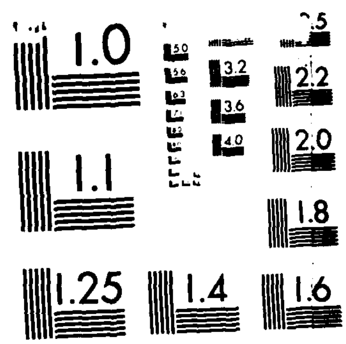
AD-A163 556 CADS - A COMPUTER AIDED DESIGN SYSTEM VOLUME 1 FINAL SUMMARY REPORT (U) ROCKWELL INTERNATIONAL EL SEGUNDO CA NORTH AMERICAN AIRCRAFT D.. M C LESS ET AL. DEC 85 1/1

UNCLASSIFIED AFWAL-TR-85-3066-VOL-1 F33615-81-C-3229 F/G 9/2 NL


1.0
1.1
1.25
1.4
1.6
1.8
2.0
2.2
3.2
3.6
4.0

(12)

AFWAL-TR-85-3066
Volume I

CADS - A COMPUTER AIDED DESIGN SYSTEM
Volume I - Final Summary Report

Michael C. Less
Susan Manuel

Rockwell International
North American Aircraft Operations (NAAO)
El Segundo, California 90009

December 1985

Final Report for Period December 1981 - May 1985

Approved for public release; distribution unlimited

FLIGHT DYNAMICS LABORATORY
AIR FORCE WRIGHT AERONAUTICAL LABORATORIES
AIR FORCE SYSTEMS COMMAND
WRIGHT-PATTERSON AIR FORCE BASE, OHIO 45433

NOTICE

When Government drawings, specifications, or other data are used for any purpose other than in connection with a definitely related Government procurement operation, the United States Government thereby incurs no responsibility nor any obligation whatsoever; and the fact that the government may have formulated, furnished, or in any way supplied the said drawings, specifications, or other data, is not to be regarded by implication or otherwise as in any manner licensing the holder or any other person or corporation, or conveying any rights or permission to manufacture use, or sell any patented invention that may in any way be related thereto.

This report has been reviewed by the Office of Public Affairs (ASD/PA) and is releasable to the National Technical Information Service (NTIS). At NTIS, it will be available to the general public, including foreign nations.

This technical report has been reviewed and is approved for publication.

Victoria A. Tischler
VICTORIA A. TISCHLER
Project Engineer
Design & Analysis Methods Group

Fred A. Picchioni
FREDERICK A. PICCHIONI, Lt Col, USAF
Chief, Analysis & Optimization Branch

FOR THE COMMANDER

R J Hegstrom
ROGER J. HEGSTROM, Col, USAF
Chief, Structures & Dynamics Div.

"If your address has changed, if you wish to be removed from our mailing list, or if the addressee is no longer employed by your organization please notify AFWAL/FIBRA, W-PAFB, OH 45433 to help us maintain a current mailing list".

Copies of this report should not be returned unless return is required by security considerations, contractual obligations, or notice on a specific document.

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE

AD-A163 556

REPORT DOCUMENTATION PAGE

1a. REPORT SECURITY CLASSIFICATION: UNCLASSIFIED	1b. RESTRICTIVE MARKINGS: N/A
2a. SECURITY CLASSIFICATION AUTHORITY: N/A	3. DISTRIBUTION/AVAILABILITY OF REPORT: Approved for Public Release; Distribution Unlimited
2b. DECLASSIFICATION/DOWNGRADING SCHEDULE: N/A	
4. PERFORMING ORGANIZATION REPORT NUMBER(S)	5. MONITORING ORGANIZATION REPORT NUMBER(S): AFWAL-TR-85-3066, Volume I
6a. NAME OF PERFORMING ORGANIZATION: Rockwell International; 6b. OFFICE SYMBOL (If applicable)	7a. NAME OF MONITORING ORGANIZATION: Air Force Wright Aeronautical Laboratories Air Force Systems Command (AFWAL/FIBRA)
6c. ADDRESS (City, State and ZIP Code): North American Aircraft Operations (NAAO) El Segundo, California 90009	7b. ADDRESS (City, State and ZIP Code): Wright-Patterson AFB OH 45433
8a. NAME OF FUNDING/SPONSORING ORGANIZATION: Flight Dynamics Laboratory; 8b. OFFICE SYMBOL (If applicable): AFWAL/FIBRA	9. PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER: F33615-81-C-3229
8c. ADDRESS (City, State and ZIP Code): Air Force Wright Aeronautical Laboratories Air Force Systems Command Wright-Patterson AFB OH 45433	10. SOURCE OF FUNDING NOS.: PROGRAM ELEMENT NO. 62201F; PROJECT NO. 2401; TASK NO. 02; WORK UNIT NO. 49
11. TITLE (Include Security Classification) CADS-A Computer Aided Design System, Vol I-Final Summary Report (U)	

12. PERSONAL AUTHOR(S): Less, Michael C., Manuel, Susan

13a. TYPE OF REPORT	13b. TIME COVERED	14. DATE OF REPORT (Yr., Mo., Day)	15. PAGE COUNT
FINAL	FROM 12/01/81 TO 4/30/85	December 1985	42

16. SUPPLEMENTARY NOTATION

17. COSATI CODES			18. SUBJECT TERMS (Continue on reverse if necessary and identify by block number)
FIELD	GROUP	SUB. GR.	Computer Aided Design, DI-3000, Finite Elements, NASTRAN, Structural Analysis & Optimization, Model Generation, Pre and Post Processor
01	03		
13	13		

19. ABSTRACT (Continue on reverse if necessary and identify by block number)

This is the final summary report for the Computer Aided Design System, "CADS". CADS is a pre and post processor for structural analysis and optimization programs based on the finite element method. The system supports five functional modules controlled by an Executive Monitor. All of these modules communicate with a data base through a data manager. In addition a post output translator, CADSPP, is available which processes output from finite element programs, e.g. NASTRAN, directly into the data base. This report gives an overview of the capability available in CADS. Several illustrations demonstrate the funcitons of the Preprocessor, Display, and Postprocessor Modules.

20. DISTRIBUTION/AVAILABILITY OF ABSTRACT: UNCLASSIFIED/UNLIMITED ☐ SAME AS RPT. ☒ DTIC USERS ☐	21. ABSTRACT SECURITY CLASSIFICATION: UNCLASSIFIED
22a. NAME OF RESPONSIBLE INDIVIDUAL: V. A. TISCHLER	22b. TELEPHONE NUMBER (Include Area Code): 513-255-6992; 22c. OFFICE SYMBOL: AFWAL/FIBRA

DD FORM 1473, 83 APR EDITION OF 1 JAN 73 IS OBSOLETE.

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE

FOREWORD

This final report was prepared by Rockwell International, North American Aircraft Operations (NAAO), El Segundo, California for the Structures and Dynamics Division, Flight Dynamics Laboratory (FDL), Wright-Patterson Air Force Base, Ohio. The work was performed under Contract F33615-81-C-3229 which was initiated under Project No. 1401. Mrs. V. Tischler was FDL project engineer for this effort.

The "Development of a Computer-Aided Design System" (CADS) was a 41-month effort with this final report consisting of three volumes. Volume I, "Final Summary Report," presents an overview of the CADS software capabilities; Volume II, "User's Guide," contains the detailed instructions for each of the commands in the CADS software; and Volume III, "Program Maintenance Manual," describes the internal structure of CADS for use in future maintenance and enhancement of the code.

The Rockwell program manager for this effort was Mr. M. C. Less, NAAO Advanced Structures and Materials department. He was supported by Mrs. S. Manuel of the same department.

The work described in this report was initiated in December 1981 and completed in May 1985. This report was submitted for publication in May 1985.

CONTENTS

ILLUSTRATIONS

TABLES

1.0 INTRODUCTION

The widespread use of a large variety of finite element (FE) analysis codes to perform structural analysis tasks has focused attention on a common Air Force and industry problem: the relatively large amount of time and effort required to perform data preparation, data validation, and resultant FE analysis tasks with existing state-of-the-art codes. This problem is further aggravated by the relatively slow, interactive response of mainframe time-sharing computer processing systems. To reduce time and effort, a computer-aided, advanced interactive graphics, minicomputer-based, finite element modeling system has been developed. This system includes mesh generation and validation capabilities as preprocessing functions as well as interactive graphic features for postprocessing the analysis code output data.

The Computer Aided Design System (CADS) software's most important aspects are that it is targeted for 32-bit minicomputer hardware, makes use of FORTRAN 77 and device independent graphics, and supports the definition of composite elements. The CADS program utilizes VAX 11/780 hardware with secondary testing for transportability having been performed on IBM 4341 and PRIME 850 hardware. CADS is modular in nature with various functional modules accessed through a common Executive Monitor and makes use of common data base routines, as shown in Figure 1.

The contract, F33615-81-C-3229, was initiated in December 1981 and completed in May 1985 with this final report. Volume I, "Final Summary Report," presents an overview of the CADS software capabilities; Volume II, "User's Guide," and Volume III, "Program Maintenance Manual," give detailed user instruction and source code descriptions of the CADS software. These three volumes make up the final documentation of the CADS software.

