A SYSTEM APPROACH TO DESIGN - ANALYSIS

Gil White
Intergraph Corporation
Huntsville, Alabama

SUMMARY

Today, the increased use of Finite Element Analysis coupled with computer graphics interfaces presents the analyst with many alternatives. The drive toward automation has resulted in the creation of many modeling and post-processing systems that are in use today throughout the world. Many of these systems exist as islands of technology. The need for fully integrated systems is being met by Intergraph Corporation today.

INTRODUCTION

Intergraph's Rand-MICAS (IRM), a computer-aided design and analysis system, runs on DEC VAX and VAX-compatible equipment. The system includes a pre-processor for model development, a complete analysis capability and a post-processor for displaying and managing analysis results. IRM can interface to the majority of major analysis systems including COSMIC NASTRAN.

The system is designed to operate in the design-analysis environment where quick model development and editing are required. Repeated analysis can be done realtime during the design-analysis session.

PURPOSE

The purpose of this paper is to describe an integrated approach to the design process from concept to manufacturing. An attempt will be made to demonstrate the cohesive properties of Intergraph's total system. The foundation of which are graphics and non-graphics data bases operating simultaneously under the VAX/VMS system architecture. The paper will center around the analysis of a retractable automobile headlamp assembly. A static stress analysis of the headlamp will be discussed.

MODEL DEVELOPMENT

Creation of the finite element model is done graphically from any of Intergraph's workstations. Working in a graphics design file and a data management file simultaneously, the analyst creates the model by selecting commands from a graphics menu (Figure 1A and 1B). Integration between design and analysis is achieved through the use of a reference file. The reference file is a read only file that can be viewed by the analyst and used as a guide during model creation. Figure 2 shows the headlamp reference file that was used during model development. The modeling process begins with the selection of a model type from the model type/units tutorial (Figure 3). Dimensional units default to those of the design file. From the active parameters tutorial various modeling, loading, and post-processing parameters are established (Figure 4). Mesh generation is accomplished through the use of any of several meshing routines. Meshing capabilities include mapping to three dimensional "B-Spline" surfaces in addition to various project and sweep commands. All major element types are supported including mid-side node elements, gap elements, and super elements.

Figure 5 shows the completed model of the headlamp. Loading was applied to the front face of the headlamp using a fence pressure command. Fence commands allow the analyst to operate on groups of elements by defining a graphic fence. All real Eigenvalue extraction data is defined at the graphics workstation and becomes resident in the data base.

MODEL EDITING

Editing of the finite model is accomplished while at the graphics workstations. From the graphics environment, the data base can query and information obtained about any model component. Deletion and changes can be made and all associativity maintained. As in the model creation, changes to elements, nodes, loads and boundary conditions can be made one at a time or in a variety of group operations. Editing can be done from an alpha terminal as well as the graphics workstation. Figure 6 shows the model of the headlamp after editing.

ANALYSIS

After completion of the modeling process, the analyst has the option of doing the analysis on many third-party analysis packages or using the internal analysis capability of IRM. Analysis is using third-party packages is accomplished through the use of individual translators. These translators create the complete input deck to any major analysis package. In the case of COSMIC NASTRAN, the executive control deck, case control deck, and bulk data deck are all created. See Figure 7 and Figure 8.

The Intergraph Rand-MICAS analysis system consists of a basic module and six model type options. The basic module contains the data base functions as well as all of the model description and analysis processing capabilities. Included are node, element, load, and application generators. The analysis options include linear static, limited non-linear static and modal dynamic processing capabilities. Material properties include orthotropic as well as isotropic properties. IRM addresses a variety of boundary conditions including: restrained degrees of freedom, specified displacements, springs, gap spring to ground, distributed gap springs to ground, hook springs to ground, and distributed hook spring to ground.

Load type options (global or local) that are applied include:

- Distributed
- Projected surface
- Pressure
- Body force
- Nodal force and moment
- Concentrated
- Partially distributed
- Triangular
- Perimeter surface
- Wind (pressure and velocity)
- Thermal loads

The analysis system addresses general and specific translational masses as well as rotational masses.

In addition to the modeling capabilities offered through the graphics workstation environment, the user can access the same routines from the alpha environment. Both methods operate on the same data base, therefore, continuity is assured.

POST PROCESSING

Using IRM post-processing the analyst can display, manipulate and manage analysis results from the graphics environment. Results are viewed and saved in the working design file. Features of Intergraph Rand-MICAS post-processing are as follows:

- Contouring of any single value or scaler nodal function. The most common being stress, displacements and moments.

