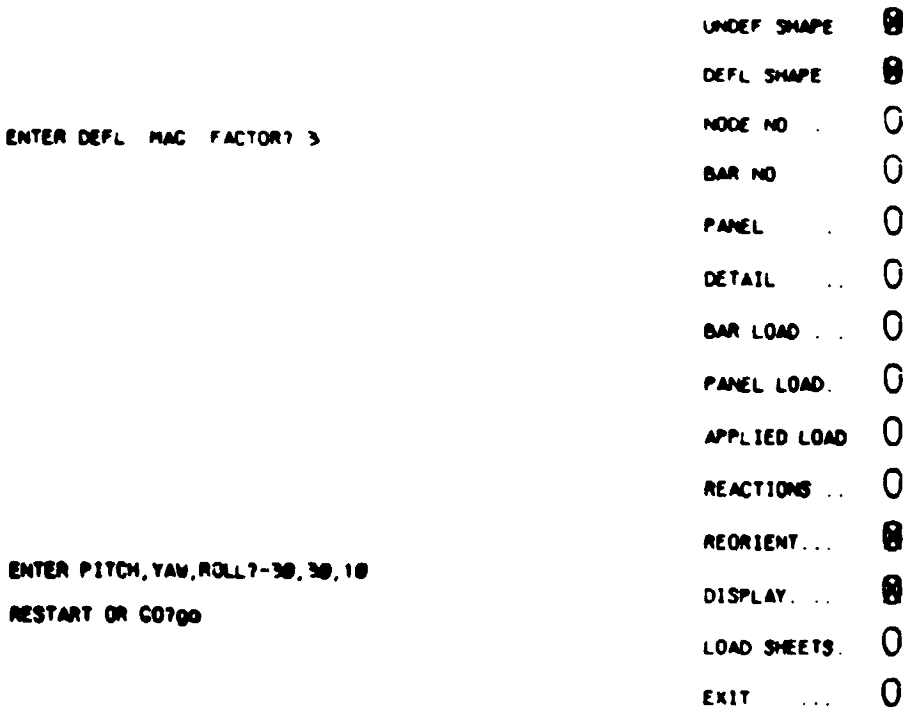


FIGURE 10.– "VUOUT" MENU OPTIONS
FOR POSTPROCESSING DISPLAY.

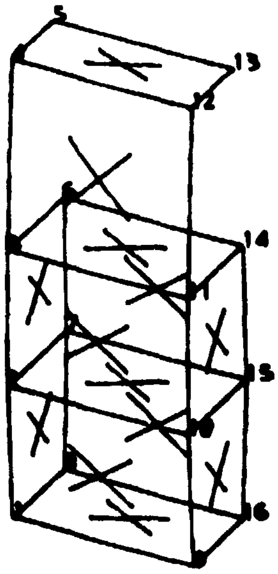


FIGURE 11.– "VUOUT" DISPLAY OF DETAIL SECTION WITH
NODE NUMBERS AND PANELS INDICATED.

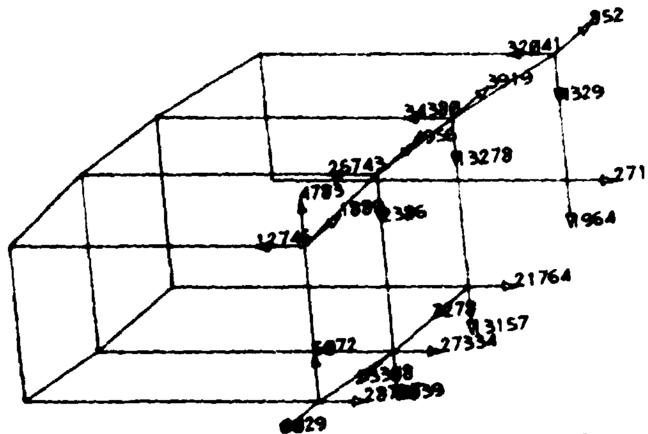


FIGURE 12.. "VUOUT" DISPLAY OF WING ROOT DETAIL WITH REACTIONS.