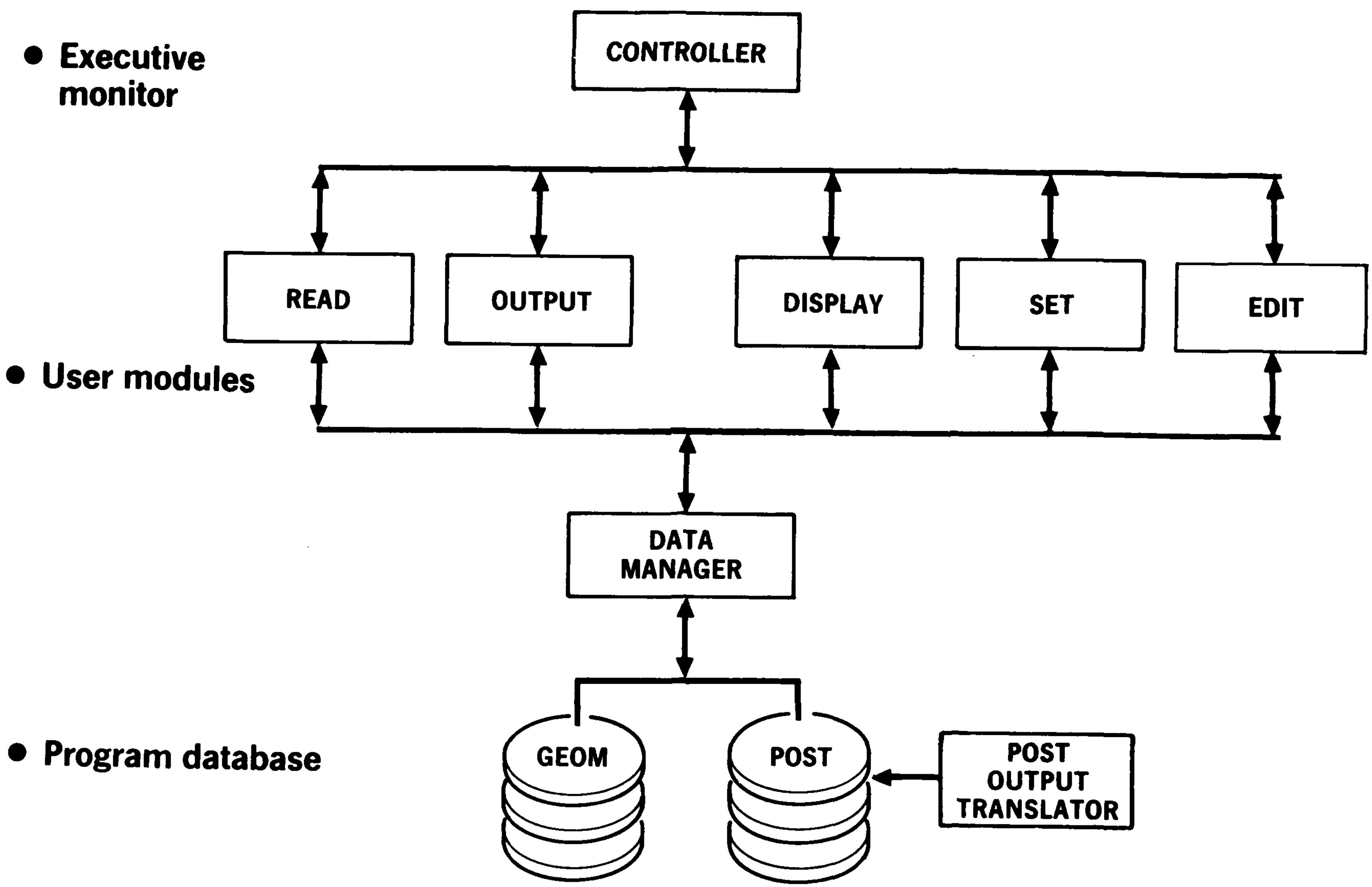


Figure 1. Modular Nature of CADS Software

This Final Summary Report presents a brief overview of the capabilities of the CADS software. Each section covers a different set of major functions. Figure 2 shows the relationship of the modules and processors which are used to perform the actual CADS commands. For example, the OUTPUT module has three processors called, ANALYZE, NASTRAN and OPTSTAT.

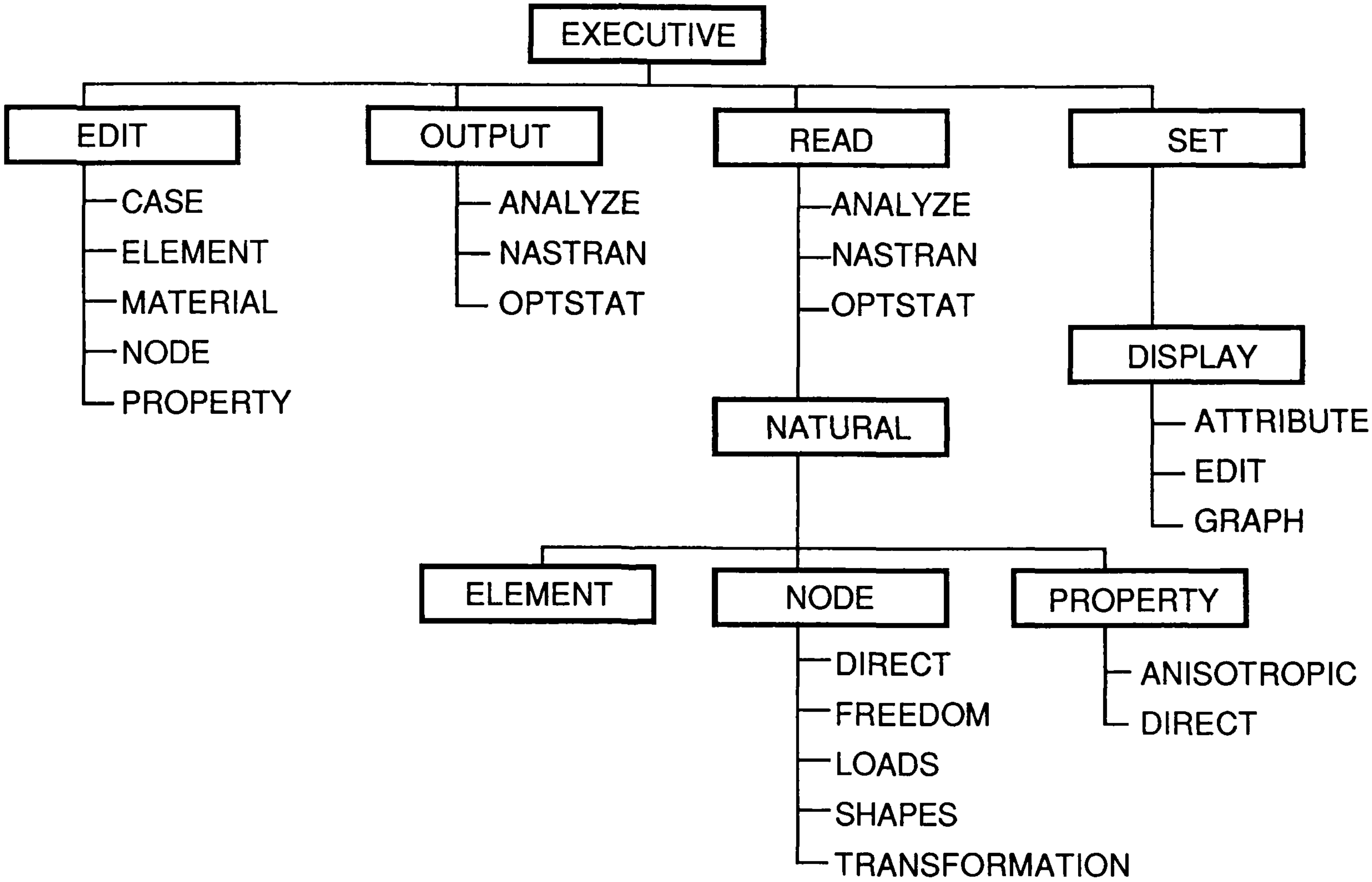


Figure 2. Overall Program Layout

2.0 PREPROCESSOR FUNCTIONS

The CADS software provides a range of functions for preprocessing information for finite element modeling. These functions help the user define complex models in an efficient, cost-effective manner. Interactive displays during the model generation process provide quick and easy validation of the new model.

The preprocessor functions perform three primary activities. The first is to generate the finite element model from user commands; the second is to read and translate existing models to the CADS internal geometry data base; and the third is to translate from CADS to an analysis program format. The generation functions are performed in the NATURAL processor under the READ module. NATURAL provides three submodules, ELEMENT, NODE and PROPERTY, to generate the model's elements, nodes, and property data, respectively.

In addition to NATURAL, the READ module contains the ANALYZE, NASTRAN and OPTSTAT processors. These processors translate existing finite element model input data to the CADS geometry data base. ANALYZE is an Air Force developed analysis program for aerospace structures using isotropic membrane elements. OPTSTAT was also developed by the Air Force. It supports the optimization and analysis of a structure subject to static loads and can use either isotropic or composite layered elements. NASTRAN is the general purpose finite element analysis program developed by NASA. These READ module translators convert the formatted input data files to the neutral CADS data base for display, editing and output functions.

The OUTPUT module performs the reverse translation from the geometry data base to the ANALYZE, NASTRAN, or OPTSTAT format based upon the particular processor which is executed. CADS will attempt to output the model as best it can, however, these three programs have varying formats and capabilities and so there is not a one-to-one correspondence in the outputs.

As an example, the ANALYZE and OPTSTAT programs do not support bending plate element types and so a translation of a model geometry with bending plates is not possible into those particular formats.

In generating a model using the NATURAL processor the NODE submodule is used to generate the node geometry data. Generally the node coordinate data is generated first using the DIRECT or SHAPES subprocessors. DIRECT generates nodes based upon key points, lines, and individual point inputs. Figure 3 is an example of the commands used to generate a simple plate node pattern.

```
BEGIN           NODE
BEGIN           DIRECT
NODE        11   3.0     9.0  2.0   TO  14    3.0   3.0   2.0
REPEAT 2    10   2.0    -1.0  0.0       10    2.0  -1.0   0.0
NODE        51  11.0     9.0  2.0   TO  54   11.0   3.0   2.0
NODE        31                      TO   51                    BY  10
REPEAT 3     1                           1                          0
END
END
```

Figure 3. Plate Node Generation

The SHAPES subprocessor allows the definition of nodes along circles, ellipses, or parabolas as defined by the user.

The ELEMENT submodule would be used after the NODE submodule to specify the element connectivities and type to be used. The element types supported by CADS are listed in Table 1.

The commands in Figure 4 are used to generate quadrilateral membrane (CQDMEM1) and surrounding axial rod (CONROD) elements for the plate nodes shown in Figure 3. Each element type must form at least one element group. The groups can be used to help specify the elements to be plotted at any one time in the DISPLAY module.

TABLE 1

ELEMENT TYPES SUPPORTED BY CADS

ANALYZE OPTSTAT		CADS ELEMENT		NASTRAN ELEMENT	CADS PROPERTY	NASTRAN PROPERTY
2	=	CROD	=	CROD	PR2	PROD
2	=	R2	=	CONROD	PR2	
		B2	=	CBAR	PB2	PBAR
3	=	TM	=	CTRMEM	PID	PTRMEM
4	=	QM1	=	CQDMEM1	PID	PQDMEM1
		TB2	=	CTRIA2	PID	PTRIA2
		QB2	=	CQUAD2	PID	PQUAD2
		TB1	=	CTRIA1	PTQ1	PTRIA1
		QB1	=	CQUAD1	PTQ1	PQUAD1
		RQ4	=	CTRAPAX	PRTQ	PTRAPAX
		RT3	=	CTRIAAX	PRTQ	PTRIAAX
5	=	QS4	=	CSHEAR	PID	PSHEAR
		QT4	=	CTWIST	PID	PTWIST
		B2A	=	CPIPE1	PB2A	PPIPE1
		TM6	=	CTRIM6	PTM6	PTRIM6
		QM8		----	PQM8	
		S04	=	CTETRA		
		S06	=	CWEDGE		
		S08	=	CIHEX1	PS82	PIHEX
		S020	=	CIHEX2	PS82	PIHEX
		AS	=	CELAS1	PAS	PELAS
		TB3	=	CTRIA3	PSHE	PSHELL
		QB4	=	CQUAD4	PSHE	PSHELL

NOTE: The nodes are defined in the same order as NASTRAN defines the nodes for these elements.

```
BEGIN      ELEMENT
GROUP       1
QM1        11      12    22    21    TO    13    14     24     23
REPEAT 3   10      10    10    10          10    10     10     10
GROUP       2
R2         11      12                TO    13    14
REPEAT 4   10      10                      10    10
GROUP      3
R2         11      21                TO    41    51     BY     10   10
REPEAT 3    1       1                       1     1             0    0
END
```

Figure 4. Element Generation Commands

After the elements are defined additional model information may be specified using the PROPERTY submodule or the FREEDOM and LOADS subprocessors. PROPERTY is used to specify the element sizes and material data. This data is applied in one of two ways; the first is through the ANISOTROPIC subprocessor and the second is through a DIRECT subprocessor. Layered composite material properties and sizes are specified in ANISOTROPIC. This is a very general function allowing the definition of up to ten different lamina orientations, each with different material properties and a different number of layers for each element. The commands shown in Figure 5 would make each of the quadrilateral elements generated in Figure 4 a layered composite element.