- Deformed shape plot including projection of scaler functions.

- Linear combination of load case results and other mathematical operations.

- Vector display of any nodal vector function. The most common being principal stress, velocity and acceleration.

- Color coding of elements for various parameters.

- Animation of displacements including dynamic modal analysis output.

As in the model creation process, the post-processing commands are selected from a graphics screen menu (Figure 9). The user has the option to write any results permanent to the design file or display them in a transient mode. Analysis output is managed and manipulated from the post-processing parameters tutorial (Figure 10). Post-processed results for the headlamp are shown in Figures 11, 12, and 13.

LIST OF FIGURES

1A Pre-Processing Menu (Page 1)

1B Pre-Processing Menu (Page 2)

2 Reference File of Headlamp

3 Model Type Tutorial

4 Active Parameters Tutorial

5 Completed Finite Element Model

6 Finite Element Model showing Editing

7 COSMIC NASTRAN Interface Tutorial

8 COSMIC NASTRAN Input Generated by Translator

9 Post-Processing Menu (Page 3)

10 Post-Processing Parameters Tutorial

11 Plot of Stress Contours

12 Plot of Displacement

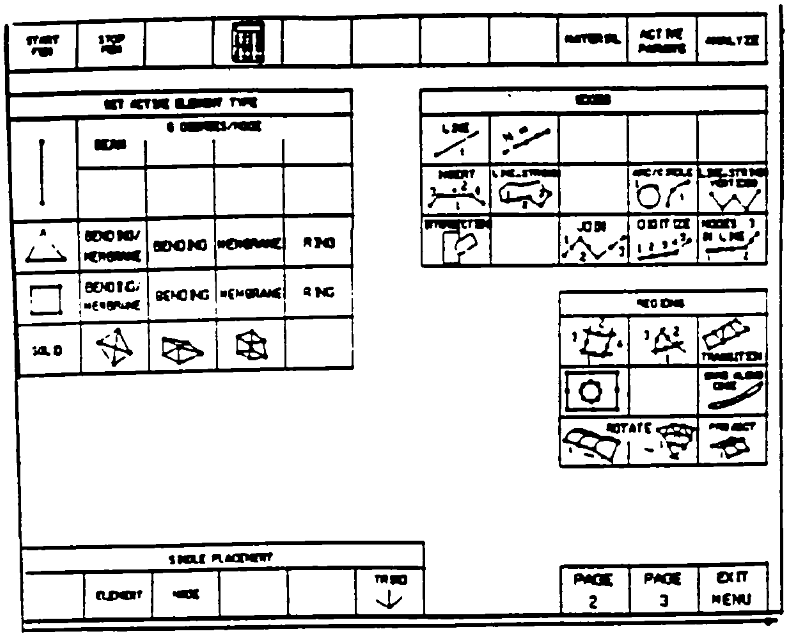

Fig. 1A - Pre-Processing Menu (Page 1)

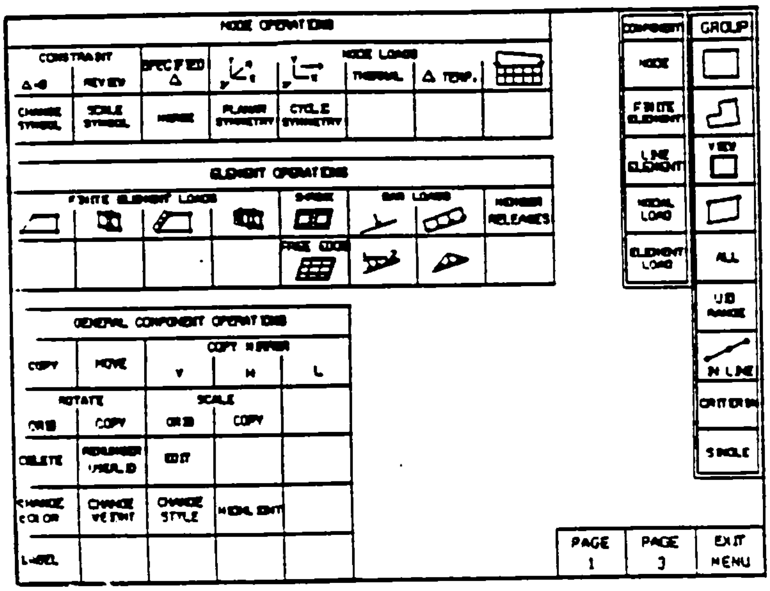

Fig. 1B - Pre-Processing Menu (Page 2)

