

## NASTRAN USED IN A PRODUCTION ENVIRONMENT

R. F. Wilkinson, J. W. Kelley, and N. K. Snead

Lockheed-Georgia Company

### SUMMARY

This paper describes a finite element analysis procedure built around the NASTRAN system. It describes a number of support programs that have either been written or modified to interface with NASTRAN and some improvements that have been made to NASTRAN itself. It shows some typical models that are being analyzed and an actual schedule that is being followed for constructing and analyzing the models to support a large design program.

### INTRODUCTION

NASTRAN is the main finite element analysis tool used in the C-5A project stress office at the Lockheed-Georgia Company (Gelac). It was first used in June '72 on an analysis which required an idealization using plate bending elements (CQUAD2). The in house program FAMAS was not used because it did not have a similar type of element. Upon using the system it became evident that NASTRAN had most of the capabilities that existed in FAMAS plus many more. One big draw back, however, to adopting NASTRAN in place of FAMAS as Gelac's main finite element analysis tool was that a number of support programs existed that specifically interfaced with FAMAS. Since then these have all been modified to interface with NASTRAN which has led to a very efficient static finite element analysis capability. This capability is currently playing an important part in a program to redesign the C-5A wing.

### FINITE ELEMENT ANALYSIS PROCEDURE AT THE LOCKHEED-GEORGIA COMPANY

An engineer at the Lockheed-Georgia Company has available to him three modes in which to execute his computer runs on two UNIVAC 1106 machines. Twenty-three UNIVAC DCT (Data Communications Terminals), which are

asynchronous key board/print devices, allows the engineer to execute jobs and edit data files in an interactive, DEMAND, mode. Jobs executed in this manner are limited to 32K and a CPU time of 10 minutes. A UNIVAC 1557/1558 display system, SCOPE, is used in a similar mode with the additional capability of displaying data on the 1558 console. Finally jobs can be executed in a batch mode which has a turn around time of between 2 and 24 hours.

The standard procedure by which an engineer concerned with making a finite element analysis uses this system is shown in Figure 1. This has been partitioned into the three main tasks involved in a finite element analysis namely "Model Construction", "Model Check Out", and "Analysis". Models are constructed thru a computer program called NABDAG (NASTRAN Bulk Data Generator). The input to this program can be as much as an order of magnitude less than the bulk data that it generates. The input can be either typed in at the DEMAND terminals or key punched and read into a mass storage file. The program is executed in the DEMAND mode, the output being saved in a mass storage BULK DATA FILE. When all the bulk data has been generated NASTRAN is executed to form the element connectivity tables and grid point coordinate tables. These tables are copied into mass storage files which are then read into an interactive graphics program executed from the SCOPE console. The interactive graphics program displays the model on the console enabling the engineer to identify any errors. These can then be corrected by editing the bulk data file in the DEMAND mode. The cycle of plotting the model and editing the file is repeated until all errors have been eliminated. Once a checked out BULK DATA FILE is available NASTRAN is again run to form the element connectivity tables. These are read into a banding program, BANDIT, which forms the necessary SEQGP card images and edits them into the BULK DATA FILE. The actual analysis, which is executed in the batch mode, is often interfaced with post processing programs. A typical such program, Loose Fit, simulates effects like the pull away of a plate from a bolt in a hole. These programs are similarly interfaced with NASTRAN by writing the necessary tables and matrices into mass storage files.

#### BULK DATA GENERATOR (NABDAG)

The NASTRAN bulk data format, by its nature, requires large quantities of input data cards. As the utilization of NASTRAN increased at Gelac the need for a condensed input format became evident. Since the inhouse finite element program, FAMAS, utilizes a very efficient input system, its sub-routines have been utilized in a NASTRAN bulk data generator program, NABDAG.

The FAMAS read subroutine, RETAK, reads unformatted data records which are delineated by a minus (-) in Column 1 of the last card of the record. The words in a record, which are read as real numbers starting in Column 4, are delineated by commas. Each record read by RETAK is stored in a single dimensional array and returned to the calling program for appropriate manipulation. The unique feature in RETAK which lends itself to generating bulk data for a finite element model is the ability to repeat or bump data words. This is accomplished thru the use of the following two special data words:

- nR Repeat.
- nB, J, b<sub>1</sub>,----,b<sub>J</sub> Bump.

The 'nR' indicates that the previous word should be repeated n times. The nB indicates that the previous J words should be repeated, as a block, n times, each time adding the increments b<sub>1</sub> thru b<sub>J</sub> to the respective words. As an example consider the RETAK record:

10,4R,2B,5,10,4R

This would be returned to the calling program as the array

10 10 10 10 10 20 20 20 20 20 30 30 30 30 30

The main features of the NABDAG program are:

- The card image output format is defined as input data, hence the program can in fact be used for any finite element system.
- A single RETAK record is used to input, for all bulk data cards of the same type, the data for each output field. For example in inputting the GRID data a single RETAK record would be used to specify say all the X2 coordinates.
- If two data fields are identical then only a single RETAK record is required. For example, if all input and output coordinate systems on a GRID card are identical then only one need be input.

The input to the program consists of three parts. The first part defines the type of each RETAK record i.e. whether it is real or integer,

and it specifies in which output field each of the records go. The second part defines the output format and the third part consists of the actual RETAK records. Specific input definitions are given in Table 1.

As an example consider a simple swept wing surface. The NABDAG input data to idealize this surface by quadrilateral membrane elements, CQDMEM, is shown in Figure 2. These 9 card images became 54 BULK DATA card images after being processed by NABDAG. Although this example is for an ideally regular model, a savings of 10 to 1 can be achieved on complicated models.

### BANDING PROGRAM


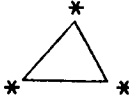
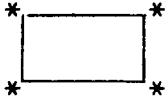
The Lockheed-Georgia Company has a program called BANDIT which will automatically resequence the grid point numbers for a model in such a way that the semi-band width of the stiffness matrix tends towards a minimum. The program requires the grid points to be numbered sequentially and works with a grid point connectivity table which lists, for each pivotal grid point, the grid points that are connected to it by elements. The details of the algorithm will not be presented here as the object of this section is only to demonstrate how simple it was to interface NASTRAN with BANDIT.

The external grid point numbering for a NASTRAN model, as constructed at the Lockheed-Georgia Company, are in general non-sequential. NASTRAN however works internally with grid point numbers that are sequential and in fact generates connectivity tables in terms of these internal grid numbers. The two NASTRAN tables that can be used to create the desired connectivity table are GPCT and EQEXIN, reference 1. The GPCT table gives the grid point connectivity in terms of the SIL number (Scalar Index List). The EQEXIN table gives the external grid number in terms of the internal grid numbers and the external grid number in terms of the SIL numbers. It was therefore a simple task to modify the BANDIT program to accept these two tables and generate the desired connectivity table in terms of internal grid numbers. The program resequences these numbers and then used the EQEXIN table to determine the resequencing for the external grid numbers. These are output to a mass storage file as SEQGP bulk data card images for direct input to NASTRAN. The DMAP instructions for generating the GPCT and EQEXIN tables and writing them out into a mass storage file is shown in Table 2.