The orientation is 60/0/90/30 with 10 layers in the 0 and 90 directions and 15 layers in the 60 and 30 directions. All layers are made of composite material 1 specified by the CID command, with EL=20.0E6; ET=5.0E6; GL=9.0E6 and T=0.00525. The EL is the longitudinal modulus, ET is the transverse modulus, GL is the shear modulus and T is the thickness for the uniaxial lamina.

The DIRECT subprocessor is used to directly apply material and size data to the generated elements. Essentially, tables of element sizes and material properties are setup. These are then associated with the specific element groups to define that data for a group of elements. Figure 6 lists the commands used to define an isotropic material with E=10.0E6 and G=9.0E6 the Young's and shear moduli respectively; an axial rod area of 0.50, and then apply those values to the CROD elements generated in Figure 4.

```
BEGIN   PROPERTY
BEGIN   ANISO
BASIS   NODE    11          14
CID      1      EL=20.0E6   ET=5.0E6    T=0.00525    GL=9.0E6
GROUP    1
PLIES   10      CID 1       LANGLE=0    ELEM 1 TO 12
PLIES   10      CID 1       LANGLE=90   ELEM 1 TO 12
PLIES   15      CID 1       LANGLE=60   ELEM 1 TO 12
PLIES   15      CID 1       LANGLE=30   ELEM 1 TO 12
END
END
```

Figure 5. Composite Material Definition

```
BEGIN     PROPERTY
BEGIN     DIRECT
MAT1      1            E=10.0E6     G=9.0E6
PR2       1            0.50
GROUP     2            PR2 = 1      MID = 1
GROUP     3            PR2 = 1      MID = 1
END
END
```

Figure 6. Isotropic Material Definition Commands

In addition to generating node coordinate data CADS has functions for defining node constraints and external load information through the FREEDOM and LOADS subprocessors. FREEDOM is used to constrain and unconstrain degrees of freedom of the model's nodes. LOADS is used to apply external forces and moments to the nodes. Figure 7 lists the commands needed to fix the top line of nodes on the plate in Figure 4, suppress all of the rotational degrees of freedom in the model, and finally, load the bottom line of nodes with a force of 500 units in the y-direction.

```
BEGIN      NODE
BEGIN      FREEDOM
SUPPRESS   TX       TY   TZ   RX     RY   RZ     NODE  11  TO  51  BY  10
SUPPRESS   RX       RY   RZ   NODE 11 TO 54
END
BEGIN      LOAD
CASE 1
FL         500.0
LOAD       Y=1.0    NODE 14 TO 54 BY 10
END
END
```

Figure 7. Constraints and Load Generation

By using the commands in the NATURAL processor complex models can be effectively generated. Detailed descriptions of all of these functions are provided in section 4.0 of the "User's Guide."

3.0 DISPLAY FUNCTIONS

The DISPLAY functions provide the major interactive graphics interface between the user and the CADS software. A wide range of functions is provided including shrink, zoom, rotate, and label options to allow the user complete control over the model display.

In order to provide complete user control over the display, SET functions are included which define sets of nodes and elements to be plotted. The user may define any amount of information from a single node or element through the complete model for display. Once defined this information is passed to the DISPLAY routines for actual display. At that time the DISPLAY functions are specified to display the model as the analyst requests.

Some of the display commands available to the user are ROTATE (to rotate the display about any coordinate axis), SHRINK (to shrink each element about its centroid), and WINDOW, PORT and VIEW (to window in or enlarge a specified section of the display). In addition, a variety of labeling commands to label nodes, elements, materials, and other model attributes are available to the user. Full details and examples of the DISPLAY commands are provided in Volume II, "User's Guide," for the CADS software.

In order to highlight some of the DISPLAY module functions, a series of examples are provided in Figures 8 through 14. Figure 8 is a plot of the small plate model whose generation was illustrated in Figures 3 through 7. The node numbers, element numbers, and element labels are shown. The BREAK option which plots each element separated from the others was used on the display. The detailed composite layer data is shown in Figure 9. The element number, layer angle, number of plies and material are displayed for a selected number of elements.

An anisotropic wing box model is shown in Figures 10 through 14. The model is in NASTRAN format and is one of the test cases used during the development of the CADS software. Figure 10 is the entire model, rotated about the Y and Z axes by 45 degrees. Note the default margin information used to document the display. This same plot is shown with hidden lines removed in Figure 11. This is a much clearer display. The upper cover elements from the root to the break were extracted to demonstrate some of the label functions. Figure 12 shows the element numbers, element sizes, and node numbers on individually plotted elements. The rear spar webs were added to the cover elements and displayed with the element name and color pattern options in Figure 13. The VIEW processing is shown in Figure 14. VIEW is used to zoom up on a selected area of the plot and re-plot it on the existing plot. This can be used to closely examine details of a model. Other zoom options are also available in CADS as discussed in the "User's Guide."

As a part of either the display functions or the Executive Monitor, an editor is available to provide basic model editing capabilities interactively to the analyst. These include adding or deleting elements, nodes, and attribute definitions for the elements, as well as analysis control cards. For instance, the analyst can add a new node to the model, redefine the material properties for a group of elements, and change the connectivity for a specified element using the EDIT module. The CASE, ELEMENT, MATERIAL, NODE, and PROPERTY processors of the editor perform these functions. Standard DELETE, CHANGE, ADD, and LIST capabilities are provided for the model data. In addition, the CASE processor is available to change the analysis program's executive and case control cards.

By making use of the variety of the SET, DISPLAY and EDIT capabilities of the CADS program, the analyst can rapidly verify and debug the finite element model of a given structure. Care must be exercised in the definition of nodes and elements, however, to ensure that the finite element model effectively represents the structure.

