A COMPARISON OF MSC/NASTRAN with SDRC/I-DEAS MODEL SOLUTION

BY

TSAY-HSIN G. HU

and

MOSES S. HUANG

Lockheed Engineering and Sciences Company
2400 NASA RD 1, Houston Texas.

ABSTRACT

This study is an effort by NASA/JSC (Johnson Space Center) and LESC (Lockheed Engineering & Sciences Company) to compare the structural/stress analysis and problem solutions obtained using various solvers and post processing methods.

The investigations include solutions for real-life problems and test cases. Solution results compared include NASTRAN output, post processing of NASTRAN output using SDRC/I-DEAS Finite element package and Model Solution in SDRC/I-DEAS.

Significant differences in solution results are revealed. For consistent results care must be used in choosing the elements, the order of these elements as well as in the application of loads to these models. Also the post processing of the solution results may in certain cases be different. This is because different solvers report information in different ways which are not always directly comparable. This study is intended to find where the differences may occur and what steps may be taken to obtain comparable answers.

INTRODUCTION

With the introduction of fast and powerful computers, more and more engineers in the field of solid mechanics and stress analysis are able to analyze complicated structural components using finite element analysis on a routine basis. This means that the creation of stress analysis models must be performed efficiently.

The result of this is the increasing popularity of certain model-generating packages that are easy to learn and use and are basically graphical input software, using pointing devices for on-screen interaction and interfacing with drafting or solid modeling type models for other purposes.

Such a program is the SDRC/I-DEAS (Structural Dynamics Research Corporation) package which has become popular with aerospace engineers. However due to the requirements of contracts for NASA and in cases where very large models are generated, the solution of these problems are usually performed using NASTRAN.

Therefore SDRC/I-DEAS is used as a preprocessor to create the model and a post-processor to display the results in graphical form. Both of these functions are not particularly a strong point of the MSC/NASTRAN package especially when implemented for batch mode processing as is the case with our facility.

For models that can be performed using the SDRC/I-DEAS Model Solution this is preferred since many steps can be eliminated to obtain a solution. In our facility at Lockheed & NASA/JSC, SDRC/I-DEAS is installed on VAXes and the capacity to solve large problems is limited by the VAX disk capacity and user quotas. Because SDRC/I-DEAS model files are usually very large the ability to run large math models in our system is limited. Therefore very large models require the problem to be run on a mainframe computer.

Using the above approach to solve structural analysis problems however assumes that all solutions are reasonably equivalent whether solved in NASTRAN or SDRC/I-DEAS Model Solutions, or post processing NASTRAN information on SDRC/I-DEAS.

It is therefore important for end users to conduct independent studies of different solvers in use, especially in the case where the theoretical manual is unavailable, in order to verify the results of the post processing and/or solution.

A total of three example cases, two real-life models and one test case were used for comparison. For each example case three solutions were obtained for comparison. The three solutions were, Model Solution in SDRC/I-DEAS, MSC/NASTRAN output and post processing of MSC/NASTRAN results in SDRC/I-DEAS (see Figure 1). In this study, MSC/NASTRAN version 65 and SDRC/I-DEAS version 4.0 are the versions used.

Real-life Stress Analysis Cases.

Two real-life stress analysis jobs performed in the course of stress analysis work were used for comparison purposes between the various solution and post processing methods.

These two stress analysis models were created to obtain stresses of certain experimental hardware which will be flown on future Space Shuttle missions. They represent real world practical models and is as refined as time and effort allows, not neccessarily ideal models.

Case 1: Dexterous End Effecter, Flange.

The FEM (Finite Element Method) model consists entirely of solid elements with three DOF at each node. A total of 981 nodes and 560 solid elements were used. The flange has varying thicknesses at the inner and outer edges. Figure 2 shows a plot of the flange model.

The flange is constrained at the outer edge at 12 bolt points, arranged 30 degrees between each bolt point. Loads were applied at the inner edge groove. Nodal point loads were used to approximate a circumferential loading on the groove.

Case 2: GAS Beam FSE Adaptor Plate

The plate is part of the support hardware that attaches the beams of the CETA (Crew & Equipment Transport Aids) experiment to the orbiter side wall. This plate like object is part of the CETA beam support at the orbiter side wall attachment at Bay 2.

This adaptor FEM model consists of 288 plate and solid elements and 560 nodes. The plate elements are linear thin shell elements with five DOF at each node. The solid elements have 3 DOF at each node. Loads to the plate are applied to pin locations as shown in Figure 3. The loads are transferred to the blocks by stiff beams. The beams and blocks represent the latch assembly where the CETA beams are attached to the orbiter attachment structures. The blocks connect to the plate at specified bolt locations in 3 translational degrees of freedom.