## INTERACTIVE GRAPHICS

An interactive graphics program has been developed at Gelac which is used to check finite element models prior to making an analysis, reference 2. This program is executed from a UNIVAC 1558 graphics display console which has a 12-inch square viewing area, see Figure 3. The program was initially developed to plot models coded for the in house FAMAS system. It was found that the simplest way to plot a model coded for the NASTRAN system was to modify the program to accept the EST, BGPDT, MPT, SIL, and EQEXIN tables generated in a NASTRAN run, see reference 1. The EST table gives the element connectivity data in terms of SIL numbers. The BGPDT table gives the grid point coordinate data. The MPT table gives the element material data. The SIL and EQEXIN tables give the external grid point numbers in terms of the SIL numbers and the internal grid point numbers. The DMAP instructions for generating these tables and writing them on a file for input to the interactive graphics program are shown in table 3.

The interactive graphics program plots the following three basic shapes:

- A two node line element 
- A three node triangular element 
- A four node quadrilateral element 

It was therefore necessary to associate each of the NASTRAN elements to one of these shapes. The interactive graphics program also uses the concept of element groups where a group consists of a set of elements all of the same type. This feature is convenient for selecting various model views especially when idealizations overlap each other.

An engineer working in real time at the scope console first selects the groups he desires to plot. He may then further partition his model by specifying a plane, box, sphere, cylinder, or a specific quadrant. Then only those elements within the defined region will be displayed. Once the desired partition is displayed he is able to rotate it about any of the 3 axis, translate it, or zoom in on a specific detail. The advantage of an interactive graphics system over plots generated in a batch mode is

illustrated in figure 4. View 'a' shows how the model first appears on the console. Upon rotating the view 30° about the vertical axis then 30° about the horizontal axis an error became apparent, view 'b'. Zooming in on the region of the error allows the engineer to identify exactly what the problem is, view 'c'. Hence, where a single sitting at the console identifies the error, it would take at least two batch runs to do the same.

#### LOOSE FIT PROGRAM

This program is used to simulate a loose fit in the joint of a structure. For example two coincidental grid points may be able to displace independently within a fixed tolerance before they become locked and transmit load to one another. Another example is a beam with sunken supports. The grid points would be able to displace freely within fixed limits before they become grounded and react loads.

The algorithm used to simulate this effect is to solve the equation.

$$\{R\} = \{P\} - [K] \times \{U\} \quad (1)$$

where

[K] is the relative stiffness matrix for the joints that have a loose fit.

{P} is the loads matrix for the joints when they are rigidly connected.

{u} is the relative displacement of the joints.

The equation is solved for {u} subject to the following constraints.

$$u_{\min} \leq u \leq u_{\max}$$

$$\text{If } u_{\min} < u < u_{\max}, \quad R = 0 \quad (2)$$

$$\text{If } u = u_{\min}, \quad R \leq 0$$

$$\text{If } u = u_{\max}, \quad R \geq 0$$

Where  $u_{\min}$  and  $u_{\max}$  are the tolerances for the displacements {u}. The solution is an iterative process which is accomplished by the following steps.

1. Initially  $\{R\}$  is assumed to be zero and equation (1) is solved for  $\{u\}$ .
2.  $\{U\}$  is inspected and any displacements outside the tolerances are fixed to be equal to the respective tolerance. These are then partitioned out of equation (1) and the remaining equations again solved for  $\{R\} = 0$ .
3. The residuals  $\{R\}$  are now solved, thru equation 1, for the freedoms that were eliminated in step 2. These are inspected and any that violate the last two constraints in equation (2) are freed from having these displacements fixed at a tolerance. The displacements  $\{u\}$  are also reexamined and any additional ones that are outside the tolerance are likewise fixed to be equal to the respective tolerance.
4. The new partition of equation (1) is again solved for  $\{u\}$ . Stages 3 and 4 are repeated until all constraints in (2) are satisfied.

To illustrate how this program is interfaced with NASTRAN consider the problem of a beam on sunken supports shown in Figure 5. First the problem is analyzed using rigid format 1. The vertical displacements for the grid points are included in the "A" set and the rigid format is altered to output the reduced "A" set stiffness and load matrices, (KLL) and (PL), onto a file immediately after they have been formed and terminate. These matrices are read into the Loose Fit Program together with the tolerance

$$0 \leq u_a \leq 0.00009$$

Loose fit solves for the "A" set displacements, ULV, and generates the DMI cards for direct input back into NASTRAN. Rigid format 1 is then restarted using these "A" set displacements and completes the analysis. Table 4 shows the executive control decks for both NASTRAN analysis and the DMI cards specifying the ULV displacements.

## GELAC IMPROVEMENTS TO NASTRAN

Improvements have been made to the NASTRAN system at Gelac by way of including additional elements. The two elements that have so far been added are a Nine Node Crack Element and a Fastener Element. The nine node crack element is used to calculate the stress intensity at the tip of a crack for crack growth and stability studies. The formulation for this element and how it was incorporated into NASTRAN as a dummy element was presented at the last colloquium, reference 3. The fastener element is simply two orthogonal CELASI elements which have the same stiffness. This is a desirable element for representing a fastener attaching two plates. The advantage of using this element over two CELASI elements is of course that it halves the amount of input data but more important it is able to directly output the resultant load on the fastener. Currently this element is limited to an elastic material property. Plans are underway to include plasticity so that it can truly represent the nonlinear load deflection curve for a fastener.

## C-5A WING REDESIGN SUPPORT

The C-5A wing redesign program requires the analysis of nineteen finite element models. The models represent local areas in the wing where cracks developed during the cyclic testing of a full scale fatigue article. The purpose of the analyses is to find an optimum design that minimizes the concentrations that causes the cracks. The models are also used to generate stress intensity factors, using the crack element, for crack growth studies.

The analysis for these models are spread over an eight month period as shown in Figure 6. The models vary in complexity, and require between four and eight weeks to construct and analyze. The construction phase takes between one and three weeks, the check out phase between one and two weeks, and the analysis phase between two and three weeks. The analysis phase generally includes a number of iterations involving geometry and property changes. Three models, which are typical of the nineteen shown in Figure 6 will now be described in detail.

The model shown in Figure 7 represents a typical stringer run out at a chordwise splice. The initial configuration was designed such that the skin tab lined up with the centroid of the skin stringer combination. Strain gage measurements indicate that with this configuration there are



large bending stresses in the tab, also cracks developed at the holes in the tab on the test article. The purpose of the analysis is therefore to investigate the effect of varying the joggle of the tab in order to find a configuration which minimizes the bending stresses. The model idealizes one half of a skin/stringer combination and has boundary constraints which represent an infinite panel. The splice plates and the skin are idealized by plate bending and membrane elements. The riser web is idealized by membrane elements and the riser flange by axial elements. The fasteners are idealized by bar elements with constraint equations written to equate the rotations at the ends of the bar. The moment of inertia for the bars are then calculated from the equation

$$\frac{P l^3}{12EI} = \frac{P}{K}$$

where K is the fastener stiffness obtained from test.