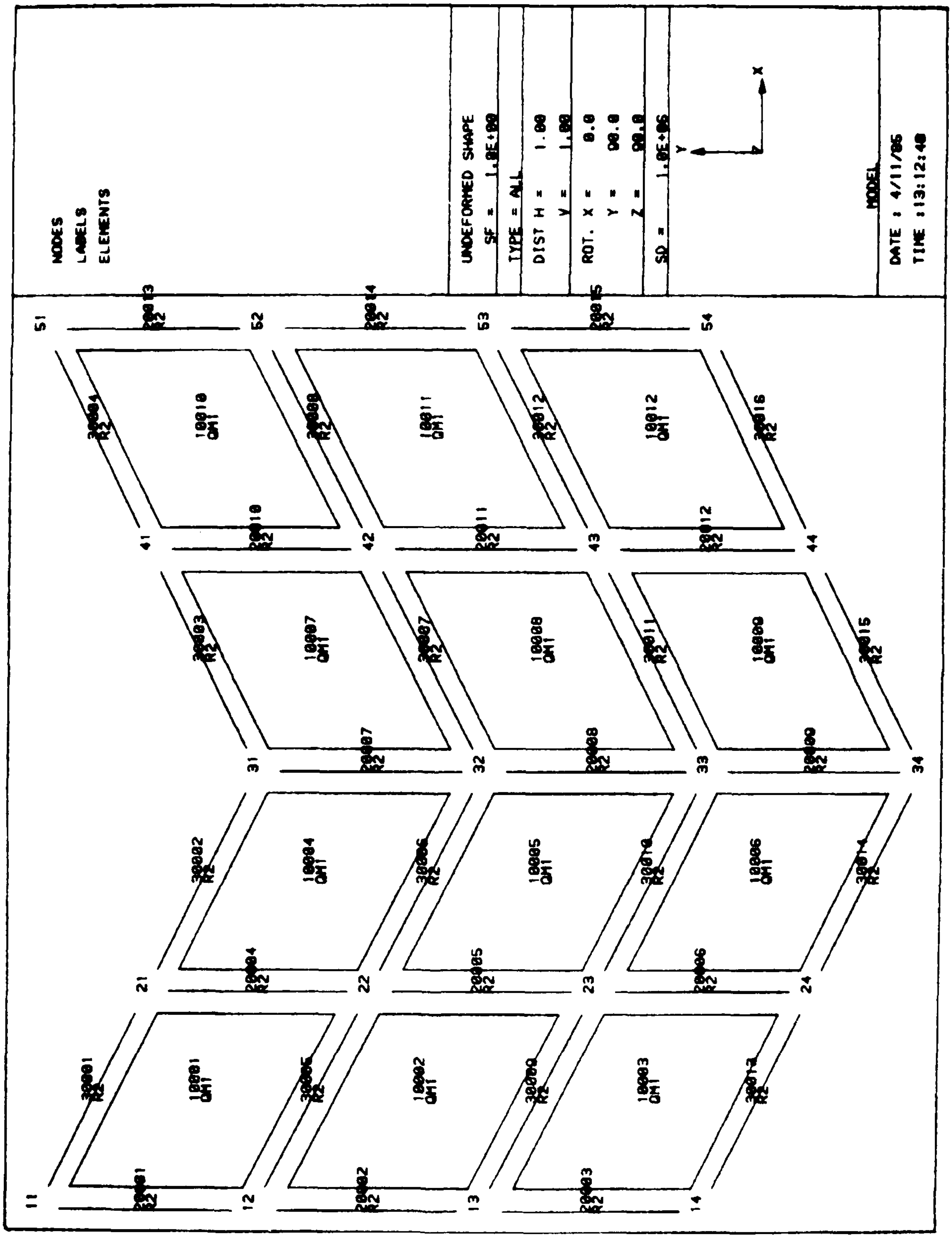


Figure 8. Plot of Plate Generation Model

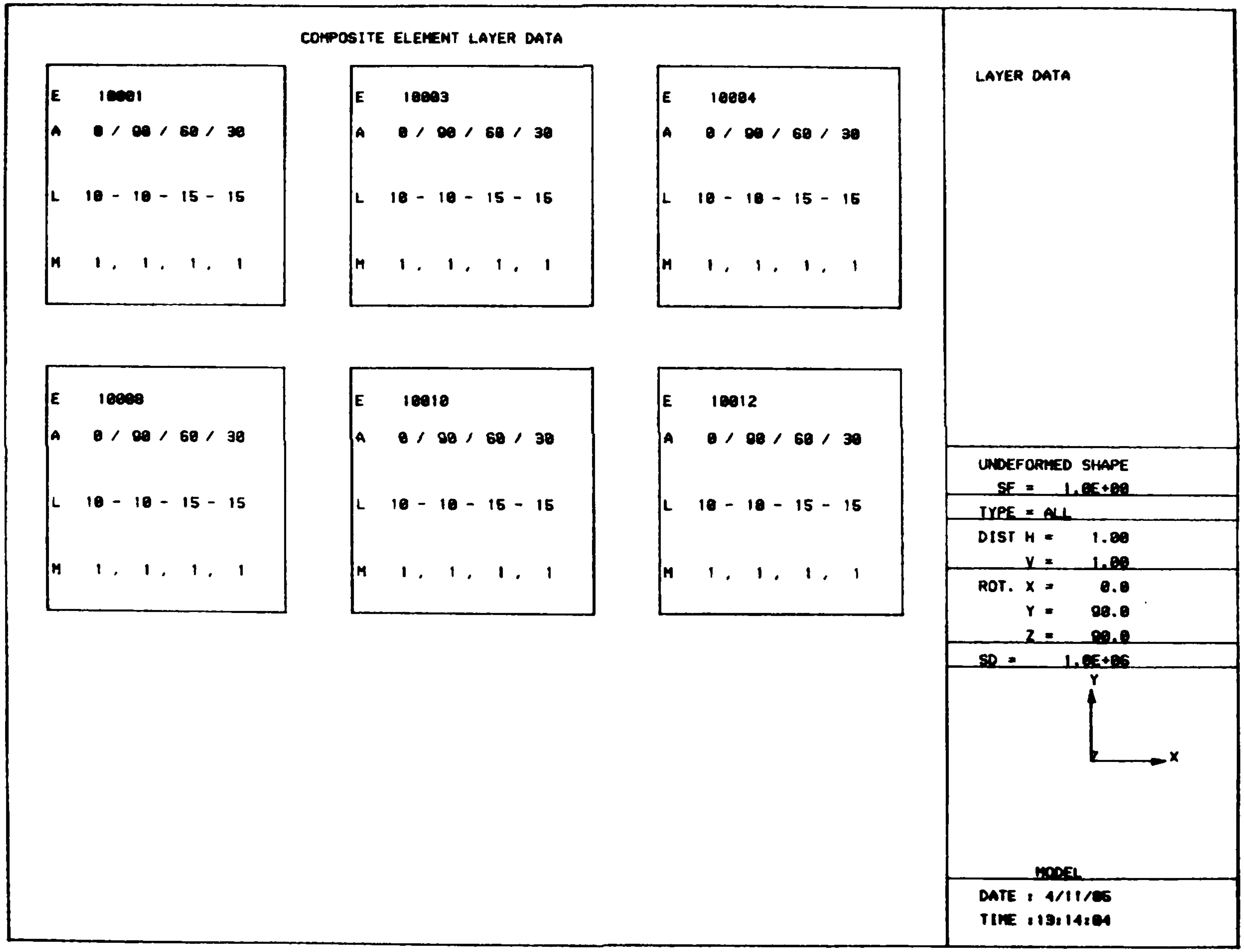


Figure 9. Details of Selected Layered Composite Elements

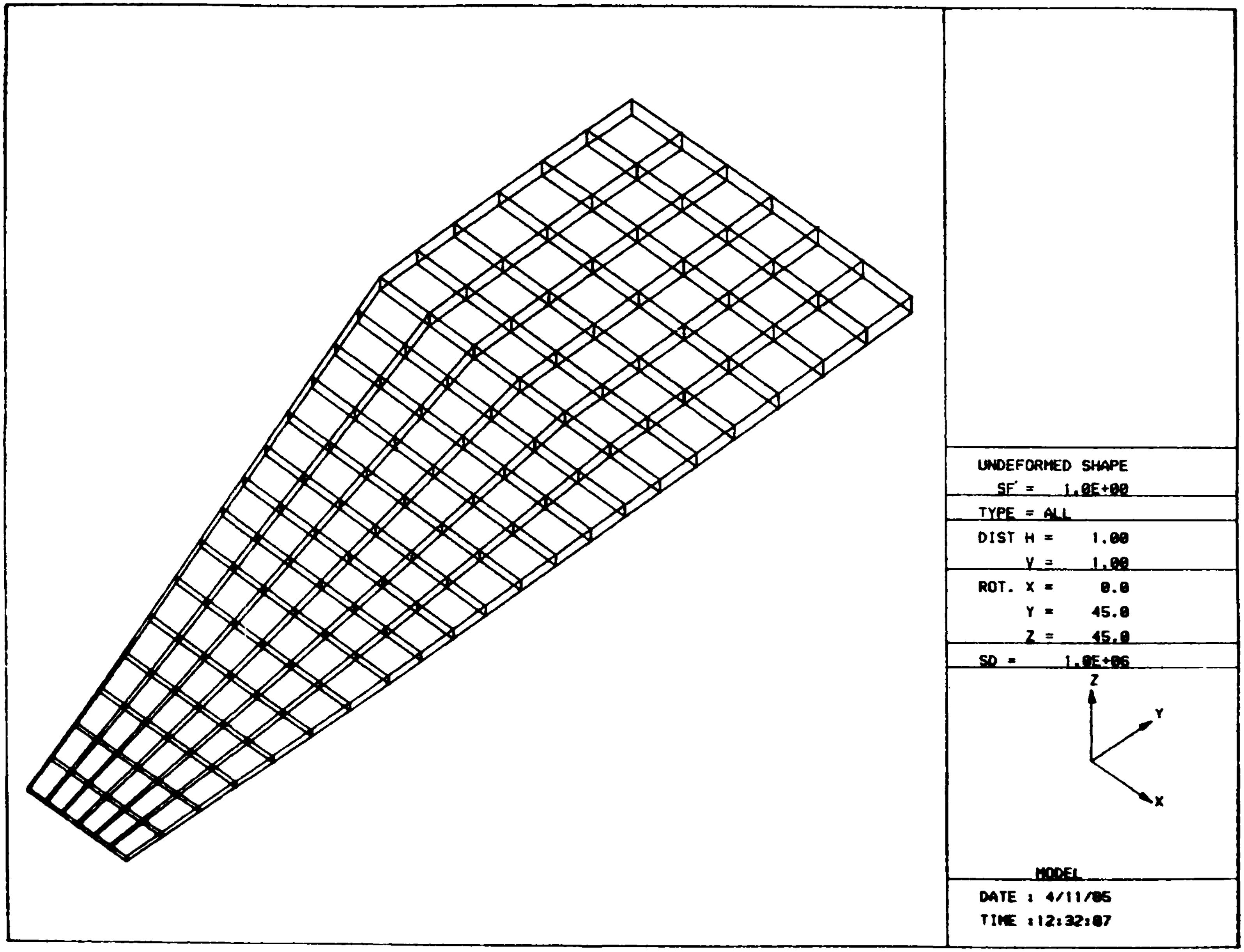


Figure 10. Wing Box Structure Model

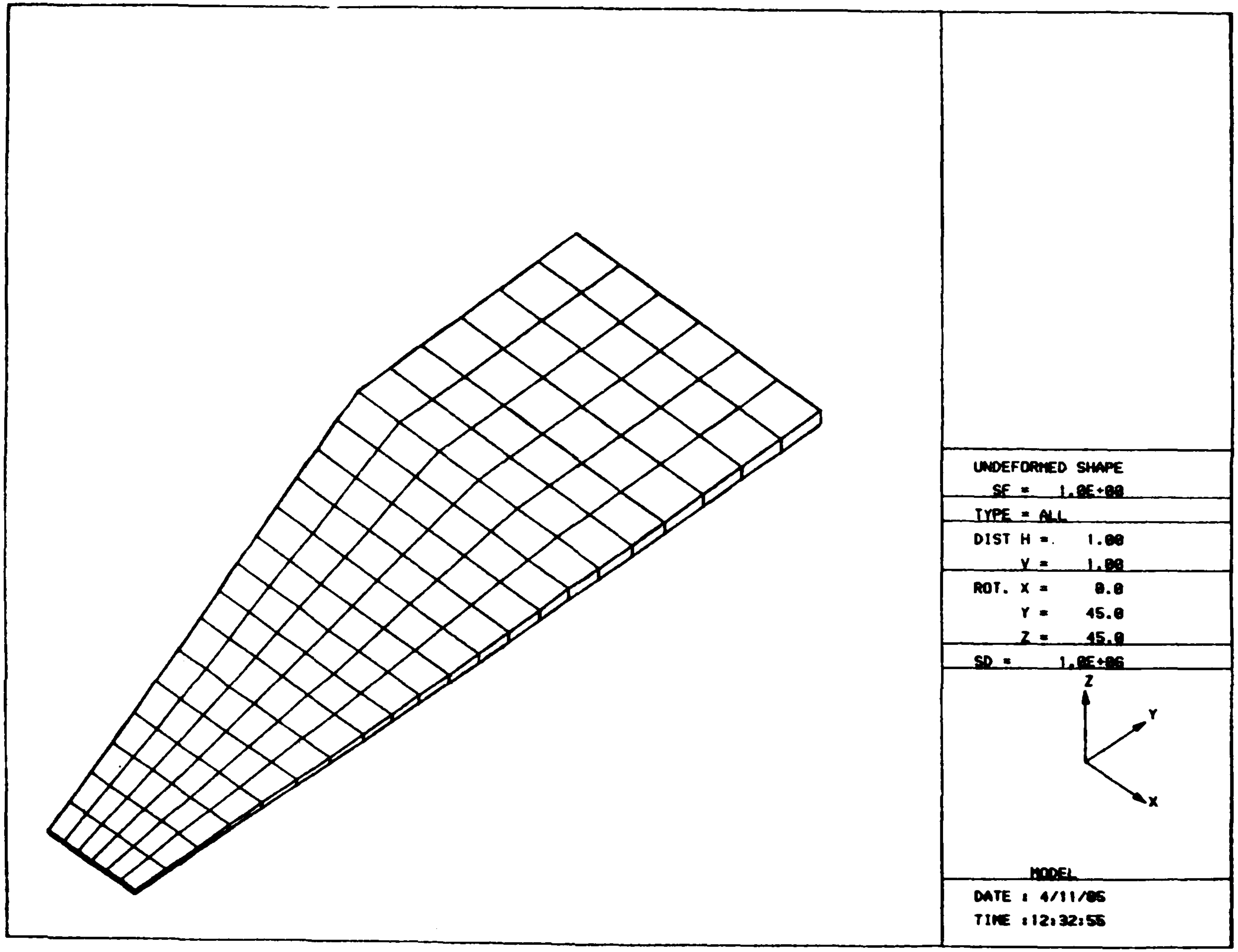


Figure 11. Hidden Line Display of Wing Box

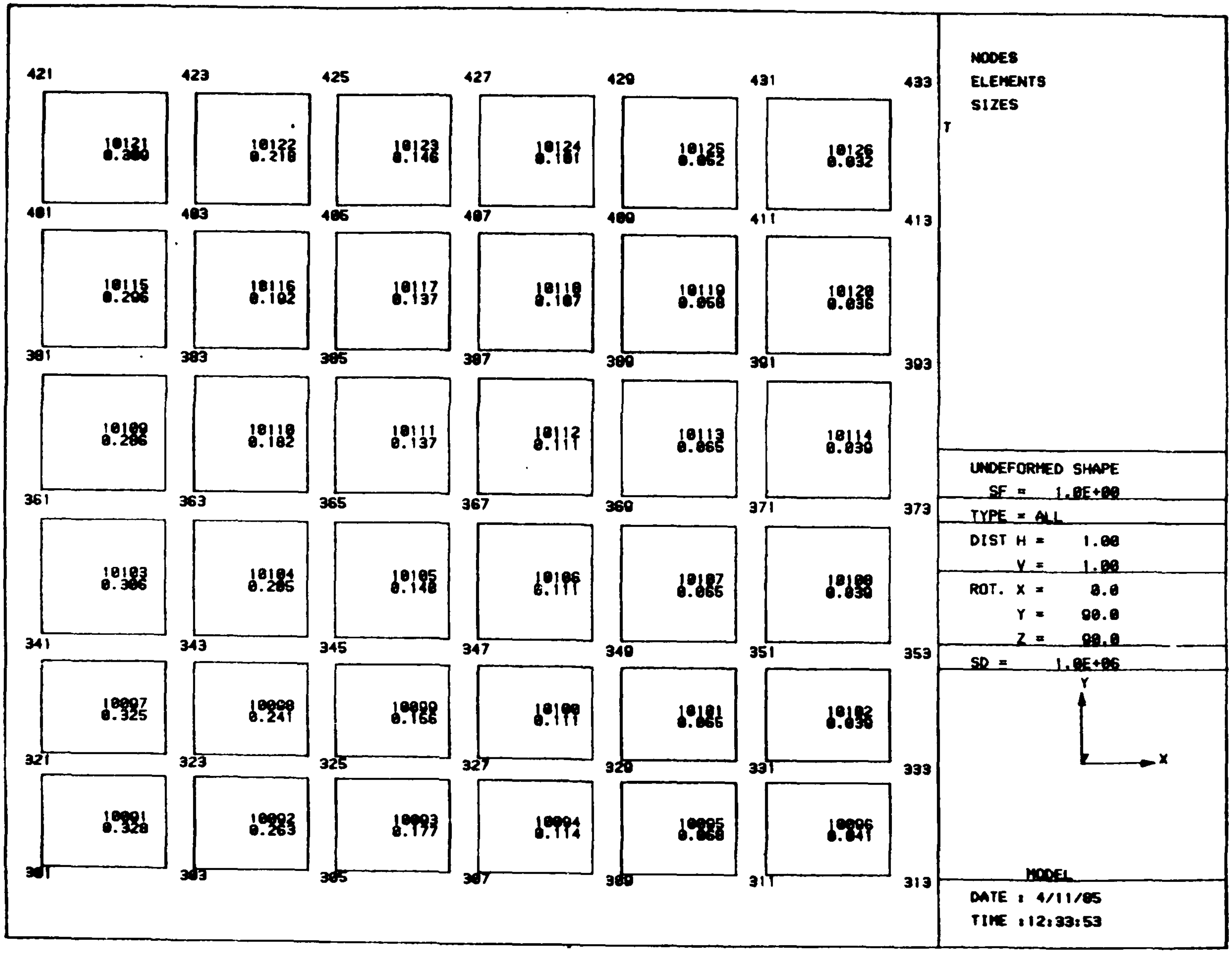


Figure 12. Element Numbers, Sizes, Node Number Display

