COMPARISON OF MSC/NASTRAN ANALYSIS RESULTS TO SOME WELL KNOWN CLOSED FORM SOLUTIONS TO GUIDE MODELING OF VEHICLE STRUCTURES

Bijan K. Shahidi
John V. Fazio

Ford Motor Company
Dearborn, MI USA

ABSTRACT

The art of modeling is a basic yet integral part in obtaining a valid correlation between numerical (FEA/CAE) analysis and vehicle testing. This perhaps is the key ingredient in creating a confidence level among designers, analysts and test engineers so that they can derive the best design using CAE and analytical prototypes.

There are several options available in MSC/NASTRAN when one desires to create a finite element model of a vehicle structure. The question that remains unanswered is, what is the best way to model a vehicle in a real world environment economically, and believe in the results without creating the actual part for testing? In order to gain some insight and answer this question, one often looks into the well-known classical mechanics problems in literature where there is either a closed form solution or a repeatable lab experiment conducted in a controlled environment, to compare with modeling techniques.

In this paper, several classical problems are sought from literature, which are modeled using MSC/NASTRAN, and the results are compared to one another. The problems range from linear static analysis of slender beams to free vibration and nonlinear static analysis. The conclusions are drawn from the comparison of several modeling methods to the closed form solutions available to the authors . It is found that one must take great caution when modeling a vehicle structure on choice of element types, their size and range validity. Discussions on the accuracy of results in deflection analysis vs. stress or vibration analysis are made by using different modeling methods and rationalizing the comparison of the results to the analytical solutions.

INTRODUCTION

In MSC/NASTRAN there are several ways to model and simulate the structural performance of car or truck like vehicles. For instance, the beam like structure of a rocker panel can be modeled using cbeams or cbars as well as using cshells, similarly joint mechanisms such as a-pillar to roof or b-pillar to rocker can be simulated with number of c-elas elements or it can be represented by its own geometry using cquads.

The interesting and sometimes difficult question to answer is which way is the correct way to model and how does it correlate to real situations. Obviously, the better and simpler answer is how accurate does one want the result be, better yet how much does one want to pay for the result or what information one has about the structure. In other words, at what stage of the design one desires the results and how fuzzy the constrains and loads are at the time of analysis.

In order to take a shot at these questions and rationalize the thinking process behind the decisions one would want to make in creating such math models, the authors have tried to compare the results obtained from several MSC/NASTRAN models of beam like structures by modeling them different ways and comparing the obtained MSC results to classical beam theory, closed form solutions.

Different discipline of mechanics such as linear static analysis, free vibration analysis and plasticity are examined within the context of beam theory. The comparisons of the different models using beam elements as opposed to shell elements, the number of elements verses the validity of beam theory (the length of the beam as compared to the thickness of the shell) are all compared to the closed form solutions from Timoshenko's various books [4][5][6] in these subjects.

MODEL

There are several models created to evaluate the validity (its range) of the results when compared to the classical closed form solutions of beam like structures. For this purpose, a cantilevered beam fixed at one end and free at the other was modeled as several cbars, later the model was switched to shell elements. The thickness of the shell element, the number of elements, and length of beam were varied and compared against the theoretical results.

ANALYSIS

Several different analysis were cited for the cantilever beam problem. The area of interest ranged from deflection and stress analysis in linear static sol 101 analysis to vibration analysis sol 103. Later small strain plasticity analysis added into the list since the recent trends in design requires such tools to replace test and verification criteria for vehicle and component designs.

DEFLECTION

The first area of investigation was the comparison between the closed form solution of deflection to the finite element analysis prediction using MSC/NASTRAN. Several finite element analysis was carried out using beam and shell element models. The results of the different models allowed for the comparison of the number of elements and their effect on the accuracy of the deflection correlations. Hand calculated deflection values were obtained by using the deflection equation below.

$$y(x) = \frac{PL^3}{6EI} - \frac{P(L-x)^3}{6EI} - \frac{PL^2 x}{2EI}$$

It appears that as the number of elements increased the greater the difference it resulted from the comparison of the FEA to the closed form solution. The magnitude of deflection increased as the number of elements increased. The difference in values changed parabolically along the length of the model, with its maximum at the mid-section of the beam (Figure 1). Even though the difference in deflection analysis became a maximum at the mid-section, and with the deflection increasing along the length, the error decreased exponentially along the length of the beam (Figure 2).

All beams modeled in shell elements followed this trend. According to the comparisons there is a minimum number of elements to be used when modeling that will allow one to obtain the most accurate values for the deflection of a modeled structure (Figure 3).