The model shown in Figure 8 represents a tab out in the skin panel at the front spar which picks up the leading edge structure. Cracks originated in the fatigue article at the hole adjacent to the tab out radius. The model was set up so that the location of the hole relative to the radius could be varied in order to find the configuration that gave the minimum concentration effect. The results of the analysis were intended to be used directly in a fatigue analysis, so the actual hole was modeled in both the skin and the leading edge structure. The effect of the bolt on the distribution was included by modeling the bolt and simulating the pull away effect by the Loose Fit Program. The way this was done is illustrated in Figure 9. Scalar points were defined for each grid point on the skin and leading edge structure around the hole. Multi point constraint equations were then written such that these scalar points defined the relative radial displacement between the skin and the bolt and the leading edge structure and the bolt. These freedoms were then defined in the A set and the procedure previously described used to interface with the Loose Fit Program. The constraints on the relative displacement used in the Loose Fit program were:

$$0 \leq U_a < \infty$$

The resulting relative displacements are shown in Figure 9.

The region modeled in the third example, shown in Figure 10, represents a main joint in the rear spar of the wing. The model is split into two substructures representing the center and inner wing sides of the joint. Details are shown for the idealization of the inner wing substructure only. The various components, web, fitting, and cap are idealized individually and connected together by fastener elements. The purpose of the model is to size the fitting and web in such a way that the concentration effects caused by the web dumping load into the cap at the joint is minimized. The two substructures were initially analyzed and iterated with rigid restraints imposed at their interface to obtain an approximate sizing. They were then coupled together and again iterated to obtain the final sizing. The coupling analysis was divided into three phases. Phase 1 involved an individual analysis for each substructure to obtain the boundary stiffness and loads matrices. The coupling freedoms were included in the "A" set and the "A" set stiffness and loads matrices written out onto a user tape. The alters to rigid format 1 to do this are identical to those shown in table 4 for the Loose Fit program. Phase 2 involves the actual coupling analysis which solves the equilibrium equations for the boundary displacements. The DMAP instructions for the coupling analysis are shown in table 5. Finally Phase 3 solves for the element stresses. Here again the alters to rigid format 1 are the same as those shown in table 4 for the Loose Fit program.

#### CONCLUSION

The intent of this paper has been to demonstrate how effectively NASTRAN can be used in a production environment. The key to accomplishing this was in being able to interface NASTRAN with existing finite element support programs. We at Lockheed-Georgia Company credit our success to the excellent documentation provided with NASTRAN.

#### REFERENCES

1. The NASTRAN Programmers Manual. NAS SP-223(01), Sept. 1972.
2. Batdorf, W. J., Holliday, J.F., Peed, J. L.: A Graphics Program for Aircraft Design - GPAD System. AIAA Thirteenth Aerospace Sciences Meeting. 75-136.
3. Wilkinson, R. F., Kelley, J.W.: A Failsafe Analysis Using NASTRAN's Piecewise Linear Analysis and a Nine Node Linear Crack Element. NASTRAN User's Experience. NAS TM X-3278, 1975.

TABLE 1 Input Instructions for NABDAG

RECORD	FORMAT	VARIABLE	DESCRIPTION
1	10I5	M	OUTPUT UNIT #
		NFOR	# OF A6 WORDS NEEDED FOR THE OUTPUT FORMAT
		NWPC	# OF DATA FIELDS ON A BULK DATA CARD
		NRECS	# OF INPUT RECORDS
		INT(1)	INPUT RECORD TYPE SPECIFICATION 0 INDICATES INTEGER DATA 1 INDICATES REAL DATA
		INT(NRECS)	SPECIFY A NUMBER FOR EACH OUTPUT FIELD INDICATING WHICH INPUT RECORD IS TO BE USED
		MSEQ(1)	
MSEQ(NWPC)			
2	12A6	IFOR(1)	OUTPUT FORMAT
		IFOR(NFOR)	
3	RETAK	DATA	INPUT DATA
		DATA	
2 + NRECS	RETAK		

TABLE 2 DMAP Listing for Interfacing NASTRAN With BANDIT

```

ID BANDIT, TABLES
APP DMAP
TIME 10
DIAG 1,14
BEGIN$
GP1 GEOM1,GEOM2, /GPL, EQEXIN, GPDT, CSTM, BGPDT, SIL/V,N, LUSET/C,N,
    123/V,N, NOGPDT $
SAVE LUSET $
GP2 GEOM2, EQEXIN/ECT $
GP3 GEOM3, EQEXIN, GEOM2/SLT, GPTT/C,N, 123/V,N, NOGRAV/C,N, 123 $
TA1, ,ECT, EPT, BGPDT, SIL, GPTT, CSTM/EST, ,GE1, ECPT, GPCT/V,N, LUSET/C,N,
    123/V,N, NOSIMP/C,N, 0/V,N, NOGENL/V,N, GENEL $
OUTPUT2 GPCT, EQEXIN, , , /C,N, 0/C,N, 12 $
END$
CEND
TITLE = DATA TABLES NECESSARY FOR BANDING THE STIFFNESS MATRIX
BEGIN BULK
$ * * * BULK DATA DECK * * *
ENDDATA

```

TABLE 3 DMAP Listing for Interfacing NASTRAN and 3-D Graphics

```

ID GRAPHICS, PLOTS
APP DMAP
TIME 10
DIAG 1,14
BEGIN$
GP1 GEOM1,GEOM2, /GPL, EQEXIN, GPDT, CSTM, BGPDT, SIL/V,N, LUSET/C,N,
    123/V,N, NOGPDT $
SAVE LUSET $
GP2 GEOM2, EQEXIN/ECT $
GP3 GEOM3, EQEXIN, GEOM2/SLT, GPTT/C,N, 123/V,N, NOGRAV/C,N, 123 $
TA1, ,ECT, EPT, BGPDT, SIL, GPTT, CSTM/EST, ,GE1, ECPT, GPCT/V,N, LUSET/C,N,
    123/V,N, NOSIMP/C,N, 0/V,N, NOGENL/V,N, GENEL $
OUTPUT2 SIL, BGPDT, EQEXIN, EST, MPT//C,N, 0/C,N, 12 $
END$
CEND
TITLE = DATA TABLES NECESSARY FOR PLOTTING ON THE SCOPE
BEGIN BULK
$ * * * BULK DATA DECK * * *
ENDDATA

```

TABLE 4 DMAP Listings for a Loose Fit Analysis

```

ID LOOSE,FIT
CHKPNT YES
APP DISPLACEMENT
SOL 1,1
ALTER 103,126
OUTPUT2 KLL,PL,///C,N,-1/C,N,12/C,N,LABEL $
ALTER 136,137
ENDALTER
TIME 20
DIAG 1,14
CEND
TITLE = LOOSE FIT ANALYSIS EXAMPLE
OUTPUT
DISPLACEMENT = ALL
ELSTRESS = ALL
SPCFORCE = ALL
MPC = 1
SPC = 1
SUBCASE 1
LOAD = 1
BEGIN BULK
$ * * * BULK DATA DECK * * *
ENDDATA
    
```