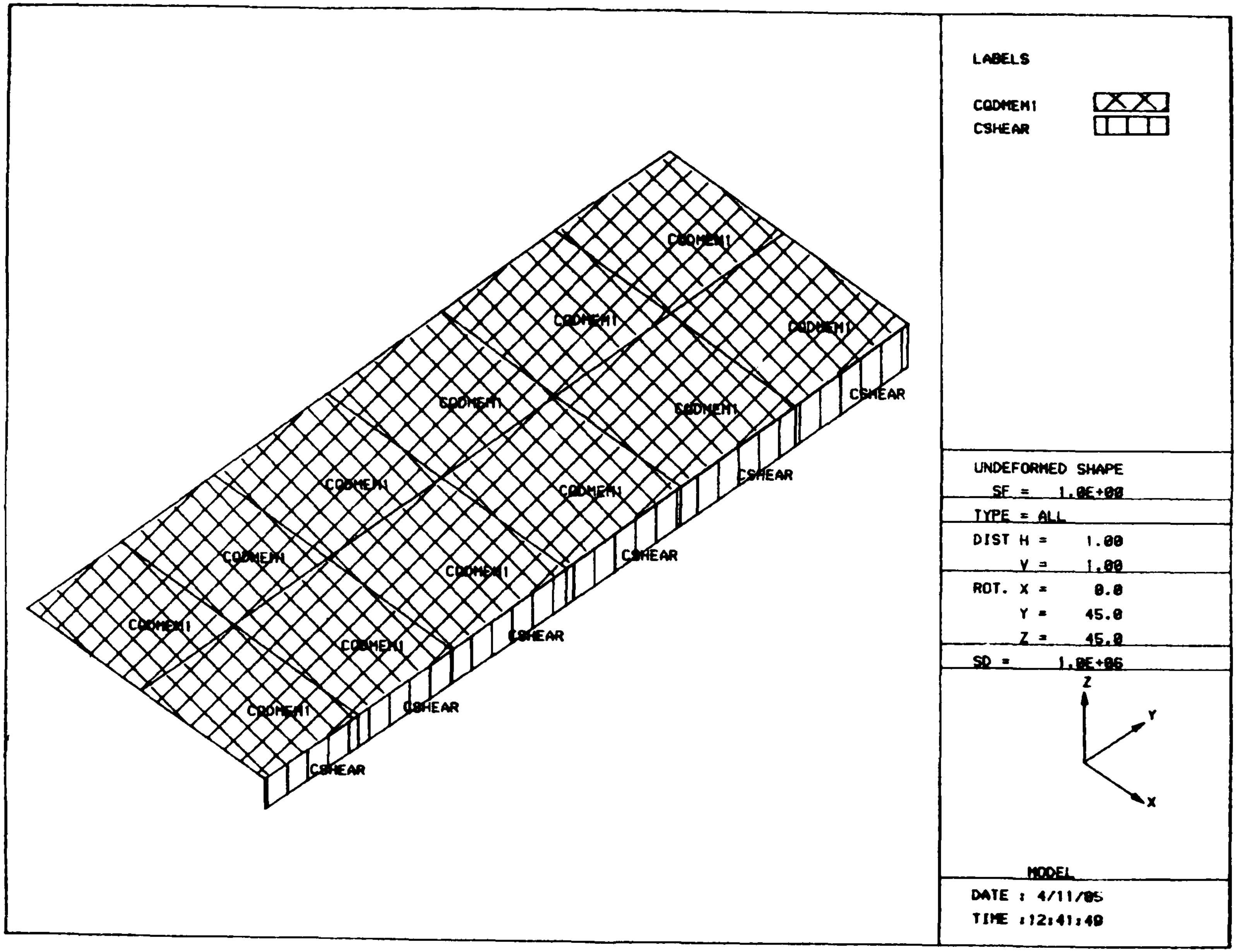


Figure 13. Color Pattern and Label Option Display

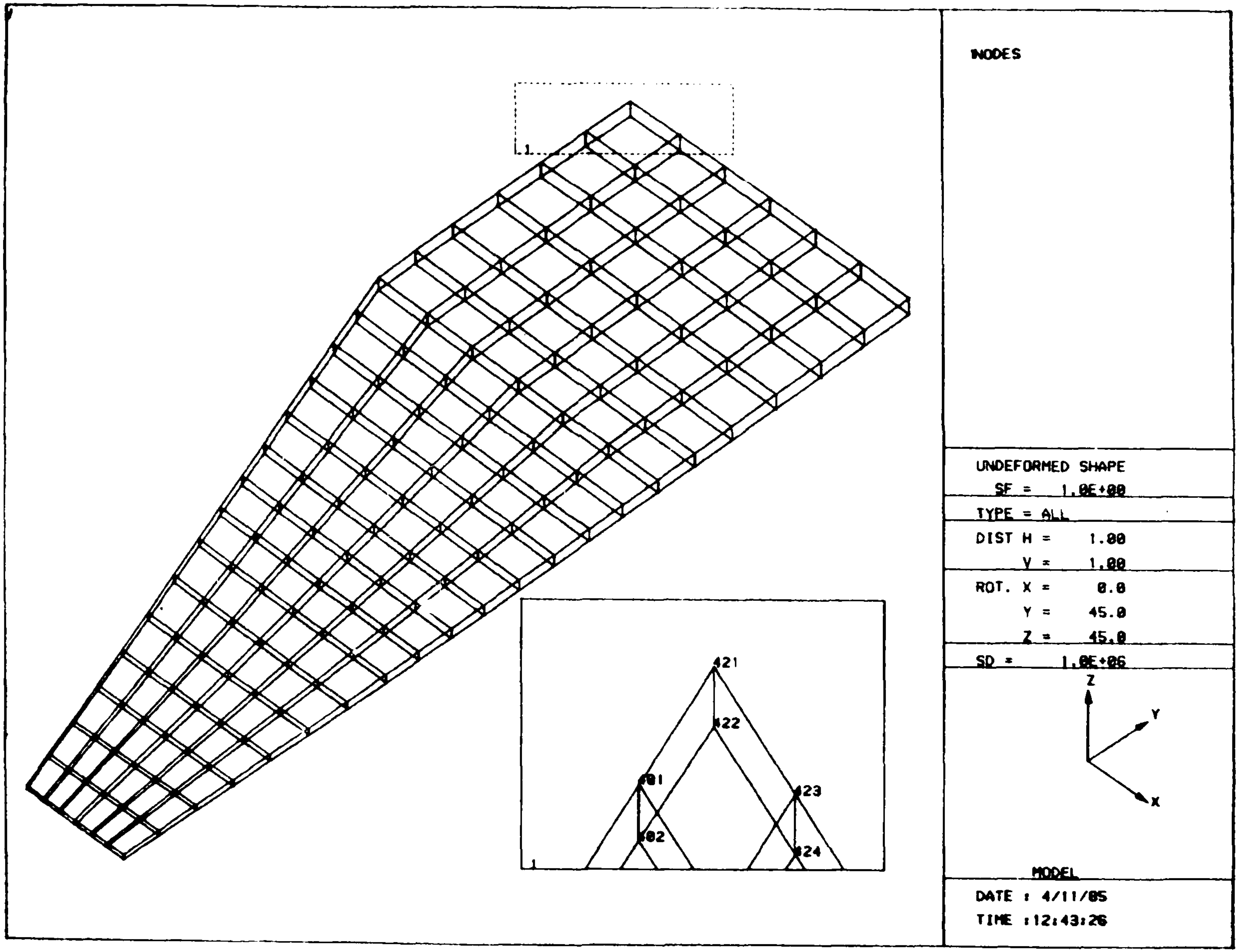


Figure 14. VIEW Processing Example

4.0 POSTPROCESSOR FUNCTIONS

The postprocessor functions in CADS are composed of an interface program which stores the output analysis results to a data base as well as routines which operate upon that data base for display purposes.