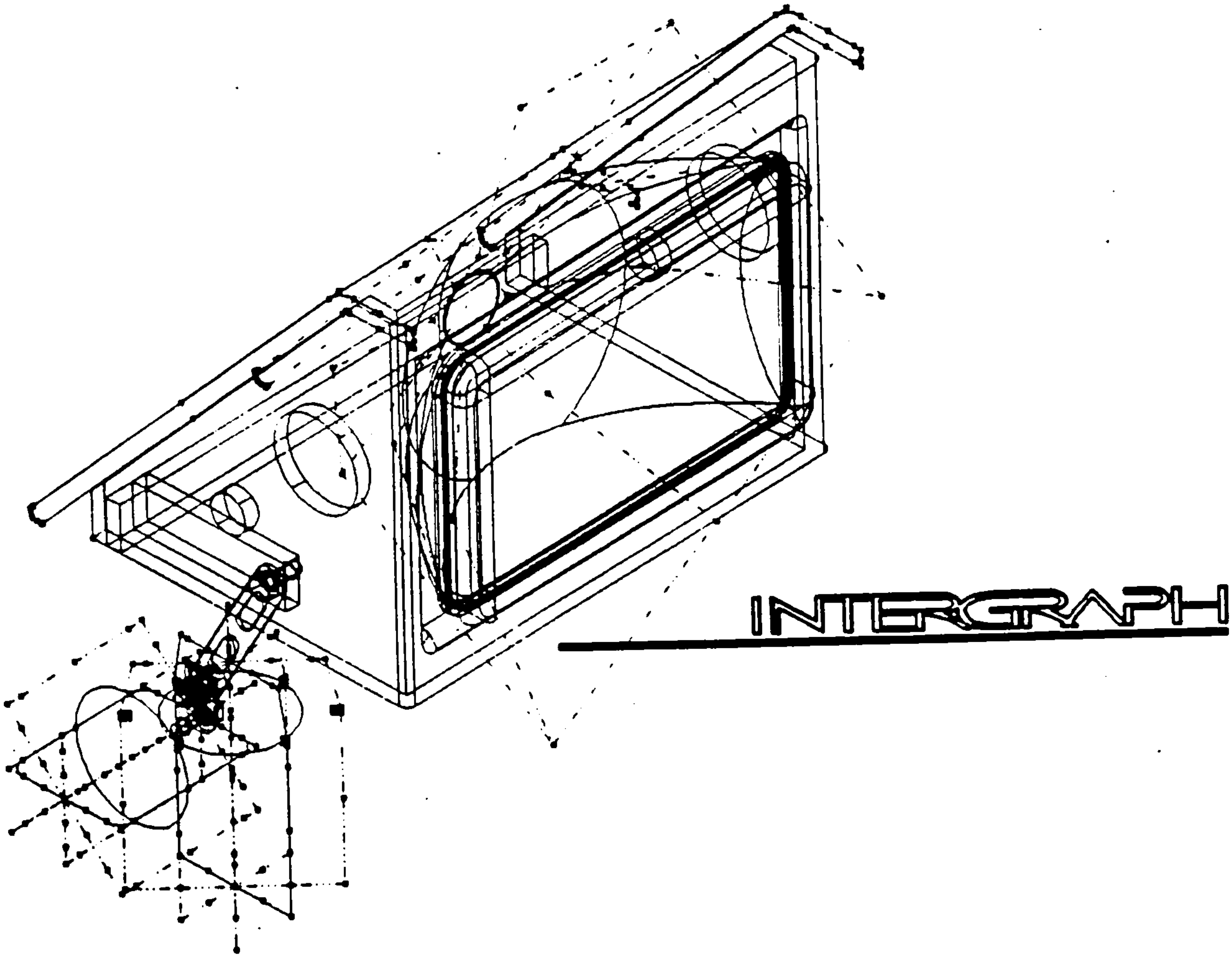

Fig. 2 - Reference File of Headlamp

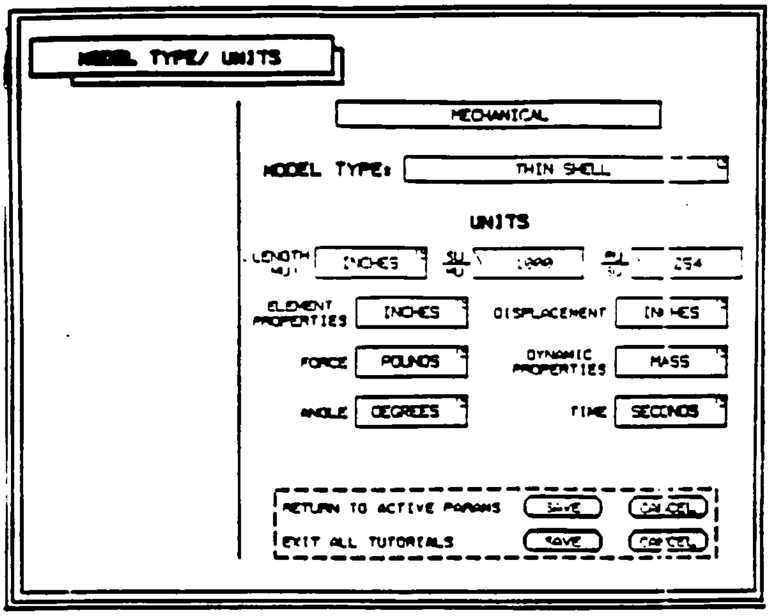

Fig. 3 - Model-Type Tutorial

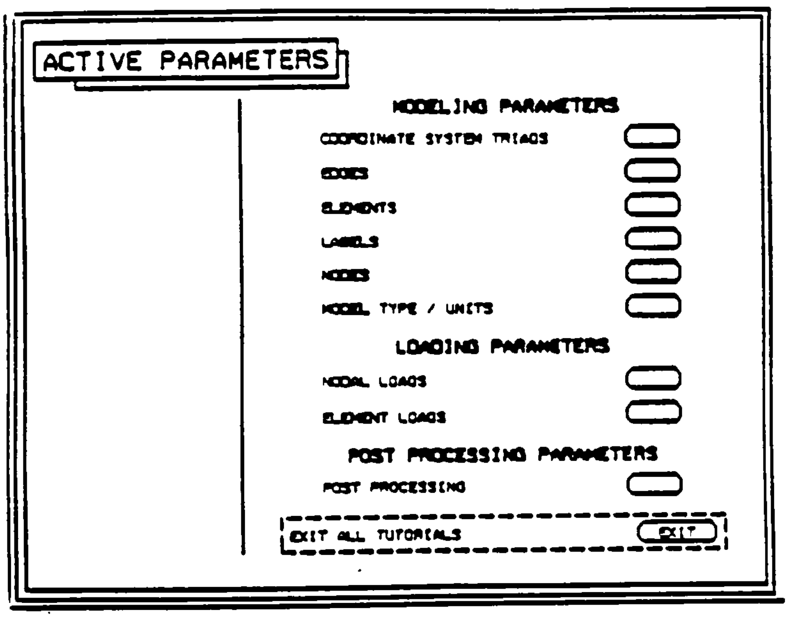

Fig. 4 - Active Parameters Tutorial

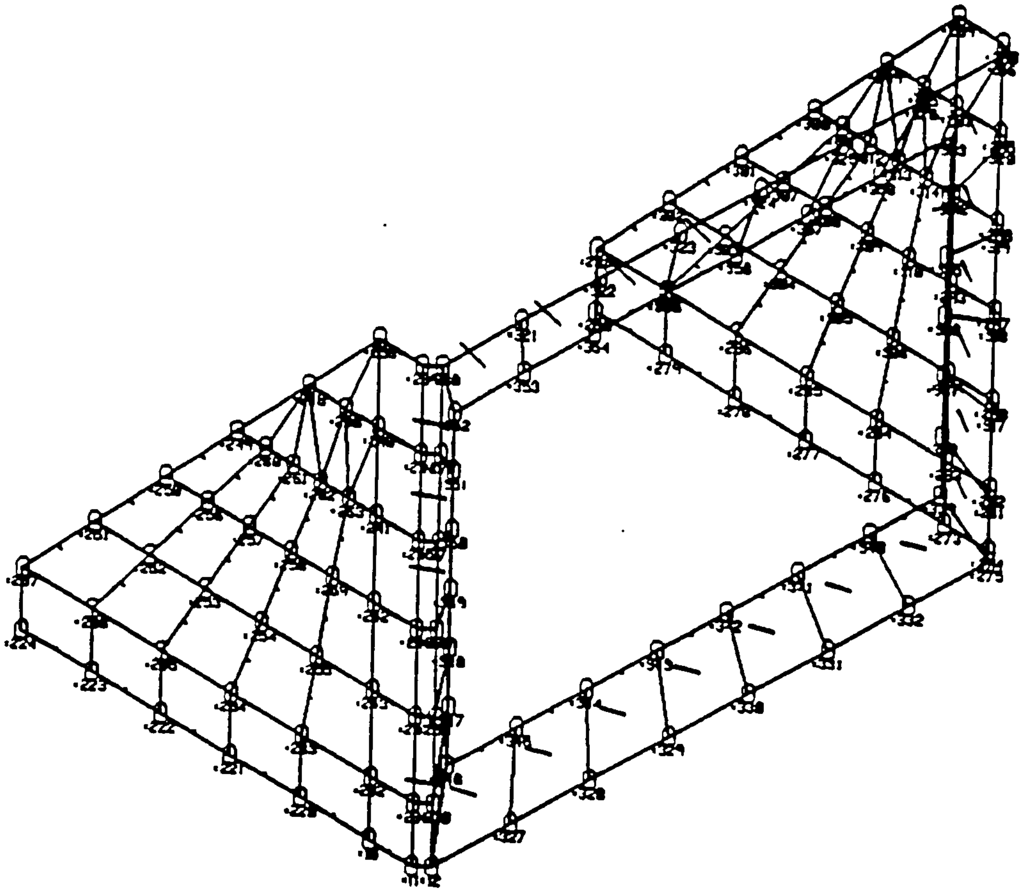

Fig. 5 - Completed Finite Element Model

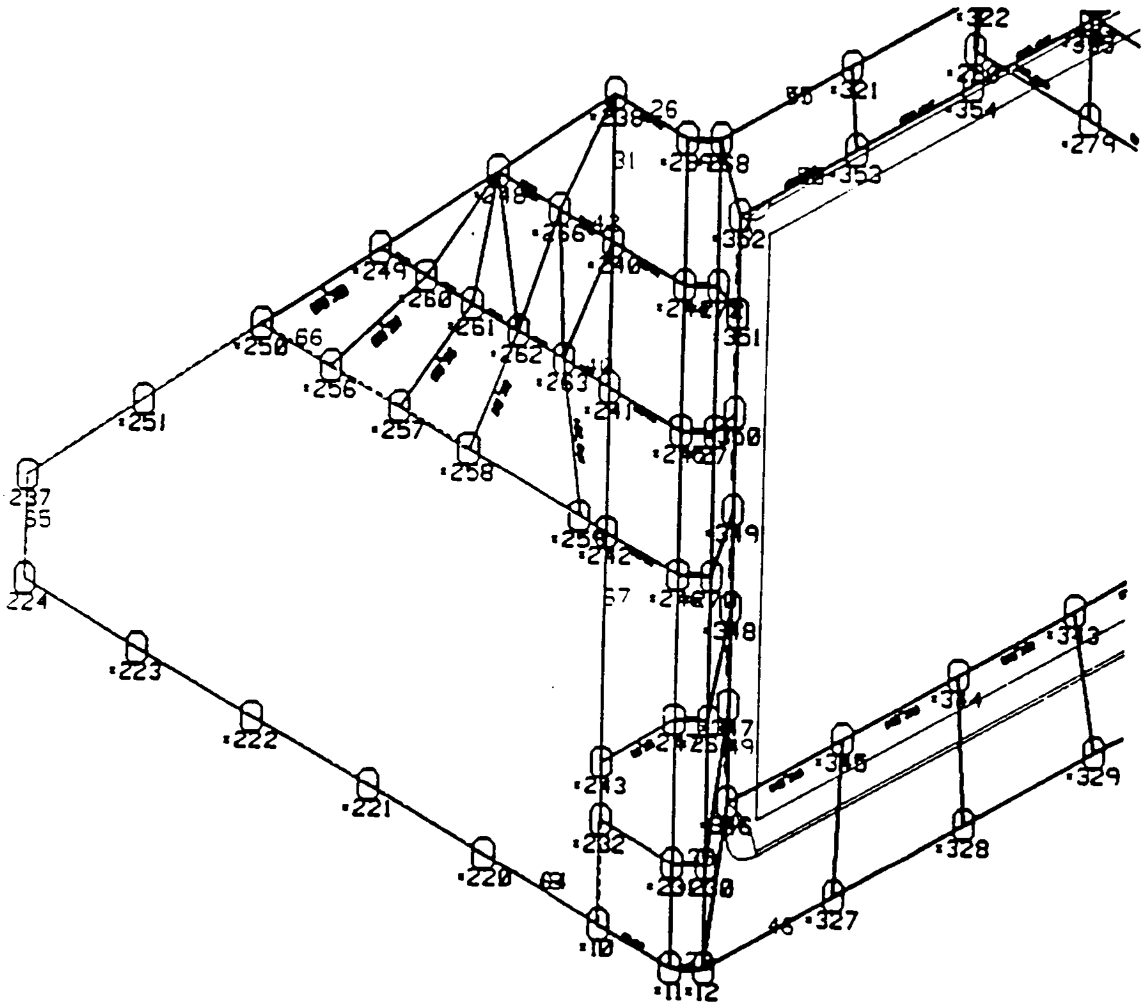

Fig. 6 - Finite Element Model Showing Fairing

COSMIC NASTRAN OPTIONS

CASE CONTROL GENERATION	
YES	NO

CURRENT

BULK DATA FORMAT		
SINGLE FIELD	DOUBLE FIELD	AUTO

CURRENT

DATABASE TO BE TRANSLATED :FEM1 .DBS

BULK DATA OUTPUT FILE :FEM1.DAT