In both cases the material properties were that for aluminum with Young's Modulus of 10 Msi and Poisson's ratio 0.33.

Comparison of the results are based on the following output parameters, the Maximum Principal Stress, the Minimum Principal Stress, the Maximum Shear Stress, the Von Mises combined stress and the Maximum Displacement. Plate elements include Shell Top and Bottom surface.

In addition to the magnitude of stresses obtained a comparison of the nodal stress point location where the maximums occur is also made.

Tables 1 through 2C summarize the results of the analysis for the above two cases.

TEST CASE

From the above results, it was noted that there exist differences in the three solution methods: I-DEAS/MODEL SOLUTION, MSC/NASTRAN and MSC/NASTRAN results post-processed in I-DEAS. Differences in the maximum stresses were significant. A test case was created to compare the various solution schemes versus the different element types used.

The test structure created is a cantilever tube with dimensions of 10 inches length, 2 inches nominal diameter and thickness of .08 inches. The thickness is chosen so that D/t > 10.0 for thin shell solutions. The material properties of Young's Modulus 30 Msi and Poisson's ratio 0.29 are assumed. For plate type elements the shear area ratio in the shell property input was assigned a value of 1.0 instead of the default 0.833 (CQUAD4, CQUAD8 CTRIA3 CTRIA6).

Figure 4 shows the tube structure and its boundary conditions as well as loading condition (tip load of 1000 lbs.). The points A, B, D, E and F are the points used for stress comparisons. Because the plotting routine displays the maximums, stresses under the load points are recorded also although we would note here that stresses at singularity points should not in general be compared.

Six types of element were investigated. The corresponding element name in MSC/NASTRAN and SDRC/I-DEAS are listed in Table 3A. The short forms are also listed in Table 3A for easy tabulation. For example, PQSM denotes the MSC/NASTRAN solution for model with CQUAD8 element.

The modelling information for the test case is shown in Table 3B. It includes the meshing, total element number and total node number for each type of element used.

The results are shown in Table 4 through Table 6. Table 4 and Table 5 present the comparisons of the Maximum Principal Stress, the Minimum Principal Stress, the Maximum Shear Stress, the Von Mises combined stress and the Displacement at the node where the maximums occurs and point B respectively.

Table 6 tabulates the stress in the longitudinal direction at auxilliary points A, B, C, D, E and F. Please refer to Figure 4 for these locations.

Table 7 summarizes the CPU and ELAPSE time when performing solutions with the different solvers.

The following observations were noted:

(1) I-DEAS/SUPERTAB does not post-process the stress results from MSC/NASTRAN in the same way MSC/NASTRAN processes the element stress information as performed using the GPTRESS command. The model using element type PQS (Parabolic Quadrilateral Shell element/CQUAD8) provided the best comparison of the stresses between the three solution methods. In general because the two solvers interpolate the element stresses differently the results are not exactly comparable. MSC/NASTRAN GPSTRESS outputs the stress at the element integration points to the corner nodes. I-DEAS/SDRC uses the element stress and the shape function to extrapolate the stresses to the element corner nodes. Displacement information however compares well in all cases.

2) The substitution of I-DEAS/MODEL SOLUTION for MSC/NASTRAN therefore, requires careful analysis and review by the end user for their own special and unique situations in the stress analysis to be performed. The PQS (Parabolic Quadrilateral thin Shell element) is the better choice if a reasonably comaparable stress analysis result is required and shell elements are required.

(3) The labor time involved is approximately 3:1 in favor of I-DEAS/SDRC processing mainly because of the ease of performing menu driven graphical interactive model creation techniques as opposed to batch files and bulk data deck manipulations when using MSC/NASTRAN. This is especially pertinent when we consider the model code debugging effort required to obtain a solution in MSC/NASTRAN bulk data files.

CONCLUSIONS

Considerable differences have been observed when a FEM model is analyzed for static loads using different solvers to obtain the stresses. Because different solvers interpret and present information in different ways care must be taken when reviewing information performed using different solvers. This is especially so in regions of rapid stress change or at points close to singularities as the study shows. While these differences are no suprise to veteran users of FEM codes, new users usually assume the results should reasonably correspond whichever solver code was used.