The CADSPP program reads a card image data set containing the appropriate finite element analysis outputs and stores those data in a direct access file usable by the CADS software. Once a POST data base has been established, the user may retrieve information from this file for further processing.

CADSPP is an interactive program resident on the DEC VAX equipment. It prompts the user for file names, for the input and output files and then processes the analysis program output information. A wide variety of data elements are stored depending upon the element type, analysis program and types of output data. CADSPP translates the formatted analysis output files into a random access POST data base. This POST data base is then used by CADS for interactive displays of the analysis results.

The details of the types of data understood by CADSPP are shown in Tables 2, 3, and 4. The NASTRAN, ANALYZE, and OPTSTAT result data is listed in these tables.

Table 2 details the types of data stored for NASTRAN. Analysis results can be stored for either dynamic or static executions of NASTRAN. For dynamic results time step information is also stored and is needed to access a particular set of data. For NASTRAN, displacement and eigenvector output results are stored for the rotations and translations at each node for each subcase or dynamic time step. For the element force output data all of the components valid for the elements supported by CADS are stored. The components are named with up to 4 character names which follow the NASTRAN naming convention. Again for element stress outputs all of the valid NASTRAN components are stored for those elements supported by CADS.

TABLE 2

VALID COMPONENT TYPES FOR NASTRAN DATA

MODE	TYPE NAME	COMPONENT	DESCRIPTION
DISPLACE or EIGEN	NODE	TX	Translation in the X direction
		TY	Translation in the Y direction
		TZ	Translation in the Z direction
		RX	Rotation in the X direction
		RY	Rotation in the Y direction
		RZ	Rotation in the Z direction
FORCE	CROD	F	Axial force
		T	Torque
	CONROD	see CROD	
	CBAR	MA1	Bending moment at A1
		MA2	Bending moment at A2
		MB1	Bending moment at B1
		MB2	Bending moment at B2
		SHR1	Shear at point 1
		SHR2	Shear at point 2
		F	Axial force
		T	Torque
	CTRMEM	none	
	CQDMEM1	none	
	CTRIA2	MX	X - moment
		MY	Y - moment
		TM	Twist moment
		SHRX	X - shear
		SHRY	Y - shear
	CQUAD2	see CTRIA2	
	CTRIA1	see CTRIA2	
	CQUAD1	see CTRIA2	
	CTRAPAX	R1	Radial force at ring 1
		R2	Radial force at ring 2
		R3	Radial force at ring 3
		R4	Radial force at ring 4
		T1	Tangential force at ring 1
		T2	Tangential force at ring 2
		T3	Tangential force at ring 3
		T4	Tangential force at ring 4
		Z1	Axial force at ring 1
		Z2	Axial force at ring 2

TABLE 2 (Continued)

VALID COMPONENT TYPES FOR NASTRAN DATA

CTRIAAX	See CTRAPAX	
CSHEAR	F41	Force from point 4 to 1
	F21	Force from point 2 to 1
	F12	Force from point 1 to 2
	F32	Force from point 3 to 2
	F23	Force from point 2 to 3
	F43	Force from point 4 to 3
	F34	Force from point 3 to 4
	F14	Force from point 1 to 4
	K1	Kick force at point 1
	K2	Kick force at point 2
	K3	Kick force at point 3
	K4	Kick force at point 4
	V12	Shear from point 1 to 2
	V23	Shear from point 2 to 3
	V34	Shear from point 3 to 4
	V41	Shear from point 4 to 1
CTWIST	see CSHEAR	
CPIPE1	see CBAR	
CTRIM6	none	
QM8	none	
CTETRA	none	
CWEDGE	none	
CIHEX1	none	
CIHEX2	none	
CELAS1	F	Axial force
CTRIA3	FX	Force in the X direction
	FY	Force in the Y direction
	FXY	Force in the XY direction
	MX	Moment about X
	MY	Moment about Y
	MXY	Moment about XY
	VX	Shear in X
	VY	Shear in Y
CQUAD4	see CTRIA3	

TABLE 2 (Continued)

VALID COMPONENT TYPES FOR NASTRAN DATA

STRESS	CROD	A	Axial stress
		ASM	Axial margin of safety
		T	Torsional stress
		TSM	Torsional margin of safety
	CONROD	see CROD	
	CBAR	SA1	Stress at A1
		SA2	Stress at A2
		SA3	Stress at A3
		SA4	Stress at A4
		A	Axial stress
		SAMX	Maximum stress at A
		SAMN	Minimum stress at A
		SMT	Margin of safety in tension
		SB1	Stress at B1
		SB2	Stress at B2
		SB3	Stress at B3
		SB4	Stress at B4
		SBMX	Maximum stress at B
		SBMN	Minimum stress at B
		SMB	Margin of safety in compression
	CTRMEM	SX	Stress in the X direction
		SY	Stress in the Y direction
		SXY	Stress in the XY direction
		ANG	Principal stress angle
		MAJP	Major principal stress
		MINP	Minor principal stress
		MAXS	Maximum shear stress
	CQDMEM1	See CTRMEM	
	CTRIA2	FDB	Fiber distance bottom side
		SXB	X-stress bottom side
		SYB	Y-stress bottom side
		SXYB	XY-stress bottom side
		ANGB	Principal stress angle bottom
		SMJB	Major principal stress bottom
		SMNB	Minor principal stress bottom
		SMXB	Maximum shear stress bottom
		FDT	Fiber distance top side
		SXT	X-stress top side
		SYT	Y-stress top side
		SXYT	XY-stress top side
		ANGT	Principal stress angle top
		SMJT	Major principal stress top
		SMNT	Minor principal stress top
		SMXT	Maximum shear stress top

TABLE 2 (Continued)

VALID COMPONENT TYPES FOR NASTRAN DATA

CQUAD2	See CTRIA2	
CTRIA1	FD1	Fiber distance one side
	SX1	X-stress one side
	SY1	Y-stress one side
	SXY1	XY-stress one side
	ANG1	Principal stress angle one side
	SMJ1	Major principal stress one side
	SMN1	Minor principal stress one side
	SMX1	Maximum shear stress one side
	FD2	Fiber distance two side
	SX2	X-stress two side
	SY2	Y-stress two side
	SXY2	XY-stress two side
	ANG2	Principal stress angle two side
	SMJ2	Major principal stress two side
	SMN2	Minor principal stress two side
	SMX2	Maximum shear stress two side
CQUAD1	See CTRIA1	
CTRAPAX	R	Radial stress for the element
	Z	Axial stress for the element
	T	Tangential stress for the element
	ZR	Shear stress for the element
	RT	Shear stress for the element
	ZT	Shear stress for the element
CTRIAAX	See CTRAPAX	
CSHEAR	MAXS	Maximum shear stress
	AVRS	Average shear stress
	SM	Margin of safety
CTWIST	see CSHEAR	
CPIPE1	see CBAR	
CTRIM6	none	
QM8	none	
CTETRA	none	
CWEDGE	none	

TABLE 2 (Concluded)

VALID COMPONENT TYPES FOR NASTRAN DATA

CIHEX1	SX	Normal X stress
	SXY	Shear XY stress
	SMX	First principal
	SMX1	First principal X cosine
	SMX2	Second principal X cosine
	SMX3	Third principal X cosine
	MS	Mean stress
	OSS	Octahedral shear stress
	SY	Normal Y
	SYZ	Shear YZ
	SMY	Second Principal
	SMY1	First principal Y cosine
	SMY2	Second principal Y cosine
	SMY3	Third principal Y cosine
	SZ	Normal Z
	SZX	Shear ZX
	SMZ	Third principal
	SMZ1	First principal Z cosine
	SMZ2	Second principal Z cosine
	SMZ3	Third principal Z cosine
CIHEX2	see CIHEX1	
CELAS1	S	Axial stress
CTRIA3	see CTRIA1	
CQUAD4	see CTRIA1	

TABLE 3

VALID COMPONENT TYPES FOR ANALYZE DATA

MODE	TYPE NAME	COMPONENT	DESCRIPTION
DISPLACE	NODE	TX	Translation in the X direction
		TY	Translation in the Y direction
		TZ	Translation in the Z direction
STRESS	2	SX	X local element stress
		ENER	Total strain energy in the element
		MS	Margin of safety
	3	SX	X local element stress
		SY	Y local element stress
		SXY	XY local element stress
		EFS1	Effective stress ratio - 1
		ENER	Total strain energy in the element
		MS	Margin of safety
	4	SX	X local element stress
		SY	Y local element stress
		SXY	XY local element stress
		EFS1	Effective stress ratio - 1
		EFS2	Effective stress ratio - 2
		EFS3	Effective stress ratio - 3
		EFS4	Effective stress ratio - 4
		ENER	Total strain energy in the element
		MS	Margin of safety
	5	SXY	XY local element stress
		EFS1	Effective stress ratio - 1
		EFS2	Effective stress ratio - 2
		EFS3	Effective stress ratio - 3
		EFS4	Effective stress ratio - 4
		ENER	Total strain energy in the element
		MS	Margin of safety

NOTE: These are the only valid element types for ANALYZE.