It was observed in the FEA results when the beam was modeled using cbars, closely matched the closed form solution (Figure 3). It is therefore recommended that whenever possible take advantage of cbars wherever appropriate.

STRESS

Maximum stress was the next area of investigation in the analysis. From classical beam theory, stresses at the top and bottom flanges where they were maximum, compared to the FEA results. This was also used as a verification between Nastran and hand calculated results.

In this part of the analysis the attention was given to the stresses at the top and bottom of the I-Beam’s cross section. The stresses varied slightly along the width of the flange (Figure 4), and therefore simplify of the analysis, these values were averaged. Error calculations where formulated by averaging the magnitude of the top and bottom flange's average stress values (Figure 5).

In the stress analysis the attention was given to the shell element modeling and the cbar element modeling was not proceeded for this purpose. Two of the shell element model were tested to see the effect of the number of elements on the stress correlation's. The general trend was as elements became smaller in size, and the number of them increased, the error decreased (Figure 3).

There was some noise in the data at both ends of the I-Beam (Figure 4). This scattering of data was due to local effects between elements. This data was neglected in the computation of the error. The theoretical stress calculations do not apply to the boundaries of the I-Beam model, and cannot be used to estimate the stresses at the boundaries in real life structures (Figure 4).This is partially due to the saint venant's boundary effects.

NORMAL MODES

The third area of interest was the accuracy of the models in the first three natural bending frequencies. Four key modeling characteristics were examined during the analysis of the beam structures. The analysis investigated the significance of the length, number of elements, type of element, and the thickness of elements of the model.

For each theoretical model the natural bending frequencies where determined by the equations shown below [3].

$$\lambda^4 = \frac{\left(\rho A \omega^2\right)}{EI}$$

$$\omega_r = \frac{\left(\lambda_r L\right)^2}{L^2}\left(\frac{EI}{\rho A}\right)^{\frac{1}{2}}$$

In the analysis the longer the beam model the closer the results correlated to the theoretical models. The increase from 254.0 to 1270.0 mm in all cases lowered the error considerably (Figure 6). Along with the overall size of a model, the number of elements can play one of the most important roles in modeling.

Although an increase in elements due to the increased length did reduce error for the first natural bending modes, the error in the second and third modes were lower for the models with fewer number of elements (Figure 6).

As the elements increased in number, when using 2nd order shell elements, the first bending modes error was less then those designed using first order elements, but for the second and third modes the error was greater for the second order element models (Figure 6).

Also if the thickness of the shell elements was reduced from 3.0 to 0.5 mm the error for the first mode dropped considerably. As before with elements increasing in number and second order ones used the error for the modified thickness was larger than the original for the second and third modes (Figure 6).

For the first bending mode, three models had the least amount of error. These three models all where 1270.0 mm long, with 0.5 mm thickness, and modeled with 2nd order elements, but each had different amounts of elements. The error was independent of the number of elements used. This observation that the number of elements for this model had no baring on the error produced, would imply that they should not be of major concern when designing complex structures which are primarily being used for natural frequency observations in low frequency computation.

PLASTICITY

The next area of concern was the analysis of plasticity and catastrophic failure. All the models in this analysis followed a consistent trend. The most apparent observation was that the elements in the model took into account local effects that the analytical method did not.

It is observed that at the fixed end of the beam the plasticity boundary layer differs from the classical beam formulation (Figures 7 & 8). The effects of adjacent elements, and boundary conditions caused this change. This variance in the boundary regions is apparent in all models, but is more distinctive as the models element number increases. The plasticity beam formulation is given by the equation below.

$$y = \frac{\sigma I}{Px}$$

More elements led to a stiffer structure, and more local effects, which allowed the model to resist catastrophic failure slightly longer then those models composed of fewer and larger elements. This analysis produced the length of the plasticity region compared to the classical solution and the magnitude of stresses when the full plastic hinge occurs.

In this analysis it is obvious that there is a discrepancy at the boundary region of every model. The boundary region is the main focal point in plasticity and failure analysis, and these results cause some uncertainty in how to properly design components that will meet durability in non-linear analysis.

CONCLUSION

For the analysis of complex vehicle structures, MSC/NASTRAN proved to be a useful tool in predicting various mechanical phenomena. It is recommended that early in the design cycle where there is little information available about the characteristics of the design, beam elements proved to be accurate for static analysis and should be used wherever possible. For dynamic analysis, specially when the basic characteristics of the structure is sought, one needs to have a deeper understanding of the structure in which shell elements are recommended. It is also observed that for economics of the computation, there is always an optimum number of elements that produce the same results as a more detailed treatment of the structure. It is however problem dependent, and it is suggested that the minimum number of elements should be used until the analyst has found more experience with the vehicle components he or she is designing.