Initial Analysis to Form the A Set Stiffness and Loads Matrices

```

ID LOOSE,FIT
RESTART LOOSE ,FIT , 6/28/76, 14106,
1, XVPS , FLAGS = 0, REEL = 1, FILE = 6
2, REENTER AT DMAP SEQUENCE NUMBER 7
3, GPL , FLAGS = 0, REEL = 1, FILE = 7
.
.
.
$ END OF CHECKPOINT DICTIONARY
APP DISPLACEMENT
SOL 1,1
ALTER 103,105
FBS L00,U00,P0/U00V $
MPYAD KLL,ULV,PL/RULV/C,N,0/C,N,-1 $
MPYAD K00,U00V,P0/RU0V/C,N,0/C,N,-1 $
CHKPNT U00V,RULV,RU0V $
ENDALTER
TIME 20
DIAG 1,14
CEND
TITLE = LOOSE FIT ANALYSIS EXAMPLE
OUTPUT
DISPLACEMENT = ALL
ELSTRESS = ALL
SPCFORCE = ALL
MPC = 1
SPC = 1
SUBCASE 1
LOAD = 1
BEGIN BULK
DMI ULV 0 2 2 2 9 1
DMI* ULV 1 1 1 1 1-4401316000D-04* 1 1
* 1 1-.7508772000D-04-.8861842000D-04-.9000000000D-04-.9000000000D-04* 5 1
* 5 1-.9000000000D-04-.8861842000D-04-.7508772000D-04-.4401316000D-04
PARAM IRES 1
ENDDATA
LINES TO EOF 37
    
```

Final Restart Analysis to Form the Displacements and Stresses

TABLE 5 DMAP Instructions for Coupling

```

ID JOB100N,KELLEY $
TIME 10
DIAG 1,14
APP DMAP
BEGIN $
INPUTT1 /,,,,/C,N,-1/C,N,0/C,N,M8300S $
INPUTT1 /B1,E1T,KLL1P,PL1PPP,PG1PP/ $
INPUTT1 /B2,E2T,KLL2P,PL2PPP,PG2PP/ $
MPYAD E1T,KLL1P,/X1/C,N,1 $
MPYAD E2T,KLL2P,/X2/C,N,1 $
MPYAD X1,E1T,/KLL1/C,N,0 $
MPYAD X2,E2T,/KLL2/C,N,0 $
MPYAD E1T,PL1PPP,/X4/C,N,1 $
MPYAD E2T,PL2PPP,/X5/C,N,1 $
MPYAD X4,B1,/PL1/C,N,0 $
MPYAD X5,B2,/PL2/C,N,0 $
ADD5 KLL1,KLL2,,,/KLL/ $
ADD5 PL1,PL2,,,/PL/ $
MPYAD PG1PP,B1,/PG1/C,N,0 $
MPYAD PG2PP,B2,/PG2/C,N,0 $
DECOMP KLL/LLL,ULL $
FBS LLI,ULL,PL/ULV $
MPYAD E1T,ULV,/ULV1P/C,N,0 $
MPYAD E2T,ULV,/ULV2P/C,N,0 $
MPYAD KLL1,ULV,PL1/RULV1/C,N,0/C,N,-1 $
MPYAD KLL2,ULV,PL2/RULV2/C,N,0/C,N,-1 $
MPYAD E1T,RULV1,/RULV1P/C,N,0 $
MPYAD E2T,RULV2,/RULV2P/C,N,0 $
ADD5 RULV1,RULV2,,,/RULV/ $
OUTPUT1 PG1,ULV1P,RULV1P,/// $
OUTPUT1 PG2,ULV2P,RULV2P,/// $
MATPRN PG1,PG2,,,ULV,ULV1P// $
MATPRN ULV2P,,,RULV1P,RULV2P,// $
MATPRN RULV,,,,// $
END $
CEND
TITLE = PHASE 2 COUPLING ANALYSIS
BEGIN BULK

ENDDATA