The most consistent plate element for almost all types of stresses is the Parabolic Quadrilateral Shell element (PQS). This element is highly recommended when performing stress analysis using plate elements, as the lowest order that should be choosen for consistent results between the different solvers and post processing schemes.

Even though there will be differences the advantages of a graphical color post processing scheme such as that offered by using SDRC/I-DEAS is irresistible. The options available to us at the present time, consisting of bulk output data and rudimentary plotting options cannot be compared with the advantages of a graphical color presentation performed on SDRC/I-DEAS finite element package.

Time savings is also a critical factor when results are usually needed as soon as possible. This is a major advantage that SDRC/I-DEAS post processing offers to stress analysts especially when presentation of results is critical to the understanding of the overall structural behavior or stress distribution. Something that cannot be discerned easily by looking at a large data output file without considerable time and effort involved.

At the time of writing we are aware of the new pre/post processing software that is beginning to be available with MSC NASTRAN and implementation in a PC or small machine environment. This will perhaps be the best option in the future when the implementation of these options occur and the use of one family of codes for the complete pre and post processing of a stress problem.

ACKNOWLEDGEMENTS

Authors would like to express appreciation for the substantial support from Dr. George Feng, Dr. Dudley Nelson and Mr. Kas Kasbaum. Also appreciated are NASA/JSC management personnel Dr. William Schneider and Mr. Larry Ratcliff.

TABLE 1

FLANGE MODEL DEXTEROUS END EFFECTER

STRESS KSI DISPL INCHES	MAX PRIN	MIN PRIN	MAX SHEAR	VON MISES	MAX DISPL
NODE ELEMENT 472	922	382	922	922	687
I-DEAS/MODEL SOL	7.49	-4.2	4.17	7.49	0.91E-3
MSC/NASTRAN	9.3	-5.1	N/R	10.4	1.065E-3
NASTRAN + I-DEAS POST PROCESSING	7.46	-5.15	5.15	9.18	1.065E-3

N/R : Not Requested from MSC/NASTRAN output

TABLE 2A

GAS BEAM FSE ADAPTOR PLATE
TOP SURFACE STRESSES

STRESS KSI DISPL INCHES	MAX PRIN	MIN PRIN	MAX SHEAR	VON MISES	MAX DISPL
STRESS AT NODES	1138	1030	1138	1138	1076
TOP SURFACE OF SHELL ELEMENT		PLATE			PLATE
I-DEAS/MODEL SOLN	4.9	-4.7	2.16	4.6	2.18E-3
MSC/NASTRAN	4.67!	-1.81#	2.87!	4.1	2.36E-3*
NASTRAN + I-DEAS POST PROCESSING	4.17	-2.77	2.26	4.1	2.36E-3*

Node 1138 belongs to a solid element.
Node 1081
! Node 1079 Plate
Node 1043 Plate

TABLE 2B

GAS BEAM FSE ADAPTOR PLATE BOTTOM SURFACE STRESSES

STRESS KSI	MAX PRIN	MIN PRIN	MAX SHEAR	VON MISES
STRESS AT NODE	1138	1065	1138	1138
BOTTOM SURFACE of SHELL ELEMENTS		PLATE		
I-DEAS MODEL SOL	5.97	-3.0	2.41	5.17
MSC/NASTRAN	4.96#	-2.49*	2.87#	5.39#
NASTRAN + I-DEAS POST PROCESSING	4.17	-2.1	2.26	4.1

Node 1138 belongs to a solid element.

Node 1079 Plate
* Node 1072 Plate

TABLE 2C

GAS BEAM FSE ADAPTOR PLATE
PLATE ELEMENTS ONLY

STRESS KSI DISPL INCHES	MAX PRIN	MIN PRIN	MAX SHEAR	VON MISES
STRESS AT NODE TOP SURFACE of SHELL ELEMENT	1043	1030	1043	1043
I-DEAS/MODEL SOL	4.9	-4.7	2.16	4.6
MSC/NASTRAN	4.67*	-1.81#	2.87*	5.29*
NASTRAN + I-DEAS POST PROCESSING	2.46	-2.77	1.79	3.17

* Node 1079
\# Node 1043

TABLE 3A

TEST CASE
DESCRIPTION OF ELEMENT TYPES USED

SDRC/I-DEAS	MSC/NASTRAN	short-form	
Linear Quadr. Shell	CQUAD4	LQS	S/M/C
Parab. Quadr. Shell	CQUAD8	PQS	S/M/C
Linear Tri. Shell	CTRIA3	LTS	S/M/C
Parab. Tri. Shell	CTRIA6	PTS	S/M/C
Solid Linear Brick	CHEXA	LSL	S/M/C
Solid Parab. Brick	CHEXA	PSL	S/M/C