Change current options, translate, or abort option definition.

TRANSLATE

ABORT

Fig. 7 - COSMIC NASTRAN Interface Tutorial

```
 2      SUBTITLE= MODELLED WITH INTERGRAPH FEMS SYSTEM
 3      LABEL= ANALYZED BY MSC/NASTRAN
 4      ECHO= SORT
 5      OLOAD= ALL
 6      GPSTRESS = ALL
 7      DISPLACEMENT(PRINT,PUNCH)= ALL
 8      ELSTRESS(PLOT)= ALL
 9      SUBCASE     1
10      LABEL= LOAD SET NUMBER     1
11      LOAD=     1
12      SPC =     1
13      OUTPUT(POST)
14      SET 10 = ALL
15      SURFACE 1 SET 10 FIBRE Z1 NORMAL X3 TOPOLOGICAL
16      BEGIN BULK

             INPUT BULK DATA CARD COUNT =    4047
```

MODELLED WITH INTERGRAPH FEMS SYSTEM

SORTED BULK DATA ECHO

CARD COUNT	. 1 ..	2 ..	3 ..	4 ..	5 ..	6 ..	7 ..	8 ..	9 ..	10 .
1-	COUAD4 1	1	1	2	22	21				
2-	COUAD4 2	1	21	22	23	20				
3-	COUAD4 3	1	20	23	12	11				
4-	COUAD4 4	1	2	3	24	22				
5-	COUAD4 5	1	22	24	25	23				
6-	COUAD4 5	1	23	25	13	12				
7-	COUAD4 7	1	3	4	25	24				
8-	COUAD4 8	1	24	26	27	25				
9-	COUAD4 9	1	25	27	14	13				
10-	COUAD4 10	1	4	8	28	26				
11-	COUAD4 11	1	26	28	29	27				
12-	COUAD4 12	1	27	29	15	14				
13-	COUAD4 13	1	8	9	30	28				
14-	COUAD4 14	1	28	30	31	29				
15-	COUAD4 15	1	29	31	16	15				
16-	COUAD4 16	1	9	10	19	30				
17-	COUAD4 17	1	30	19	18	31				
18-	COUAD4 18	1	31	18	17	16				
19-	COUAD4 19	1	11	12	705	706				
20-	COUAD4 20	1	12	13	704	705				
21-	COUAD4 21	1	13	14	703	704				
22-	COUAD4 25	1	36	35	34	33				
23-	COUAD4 26	1	34	41	44	33				
24-	COUAD4 27	1	33	44	45	32				
25-	COUAD4 28	1	32	45	5	6				
26-	COUAD4 29	1	41	40	46	44				
27-	COUAD4 30	1	44	46	47	45				
28-	COUAD4 31	1	45	47	4	5				
29-	COUAD4 32	1	40	39	48	46				
30-	COUAD4 33	1	46	48	49	47				
31-	COUAD4 34	1	47	49	3	4				
32-	COUAD4 35	1	39	38	50	48				
33-	COUAD4 36	1	48	50	51	49				