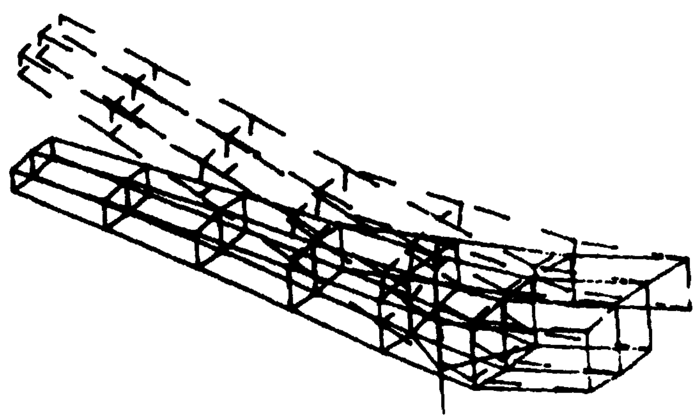


FIGURE 13.. "VUOUT" DISPLAY OF DEFLECTED/UNDEFLECTED SHAPE.

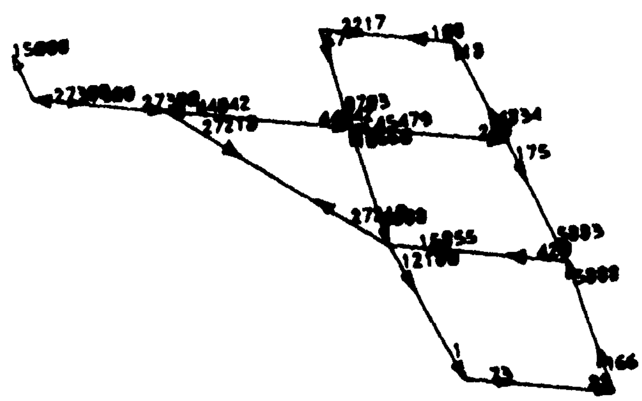
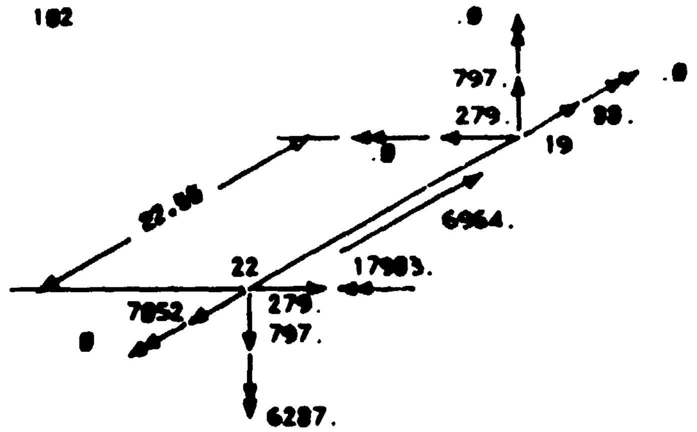


FIGURE 14.. "VUOUT" DISPLAY OF BAR LOADS ON DETAIL SECTION.

BAR NO. 102

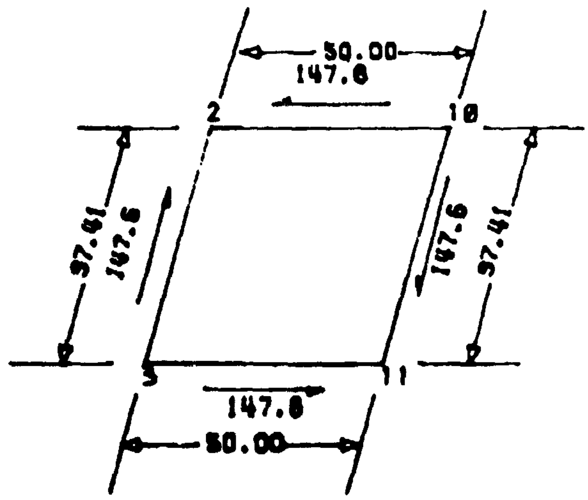


DESIGN LOAD CONDITION NO. 1

MEMBER PROPERTIES
A1= .1000 A2= .1000
IN= 2.000 IT= 2.000
J= .0000
E= 10.0 E 6

FIGURE 15.. TYPICAL "VUOUT" DISPLAY OF BAR LOAD SHEET.

SHEAR PANEL NO. 3



DESIGN LOAD CONDITION NO. 1

MEMBER PROPERTIES
T1= .020
T2= .020
G= 3.5 E 6

FIGURE 16.. TYPICAL "VUOUT" DISPLAY OF SHEAR PANEL LOAD SHEET.