TABLE 4

VALID COMPONENT TYPES FOR OPTSTAT DATA

MODE	TYPE NAME	COMPONENT	DESCRIPTION
DISPLACE	NODE	TX	Translation in the X direction
		TY	Translation in the Y direction
		TZ	Translation in the Z direction
STRESS	2	SX	X local element stress
		EFS	Effective stress ratio
		ALS1	Longitudinal tension allowable
		ALS2	Ratio of long. compress. to ALS1
		ENER	Total strain energy in the elem.
		OPTT	Optimized area value
	3	SX	X local element stress
		SY	Y local element stress
		SXY	XY local element stress
		EFS	Effective stress ratio
		ALS1	Longitudinal tension allowable
		ALS2	Ratio of long. compress. to ALS1
		ALS3	Ratio of trans. tension to ALS1
		ALS4	Ratio of trans. compression to ALS1
		ALS5	Ratio of shear allowable to ALS1
		ENER	Total strain energy in the elem.
		OPTT	Optimized thickness value
	4	SX	X local element stress
		SY	Y local element stress
		SXY	XY local element stress
		EFS	Effective stress ratio
		ALS1	Longitudinal tension allowable
		ALS2	Ratio of long. compress. to ALS1
		ALS3	Ratio of trans. tension to ALS1
		ALS4	Ratio of trans. compression to ALS1
		ALS5	Ratio of shear allowable to ALS1
		ENER	Total strain energy in the elem.
		OPTT	Optimized thickness value
	5	SXY	XY local element stress
		EFS	Effective stress ratio
		ALS1	Longitudinal tension allowable
		ALS2	Ratio of long. compress. to ALS1
		ALS3	Ratio of trans. tension to ALS1
		ALS4	Ratio of trans. compression to ALS1
		ALS5	Ratio of shear allowable to ALS1
		ENER	Total strain energy in the elem.
		OPTT	Optimized thickness value

TABLE 4 (Concluded)

VALID COMPONENT TYPES FOR OPTSTAT DATA

3	LAM	Total number of layers
	THK	Total thickness on 0° direction
	AEX	Proportion of fibers in 0°
	THK9	Total thickness in 90° direction
	AEY	Proportion of fibers in 90°
4	LAM	Total number of layers
	THK	Total thickness in 0° direction
	AEX	Proportion of fibers in 0°
	THK9	Total thickness in 90° direction
	AEY	Proportion of fibers in 90°

NOTE: These are the only valid element types for OPTSTAT. The second set of components for the type 3 and 4 elements is output when layered composite elements are used. These components replace the X, Y, and XY local element stress components.

Table 3 contains the DISPLACEMENT and STRESS values supported for ANALYZE. All of the components output by ANALYZE are available. These include the translation displacement components for each node as well as various local element stresses and effective stress ratios for the elements. The element strain energy and margin of safety information is also available. Data is stored for each load case run in ANALYZE.

Table 4 details the OPTSTAT components. These are stored for the optimum results as determined during an optimization run using OPTSTAT. Translational displacements, local stresses, various stress ratios, and element strain energy information is stored for OPTSTAT.

In order to interactively display any of these results, CADS must be entered and the appropriate POST and GEOMETRY data bases attached. The SET module is then entered and the model set to be displayed is selected and passed to the DISPLAY module. In the DISPLAY module, the ATTRIBUTE processor is used to define the analysis program type, the mode of data, and the component names of the results to be selected. This information is then used by the DISPLAY module to provide interactive plots of the analysis results. Four basic types of displays are available for plotting the analysis results.

These are value plots, contour plots, deformed plots and X-Y graph plots. An example of each type is shown in Figures 15 through 18.

A display of the element X and XY stresses is shown in Figure 15. These are the upper cover elements from the root to the break in the wing box model shown in Figure 10. Note that the component name and value is shown on each element and that with even a very few elements data will be quickly overwritten. These value plots are available for any element stress, force, node displacement, or eigenvector data.

A contour plot of the element X stresses for the entire upper cover of the wing box model is shown in Figure 16. CADS automatically determines the levels at which to draw the contour lines although the user may override them if needed. Contour plots can be used to quickly determine areas to be studied in greater detail before specifying detailed value plots.

A deformed on an undeformed plot is shown in Figure 17. The deformed shape is drawn in dashed lines with the undeformed shape in solid lines. Either displacement or eigenvector data may be plotted in the deformed mode. In addition, the user can plot just a deformed shape and can scale the deformations by any real number using a scale command.

Figure 18 is an example of a X-Y plot of the Z-displacement versus the wing span position. Up to five curves may be plotted on any one X-Y graph. CADS automatically makes a legend, scales, and grids the display. An option is also available for the user to directly input X and Y values for plotting so that CADS can be used to just plot X-Y graphs of some set of input data.

Finally, all of the standard display commands are available to the user while performing postprocessing functions. These include windows, rotations, attribute labeling, and display zooming so that detailed interactive studies of the analysis results can be made.