The fourth letter S, M, C denotes the following:

S - SDRC/Model Solution Method
M - MSC/NASTRAN Solution Method
C - Combined Nastran Solution post processed in I-DEAS/SDRC

TABLE 3B

TEST CASE
FINITE ELEMENT MESHING INFORMATION

Element Type	No. of Elements (Length X Circ.)	No. of Nodes
LQS*	240 (20 X 12)	252
PQS*	80 (10 X 8)	256
LTS*	480 (20 X 12)	252
PTS*	160 (10 X 8)	336
LSL*	240 (20 X 12)	504
PSL*	80 (10 X 8)	600

TABLE 4

TEST CASE
MAXIMUM STRESSES COMPARISON

TEST CASE RESULTS STRESS : KSI DISPL : INCHES

STRESSES		MAX PRIN	MIN PRIN	SHEAR MAX	VON MISES	DISPL MAX
LOCATION		C'	C	C	C	C
SOLN TYPE						
		(REFER TO FIGURE 4 FOR LOCATION)				
LQS	M	78.87	-147.5	74.5	148.2	.0796
	C	70.04	-106.7	55.5	105.1	.0796
	S	241.8	-343.7	159.9	310.8	.0805
PQS	M	220.7	-303.6	153.0	304.9	.08206
	C	220.6	-303.6	153.0	304.8	.08206
	S	212.9	-284.4	152.0	288.4	.08137
LTS	M	102.6	-171.7	88.42	174.3	.07581
	C	70.66	-100.7	53.82	101.1	.07581
	S	201.0	-230.2	107.5	208.7	.07340
PTS	M	169.8	-244.7	118.8	245.1	.07981
	C	170.6	-244.4	118.6	240.9	.07981
	S	57.68	-93.27	44.59	83.1	.05984
LSL	M	47.27#	-128.3	N/R	136.4	.06745
	C	47.27#	-100.8	63.62	111.5	.06745
	S	86.76*	-80.91	52.48*	91.02*	.06198
PSL	M	147.7	-265.0	N/R	270.4	.07557
	C	138.9	-236.8	110.1	228.0	.07557
	S	53.07#	-124.9	52.41	93.25	.05694

N/R Not requested from NASTRAN Output.
* at point other than C and C'
# at point A

TABLE 5

TEST CASE
POINT B STRESSES COMPARISON

STRESS IN KSI / DEFL IN IN.

ELEMENT TYPE		MAX PRIN	MIN PRIN	MAX SHEAR	VON MISES	DEFL (X 10-2)
LQS	M	25.7	1.68	12.01	24.9	1.584
	C	25.7	1.56	12.07	24.98	1.584
	S	28.98	3.20	12.98	27.47	1.589
PQS	M	29.58	2.8	13.39	28.29	1.519
	C	29.58	2.80	13.39	28.29	1.519
	S	29.39	2.58	13.41	28.19	1.519
LTS	M	24.53	1.86	11.34	23.65	1.503
	C	25.01	1.29	11.86	24.34	1.499
	S	25.17	1.92	11.63	24.19	1.498
PTS	M	24.9	-2.22	13.56	26.08	1.522
	C	24.88	-2.22	13.54	26.05	1.522
	S	20.19	-1.86	11.03	21.24	1.502
LSL	M	26.23	0.53	N/R	25.18	1.616
	C	26.33	0.73	12.8	25.10	1.638
	S	24.71	1.28	11.71	23.33	1.638
PSL	M	27.18	-.33	N/R	26.87	1.517
	C	27.18	-.31	13.75	26.82	1.517
	S	21.88	1.40	10.24	21.11	1.511

N/R Not requested from NASTRAN output.