```

# MODEL CONSTRUCTION

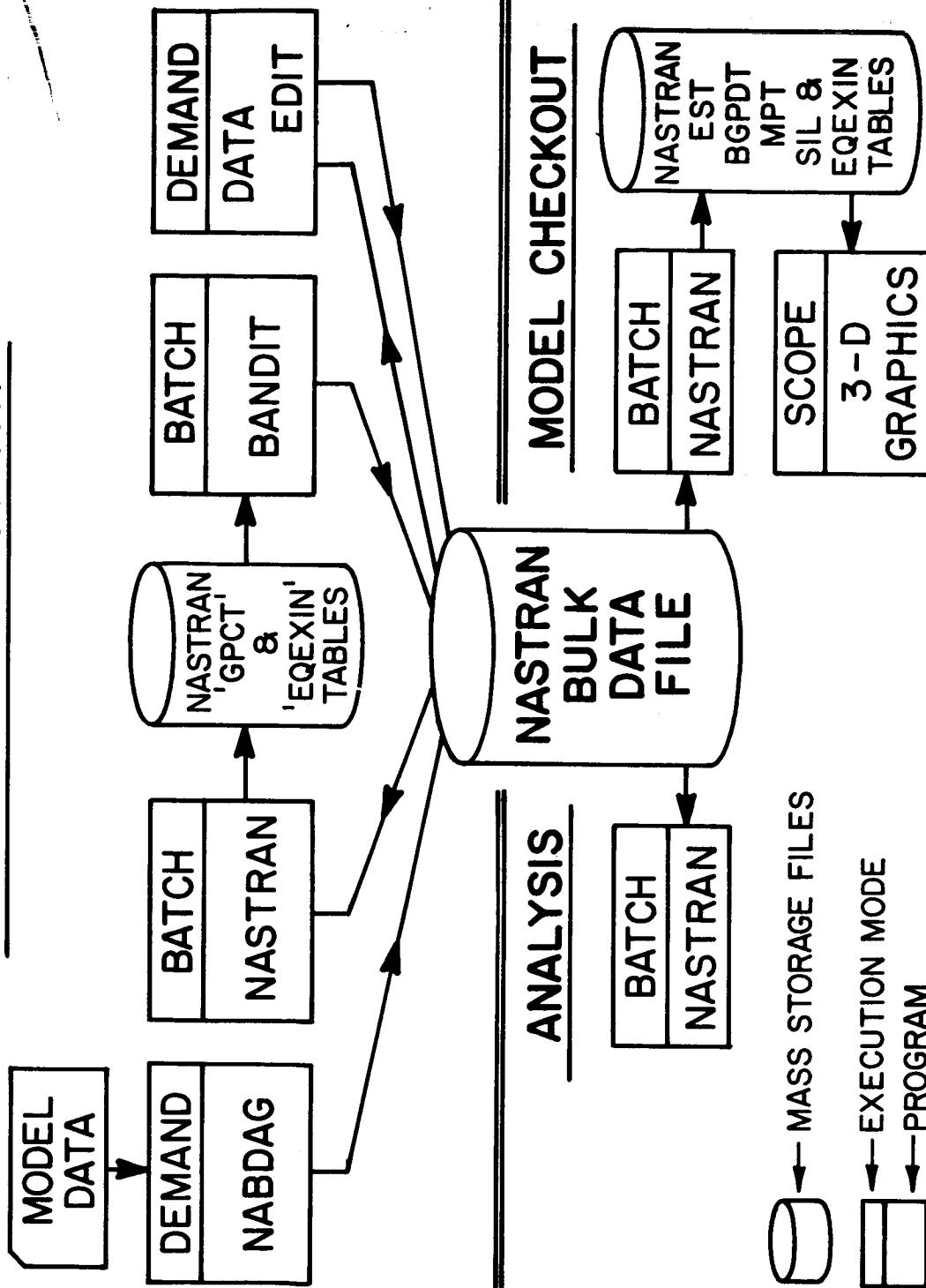
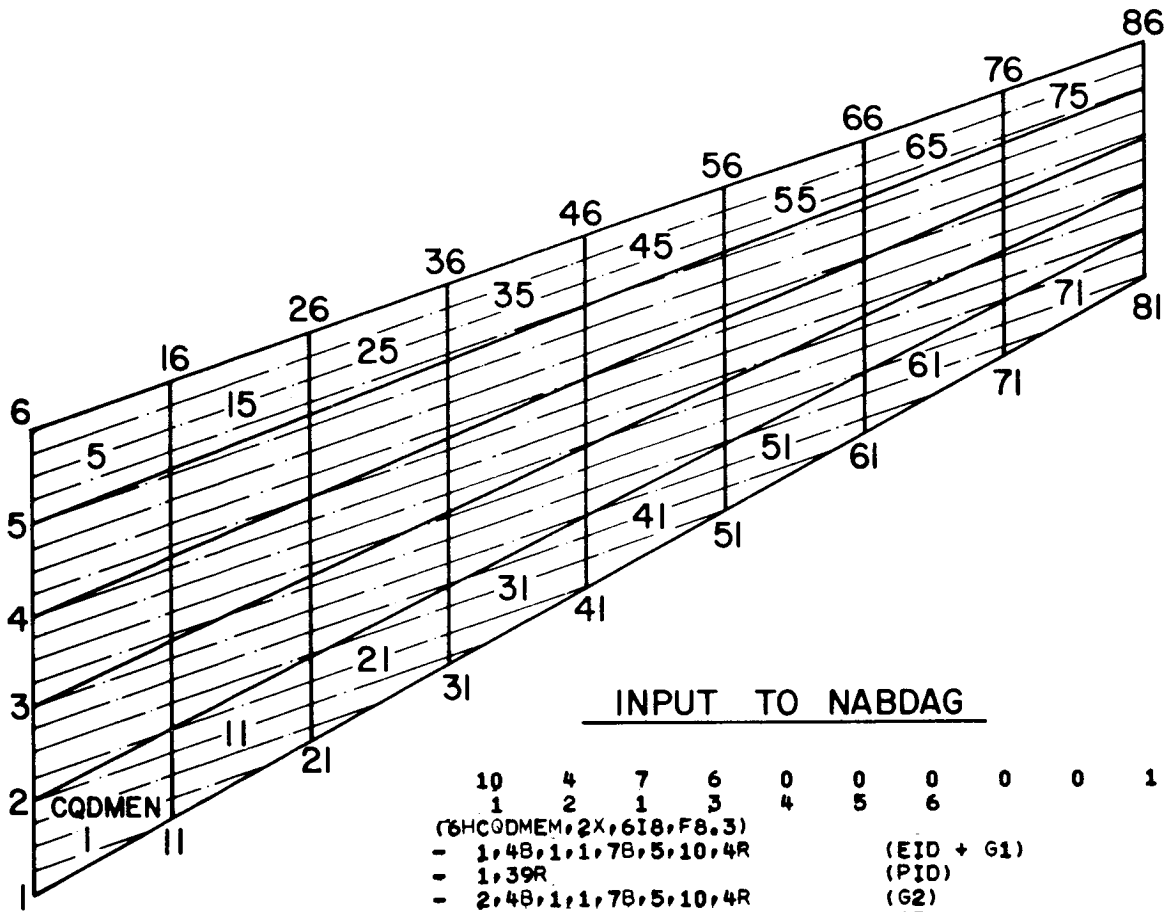


FIGURE 1 Gelac Standard Finite Element Analysis Procedure.



INPUT TO NABDAG

```

10      4      7      6      0      0      0      0      0      1
 1      2      1      3      4      5      6
(6HCQDMEM,2X,6I8,F8.3)
- 1,48,1,1,78,5,10,4R      (EID + G1)
- 1,39R                    (PID)
- 2,48,1,1,78,5,10,4R      (G2)
- 12,48,1,1,78,5,10,4R     (G3)
- 11,48,1,1,78,5,10,4R    (G4)
- 70.0,39R                 (TH)

```

OUTPUT FROM NABDAG

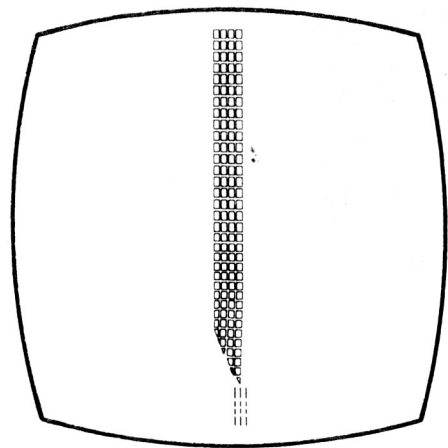
CQDMEM	1	1	1	2	12	11	70.000
CQDMEM	2	1	2	3	13	12	70.000
CQDMEM	3	1	3	4	14	13	70.000
	⋮	⋮	⋮	⋮	⋮	⋮	⋮
CQDMEM	73	1	73	74	84	83	70.000
CQDMEM	74	1	74	75	85	84	70.000
CQDMEM	75	1	75	76	86	85	70.000

FIGURE 2 NABDAG Data For Specifying the CQDMEM Elements of a Swept Wing Surface

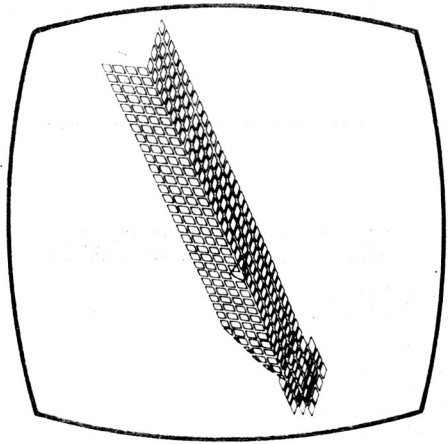




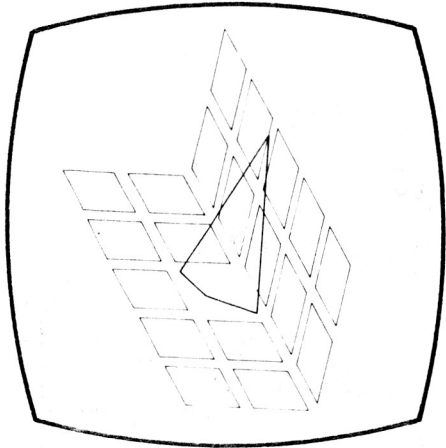
Figure 3 UNIVAC 1558 Graphics Display Console