Fig. 8 - COSMIC NASTRAN Input Generated by Translator

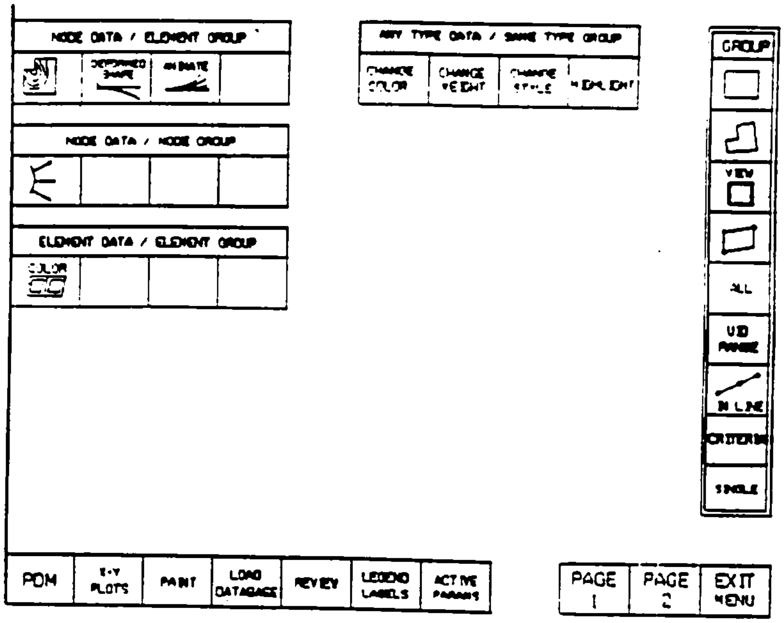

Fig. 9 - Post-Processing Menu

(Page 3)