TABLE 6

TEST CASE
LONGITUDINAL FIBER STRESSES

STRESS LOCATION		STRESS IN KSI A	B	C	D	E	F
		(REFER TO FIGURE 4 FOR LOCATION)					
ELEMENT TYPE	SOLN						
LQS	M	43.7	25.7	15.3	-3.37	-15.0	-47.3
	C	40.8	25.7	3.16	-5.31	-15.0	-44.7
	S	53.8	29.0	-42.1	-.276	-14.8	-58.8
PQS	M	46.1	29.6	2.39	-3.64	-10.64	-51.6
	C	46.1	29.6	2.39	-3.64	-10.64	-51.6
	S	47.6	29.4	19.5	-1.41	-10.96	-53.1
LTS	M	43.7	24.1	24.1	-6.84	-13.08	-46.4
	C	39.2	25.0	6.0	-4.81	-14.05	-42.3
	S	44.7	25.2	-31.	-3.22	-14.04	-48.9
PTS	M	45.5	29.4	-5.65	-7.52	-14.11	-50.4
	C	45.3	29.4	-7.68	-5.68	-14.13	-50.2
	S	45.1	20.2	34.87	-2.05	-19.87	-44.9
LSL	M	47.2	25.3	26.5	-5.89	-18.89	-48.6
	C	47.2	26.3	26.5	-5.89	-18.89	-48.6
	S	47.5	24.7	-7.73	-11.2	-20.33	-47.8
PSL	M	49.5	27.2	-2.13	-3.28	-13.8	-55.3
	C	49.5	27.2	-2.13	-3.28	-13.8	-55.3
	S	48.3	21.9	-20.2	-9.87	-19.0	-48.5

TABLE 7

TEST CASE PERFORMANCE INFORMATION

	Solver	CPU Time (sec.)	Elapse Time (min.:sec.)
LQS	MSC	62.47	4:02 (D)
	I-DEAS	83.89	3:47 (D)
PQS	MSC	108.11	4:35 (N)
	I-DEAS	79.02	2:40 (N)
LTS	MSC	66.33	3:29 (D)
	I-DEAS	83.89	3:22 (D)
PTS	MSC	136.80	5:44 (D)
	I-DEAS	116.80	3:54 (D)
LSL	MSC	129.60	7:43 (D)
	I-DEAS	88.86	4:55 (D)
PSL	MSC	207.10	13:22 (D)
	I-DEAS	153.10	10:14 (D)

D denotes the daytime (8:00 AM - 4:45 PM) at weekdays.
N denoted non-daytime.