View a



View b



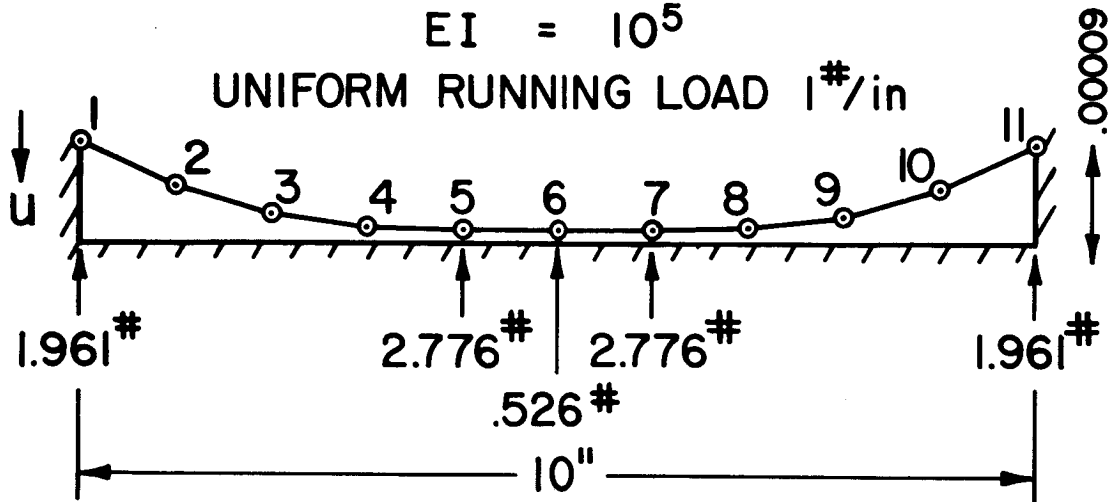
View c

FIGURE 4 The Advantages of an Interactive Graphics System

## CBAR ELEMENTS

$$EI = 10^5$$

UNIFORM RUNNING LOAD 1#/in



### DISPLACEMENTS FOR A SET FREEDOMS

$u_2$	=	.00004013
$u_3$	=	.00007509
$u_4$	=	.00008862
$u_5$	=	.00009000
$u_6$	=	.00009000
$u_7$	=	.00009000
$u_8$	=	.00008862
$u_9$	=	.00007509
$u_{10}$	=	.00004013

FIGURE 5 Loose Fit Analysis for a Beam on Sunken Supports.

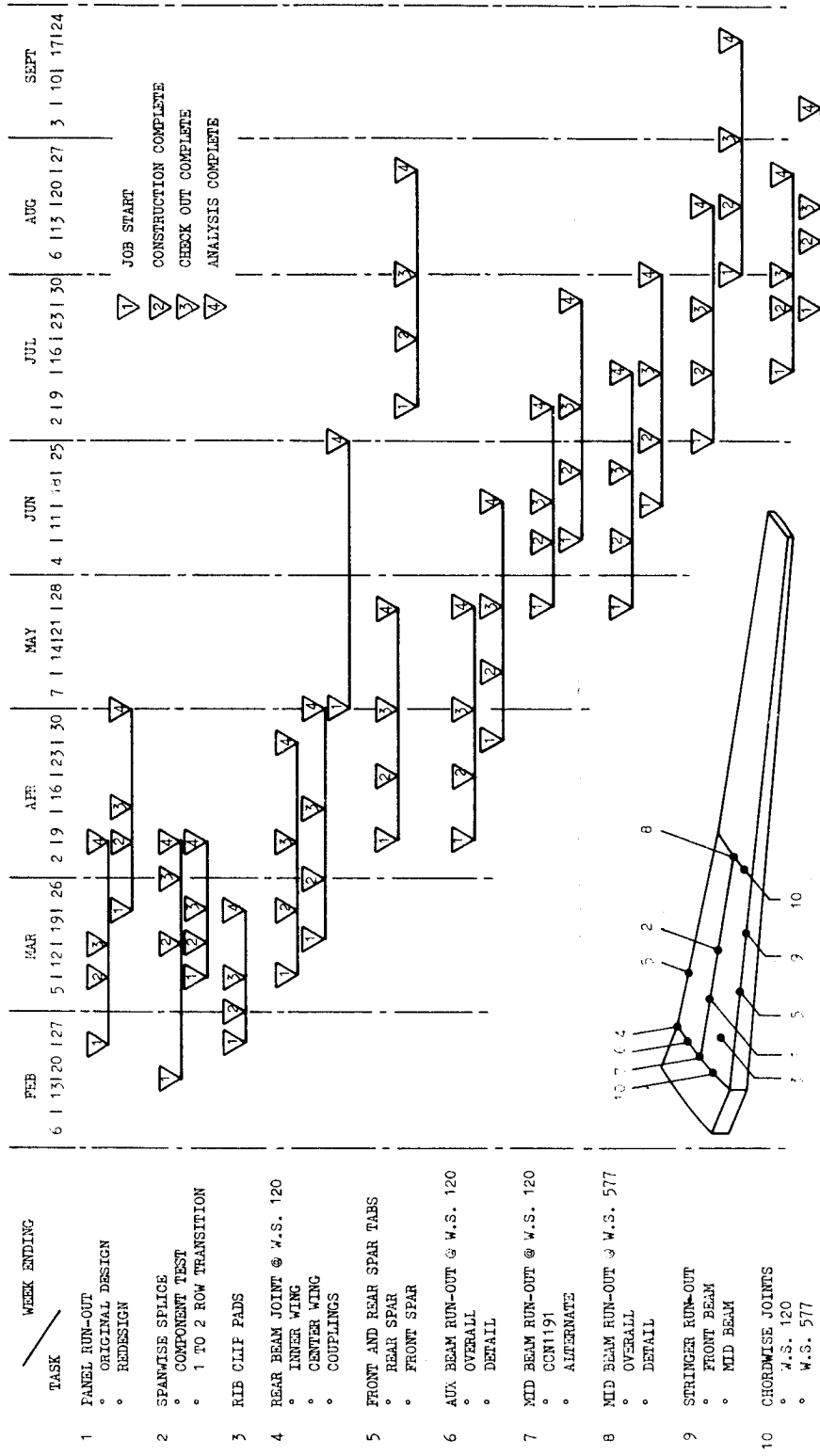


FIGURE 6 NASTRAN Finite Element Analysis Schedule For the C-5A Wing Redesign.