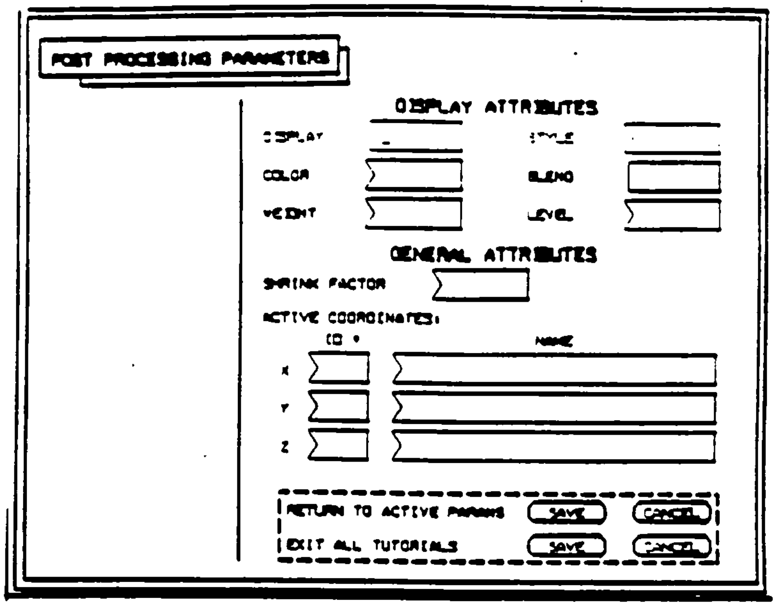

Fig. 10 - Post Processing Parameters Tutorial

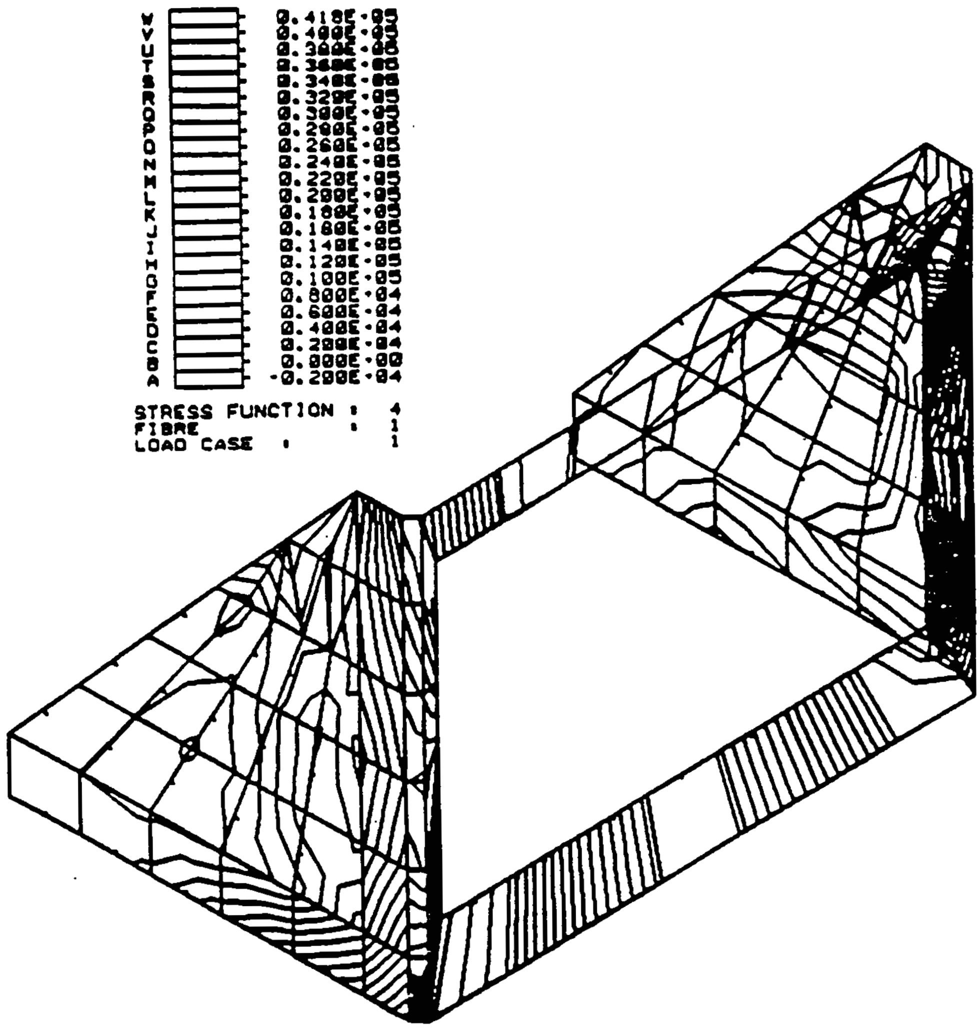

Fig. 11 - Plot of Stress Contours

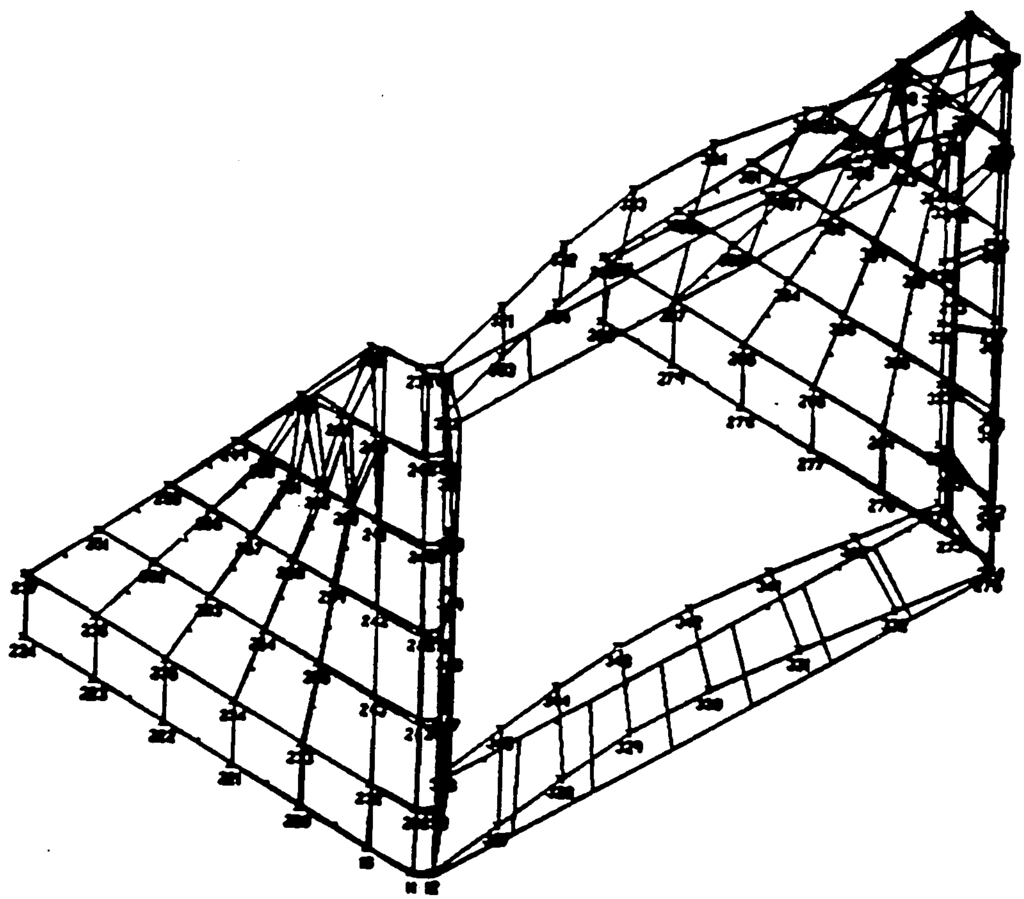

Fig. 12 - Plot of Displacement