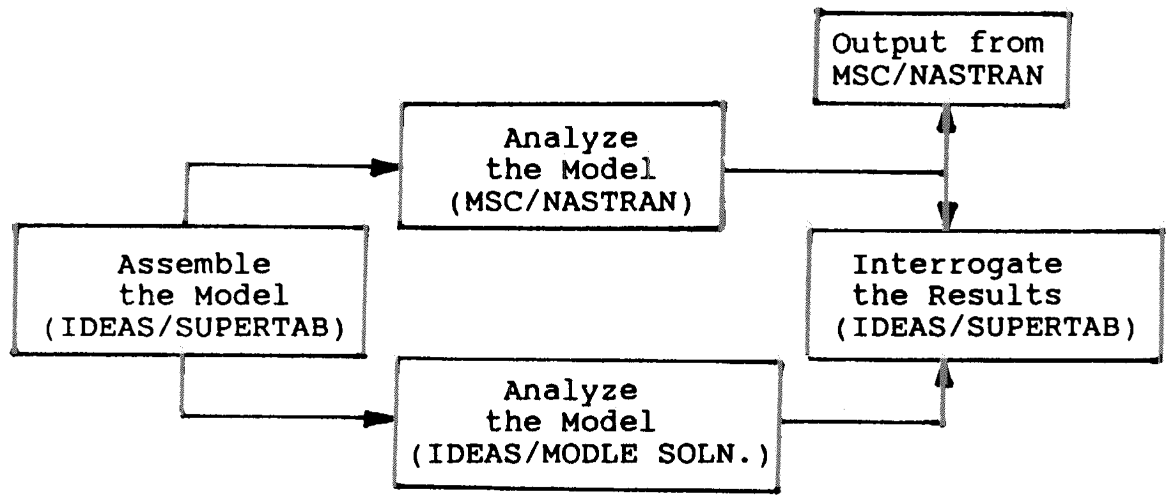


Figure 1 Flow Chart of Solutions

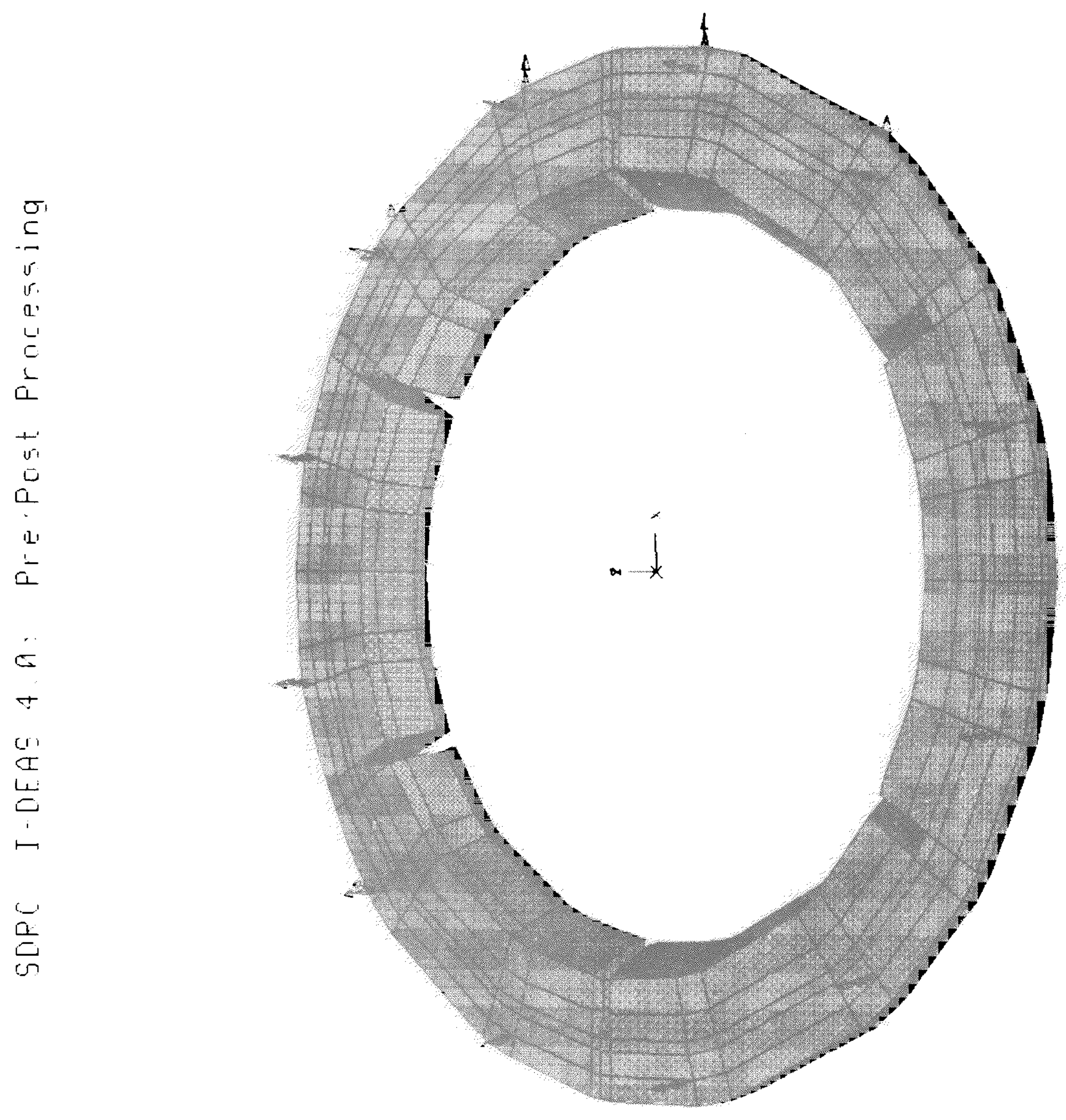


Figure 2 FLANGE MODEL (DEXTERIOUS END EFFECTER)