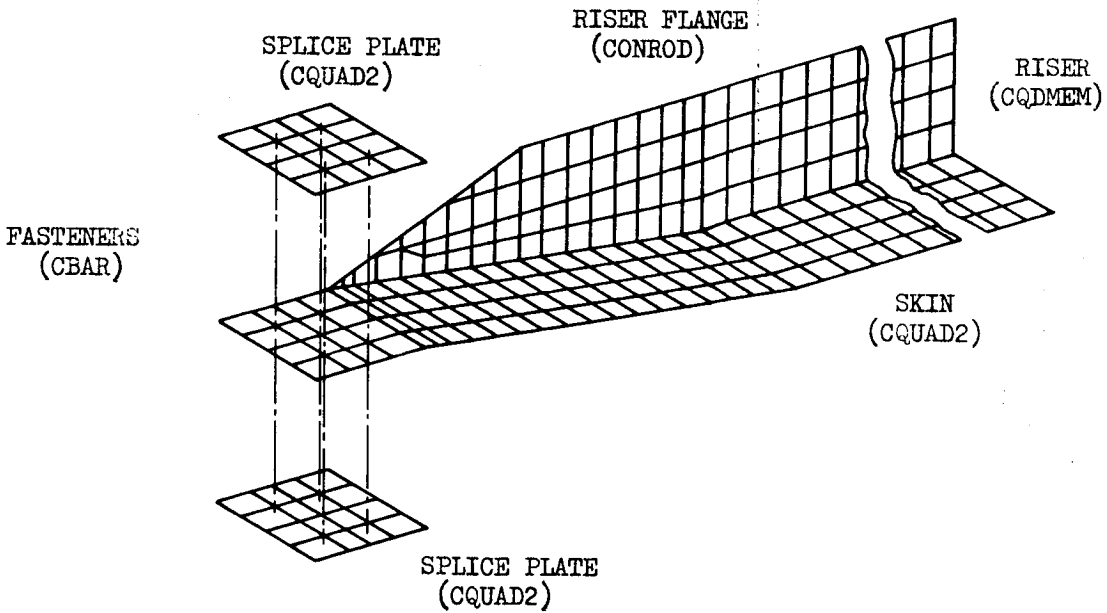
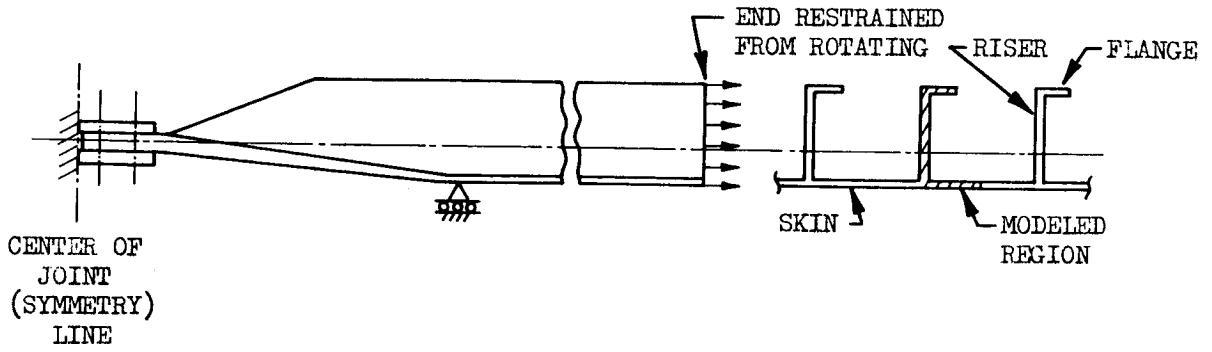


FIGURE 7 Chordwise Joint Model.

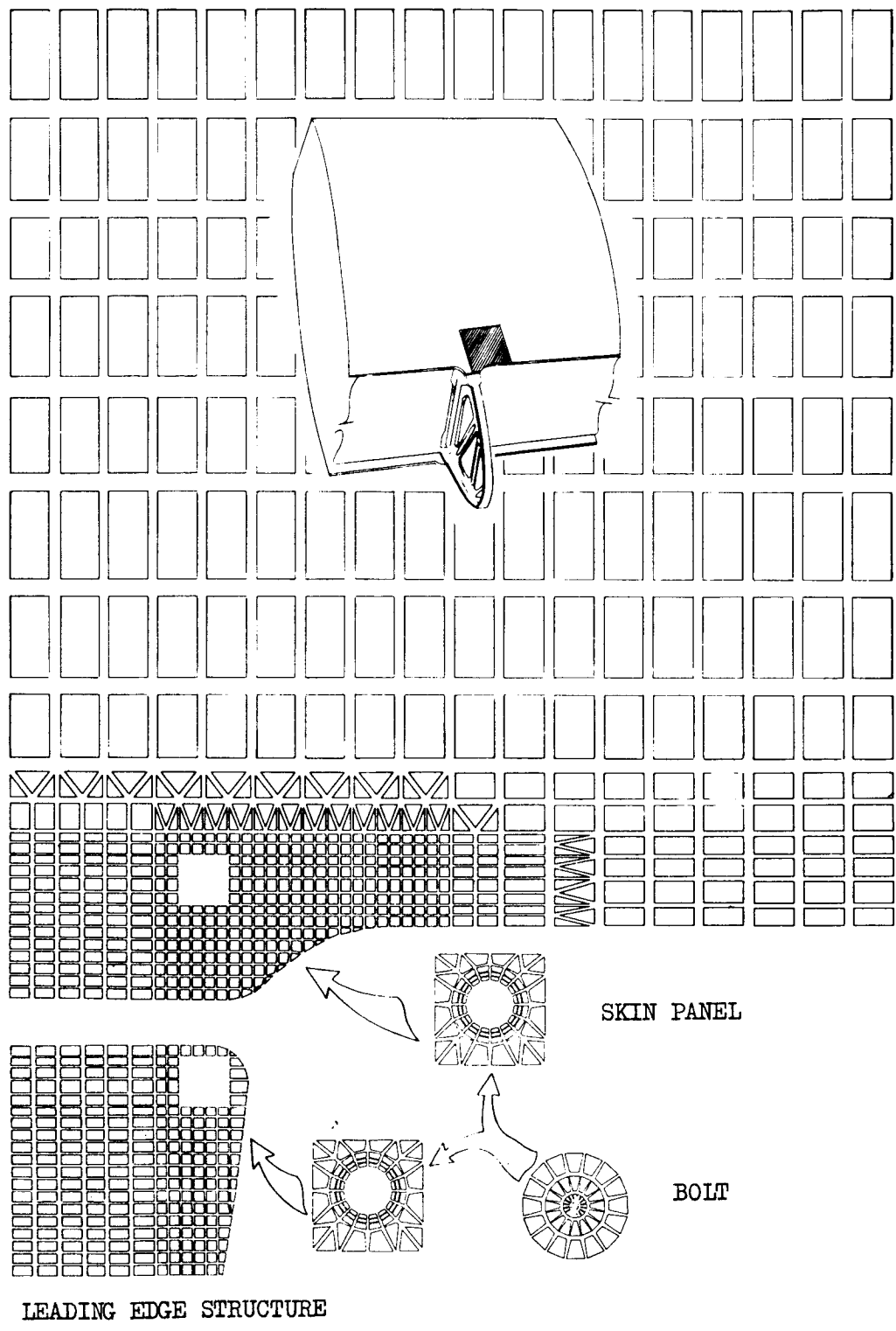
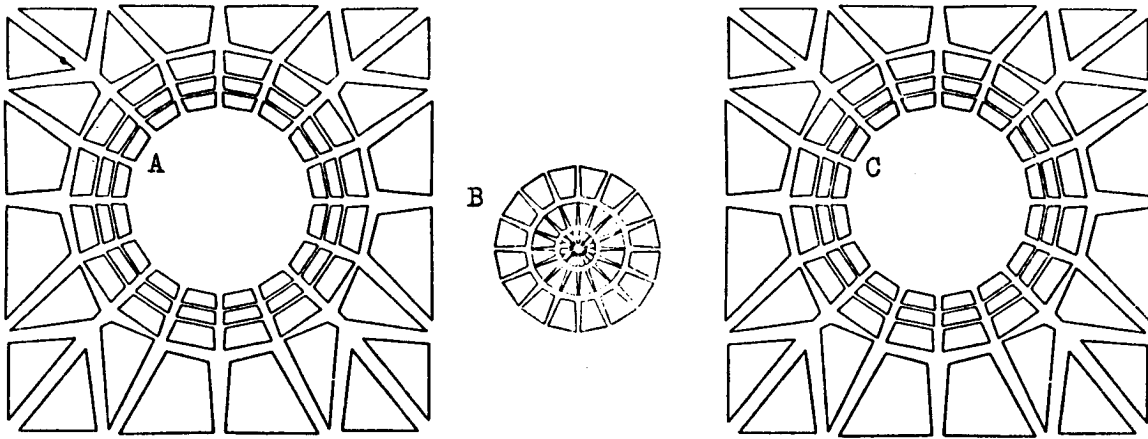