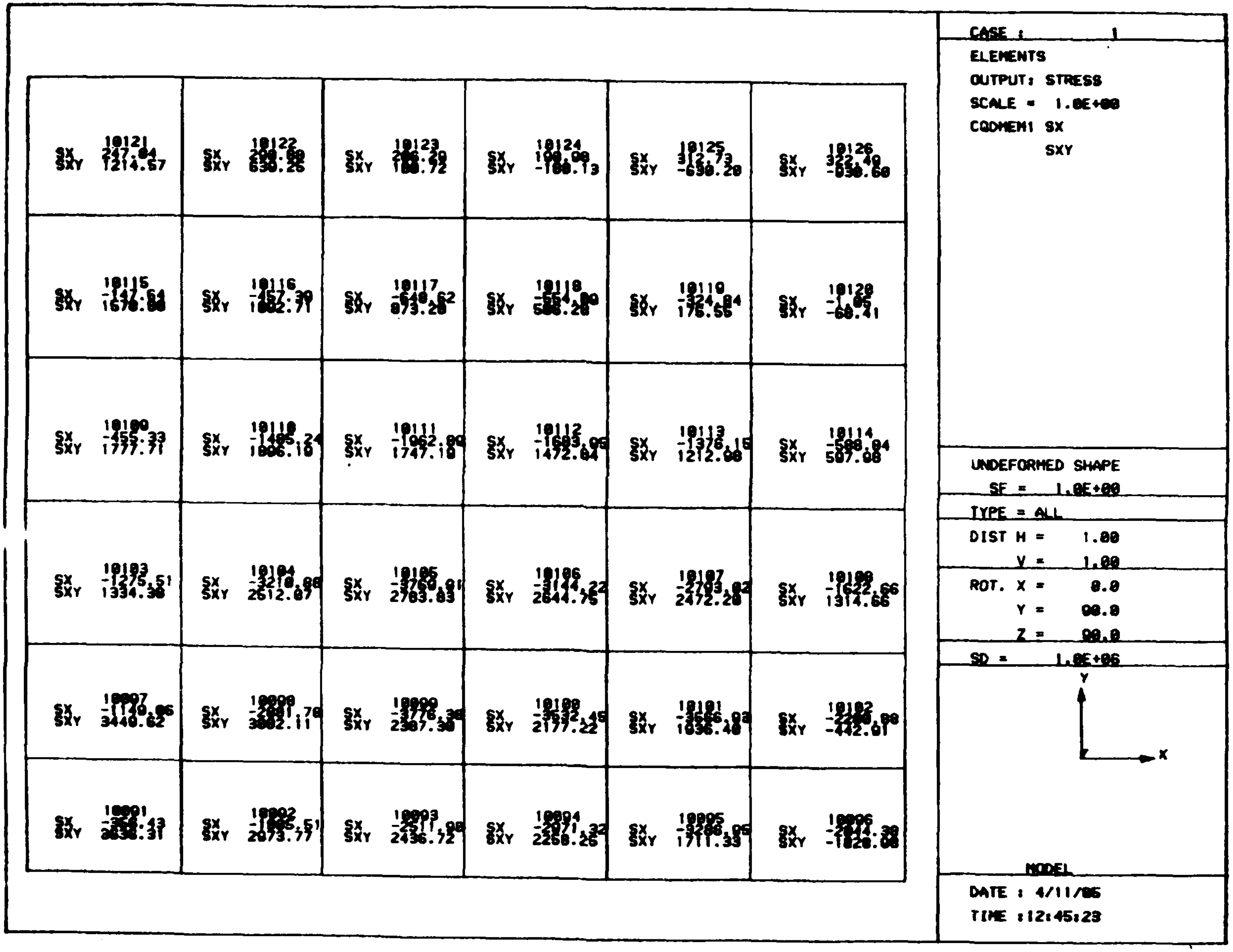


Figure 15. SX and SXY Stress Values on Elements

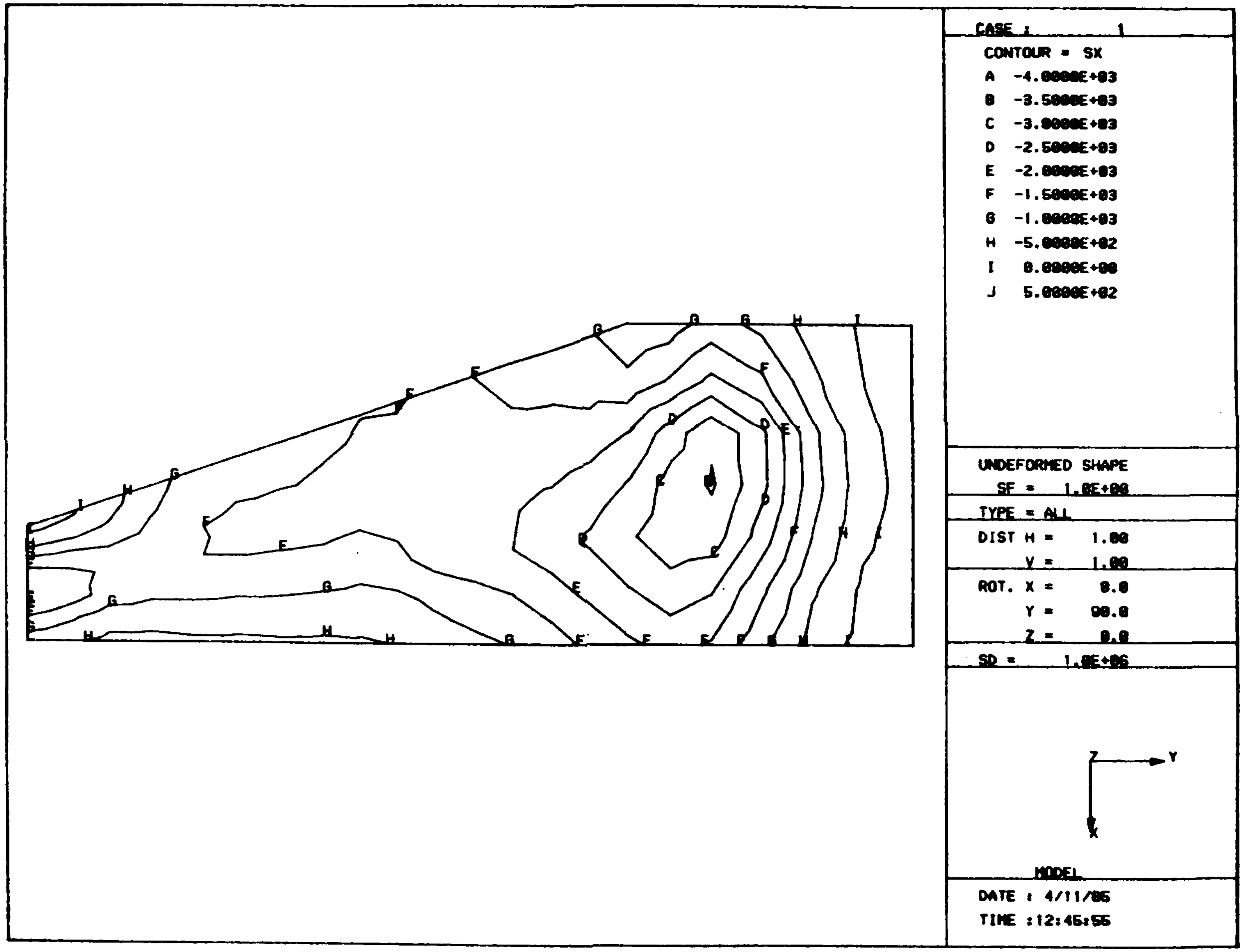


Figure 16. SX Stress Contour Plot for Entire Upper Cover

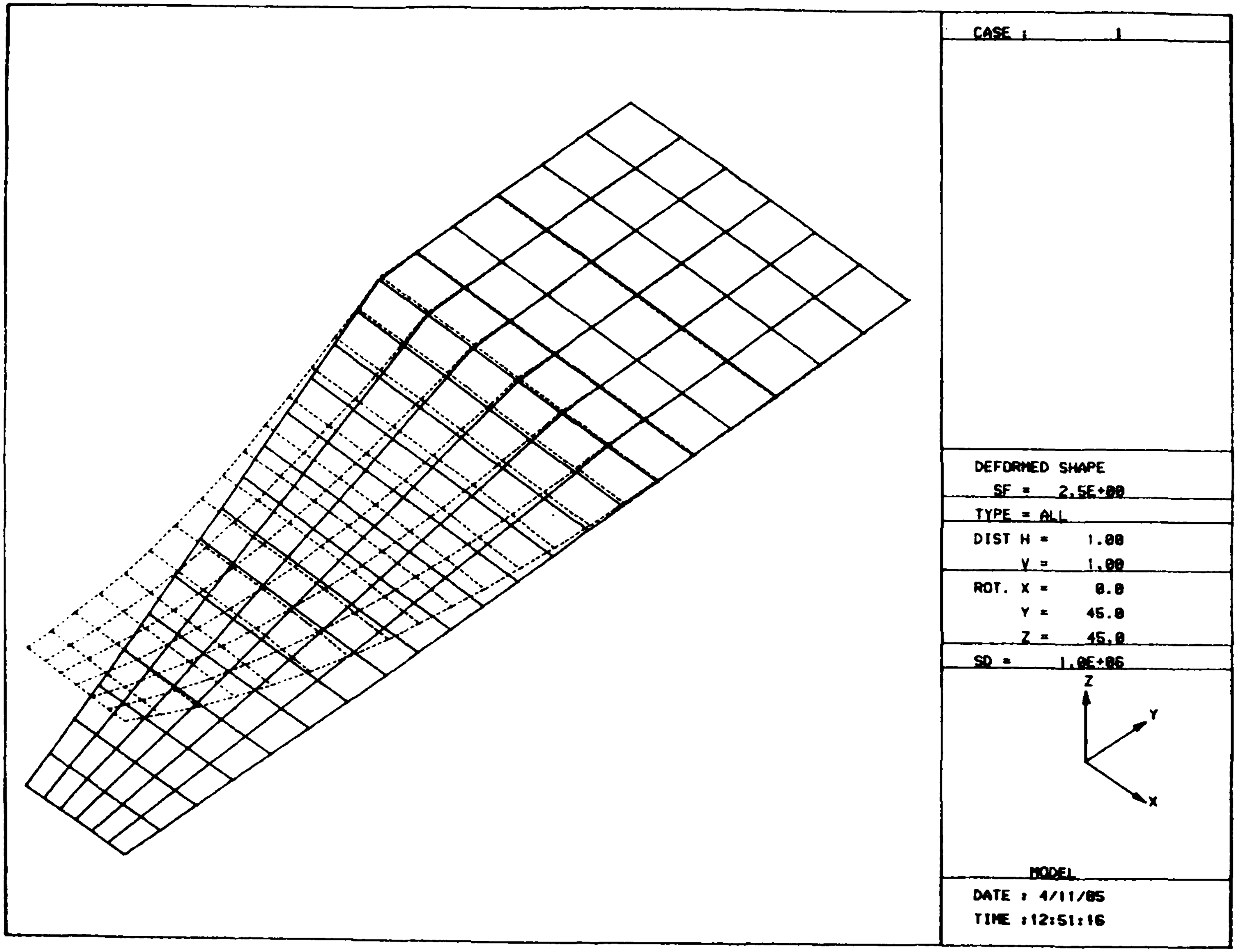


Figure 17. Deformed on Undeformed Shape Plot

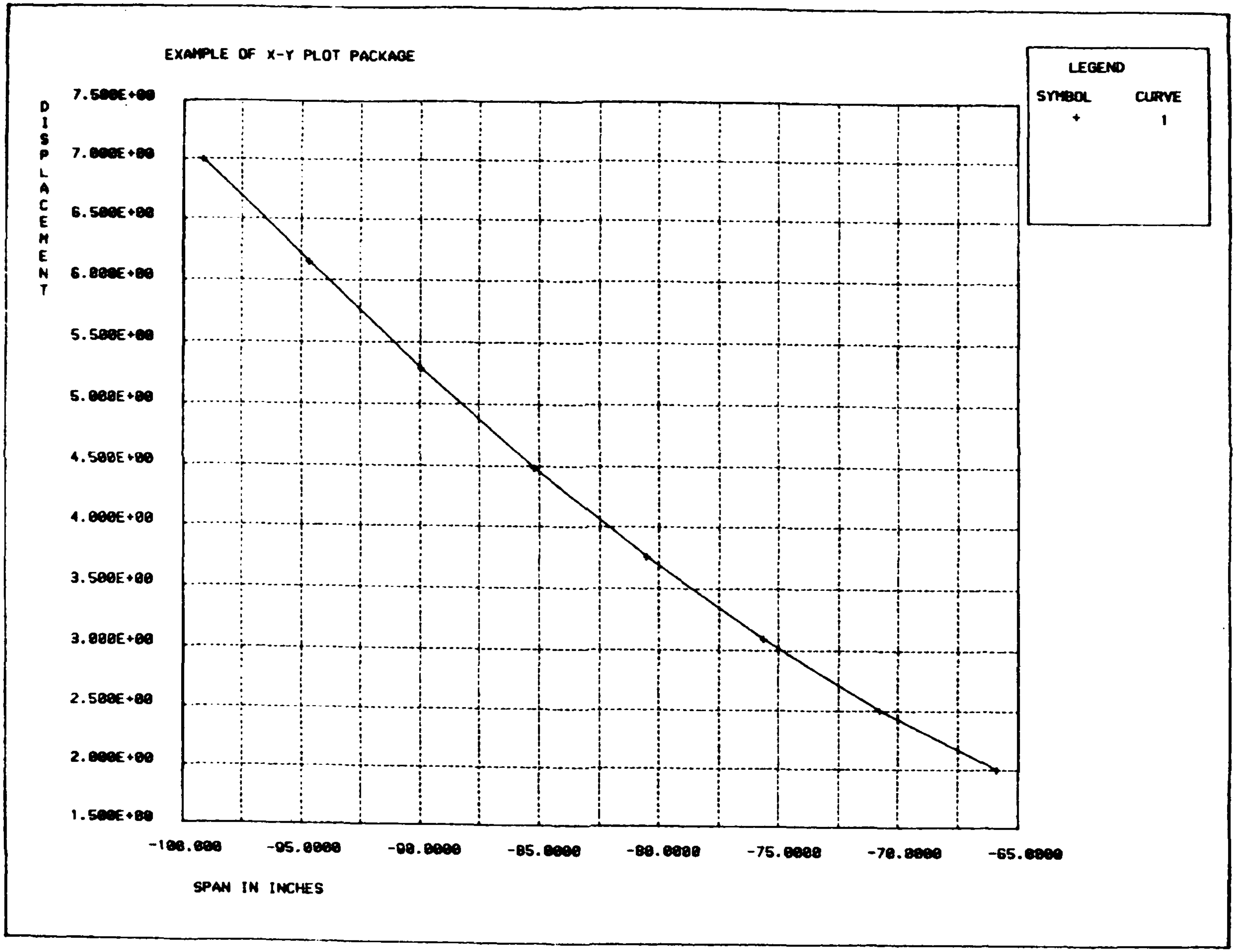


Figure 18. Graph of Z-Displacements versus Wing Span

5.0 CONCLUSIONS

The primary objective in developing the CADS software was to provide a versatile, user-friendly, finite element modeling program which was easily maintained and readily transportable. This objective was met by developing a code with a complete range of user functions, easily accessible, and capable of supporting complex modeling requirements. A commercially available, device-independent graphics package, DI-3000, standard FORTRAN 77 coding, and minimal system dependent routines are some of the techniques used to make the code easy to maintain and transport. In fact, much of the program has been readily transported to and executed on the DEC VAX 11/780 and IBM 4341 and 3081 processors.

Some of the different and more important aspects of the CADS software are its support for layered composite elements, its hidden line display functions, and its extensive support for NASTRAN, ANALYZE, and OPTSTAT analysis output data. In addition, more typical generation, display, and bulk data translation capabilities are provided.

As with any complex software system (which is used in a variety of applications) long term enhancements and support will be required for the program. As experience is gained, the user community will have requests for new features, additional element libraries, and new analysis interfaces. In addition, new terminal and processor hardware and operating software may require changes to the code depending upon the specific CADS operating environment. A central maintenance configuration control point should be established to modify, track, and distribute changes to the CADS software in order to most effectively meet the long range CADS support requirements.

REFERENCES

1. DI-3000 User's Guide, Precision Visuals, Incorporated, Boulder, Colorado, April 1981.

2. DI-3000 Installation Notes, Precision Visuals, Incorporated, Boulder, Colorado, October 1981.

3. Hedgley, David R., Jr., A General Solution to the Hidden-Line Problem, NASA reference 1085, COSMIC No. ARC-11446, Ames Research Center, Dryden Flight Research Facility, Edwards, California, 1982.

4. American National Standard Programming Language FORTRAN, ANSI X3.9-1978, American National Standards Institute, Inc., New York, NY, 1978.

5. VAX-11 FORTRAN User's Guide, Digital Equipment Corporation, Maynard, Massachusetts, April 1980.

6. VS FORTRAN Application Programming: Language Reference, International Business Machines Corporation, Endreott, NY, September 1982.

7. Venkayya, V. B., and Tischler, V. A., ANALYZE - Analysis of Aerospace Structures with Membrane Elements, AFFDL-TR-78-170, Air Force Flight Dynamics Laboratory, Wright-Patterson Air Force Base, Ohio, December 1978.

8. Venkayya, V. B., and Tischler, V. A., OPTSTAT - A Computer Program for the Optimal Design of Structures Subjected to the Static Loads, TM-F8R-79-67, Air Force Flight Dynamics Laboratory, Wright-Patterson Air Force Base, Ohio, June 1979.

U.S.Government Printing Office: 1986 — 646 067/40653

END

FILMED

3-86

DTIC