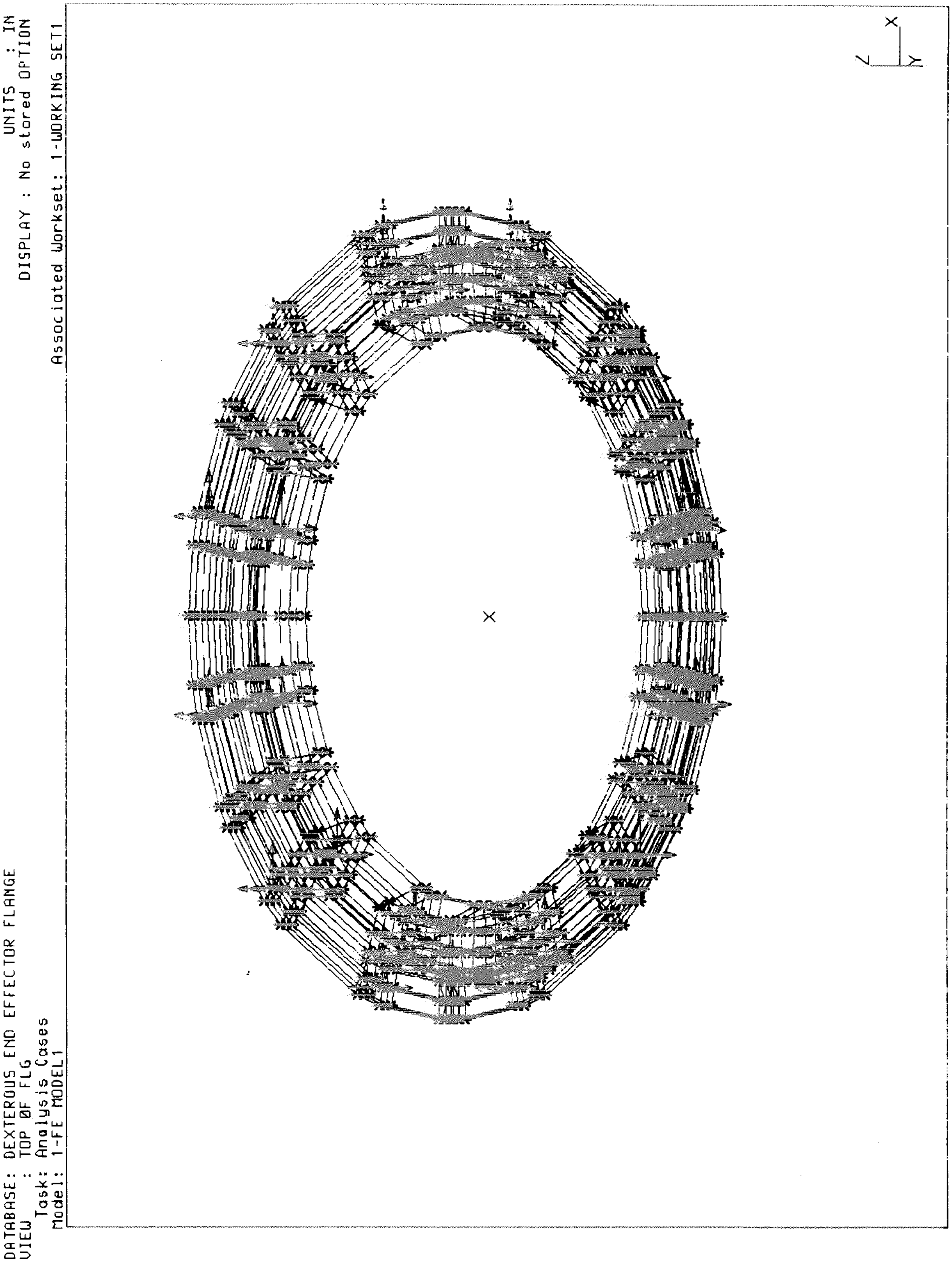


Figure 2B Flange Model Node Plot.