FIGURE 8 Finite Element Model of the Leading Edge Tab Out

CONSTRAINT EQUATIONS TO DEFINE THE RELATIVE DISPLACEMENT  $u(E)$  BETWEEN THE SKIN AND BOLT AND  $u(F)$  BETWEEN THE LEADING EDGE STRUCTURE AND THE BOLT.



$$\begin{aligned} uR(A) - uR(B) &= u(E) \\ uR(C) - uR(B) &= u(F) \end{aligned}$$

Where E & F are Scalar Points

RELATIVE DISPLACEMENTS  $u(E)$  AND  $u(F)$  FROM LOOSE FIT

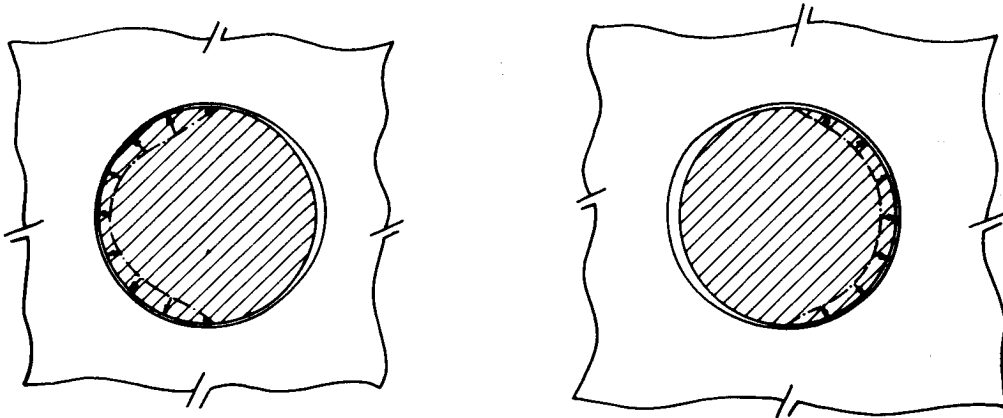
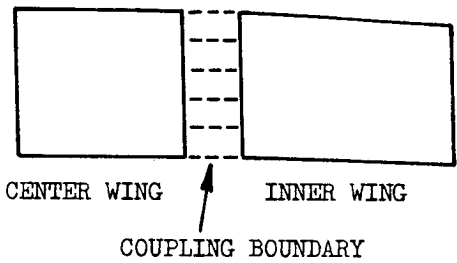
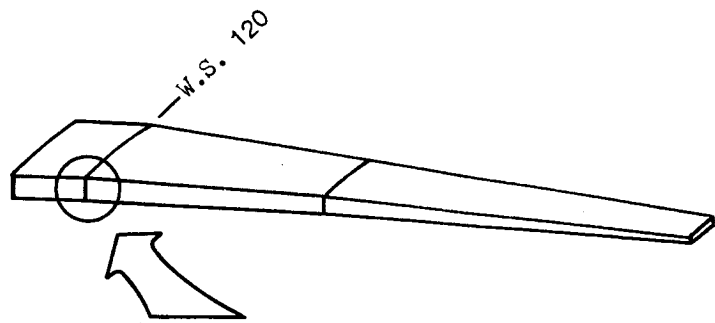


FIGURE 9 Loose Fit Used To Simulate Bolt Pull Away



INNER WING DETAIL

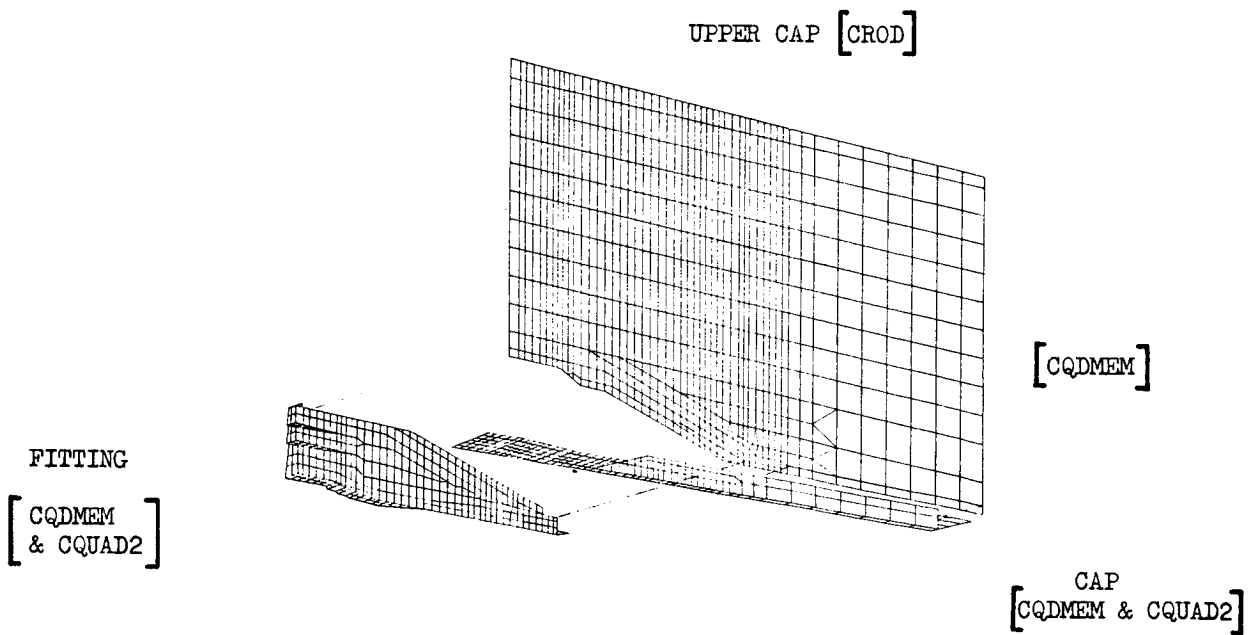


FIGURE 10. Rear Spar Joint On W.S. 120 Model