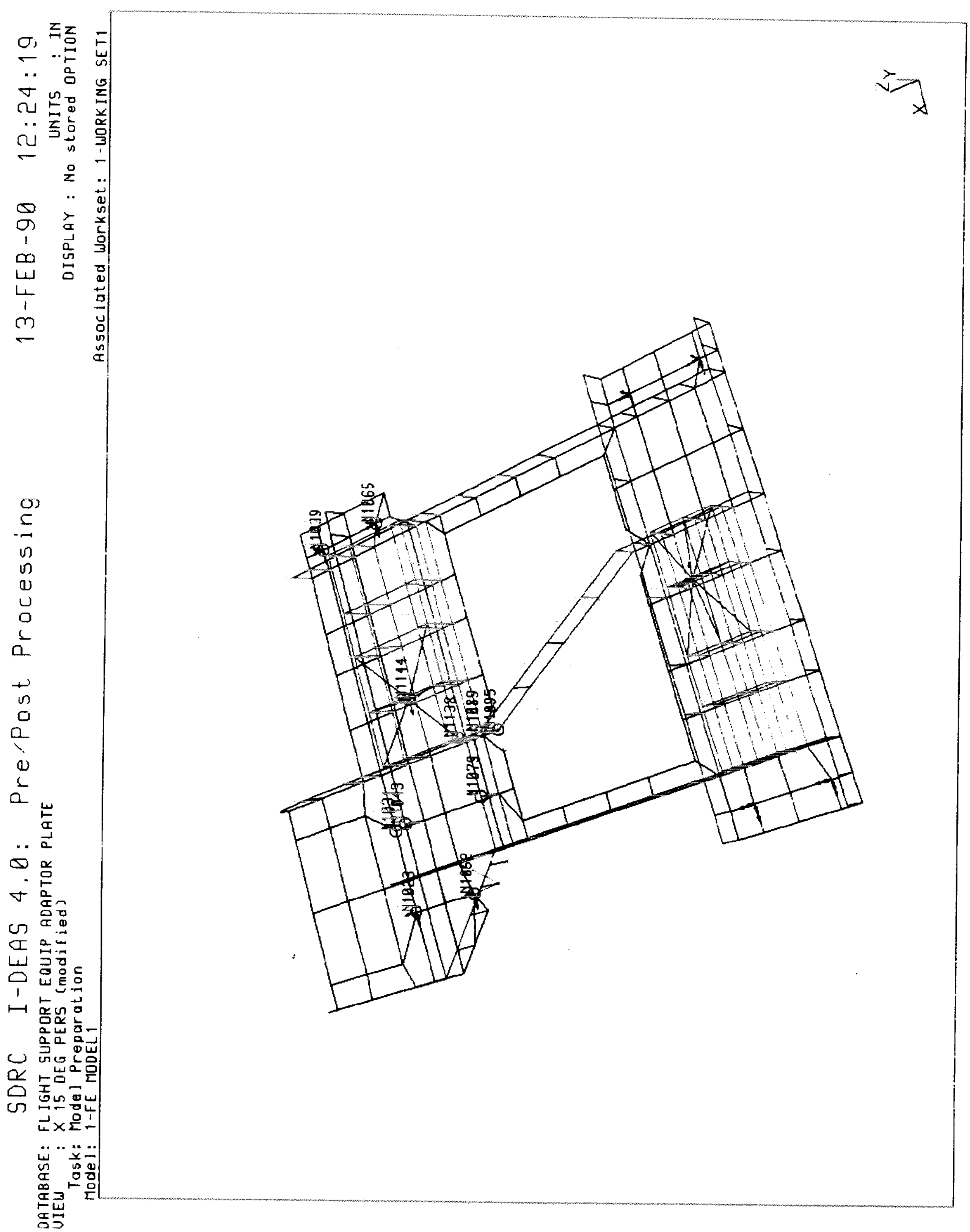
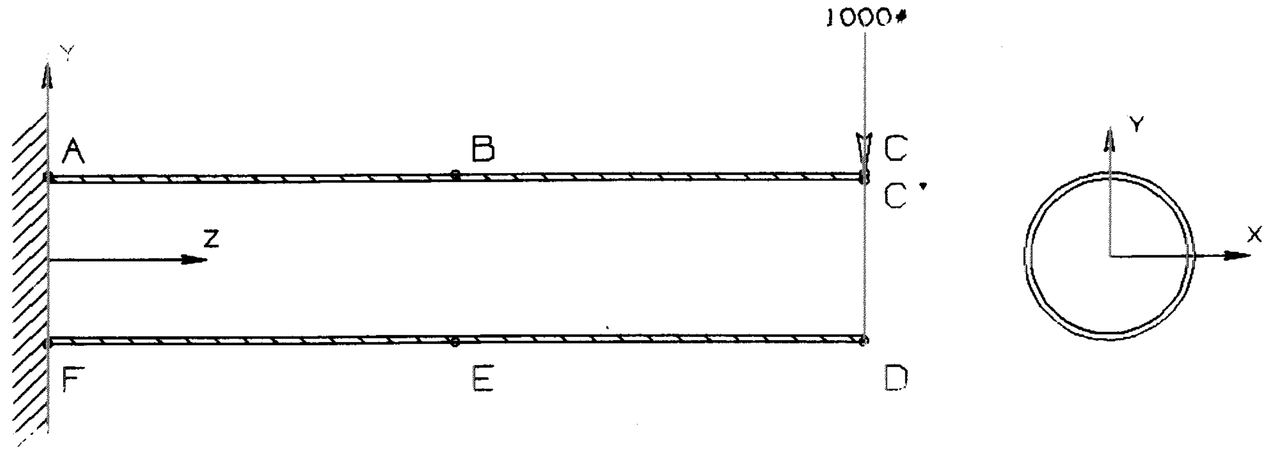


Figure 3 Adaptor Plate Model



Coordinates :

A (0., 1.04, 0.)
B (0., 1.04, 5.)
C (0., 1.04,10.)
C' (0., 0.96,10.)
D (0.,-1.04,10.)
E (0.,-1.04, 5.)
F (0.,-1.04, 0.)

Figure 4 TEST CASE