Figure 1

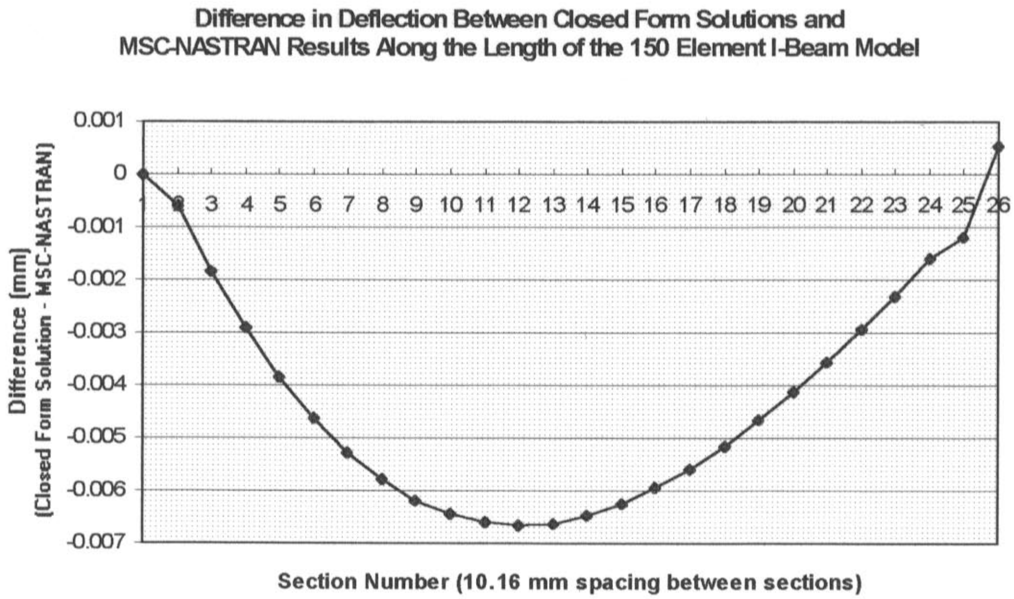


Figure 2

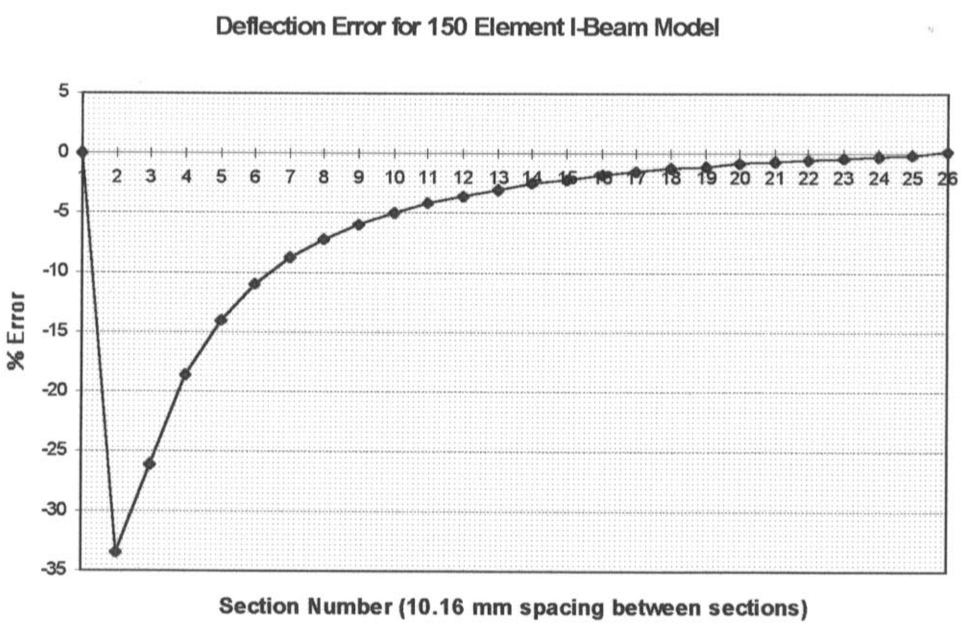


Figure 3

Model	% Displacement Error	% Stress Error
I-Beam (Elem=150)	2.77	4.82
I-Beam (Elem=900)	3.42	4.50
I-Beam (Elem=3600)	3.52	N/A
C-Bar (Elem=25)	0.00014	N/A

Figure 4

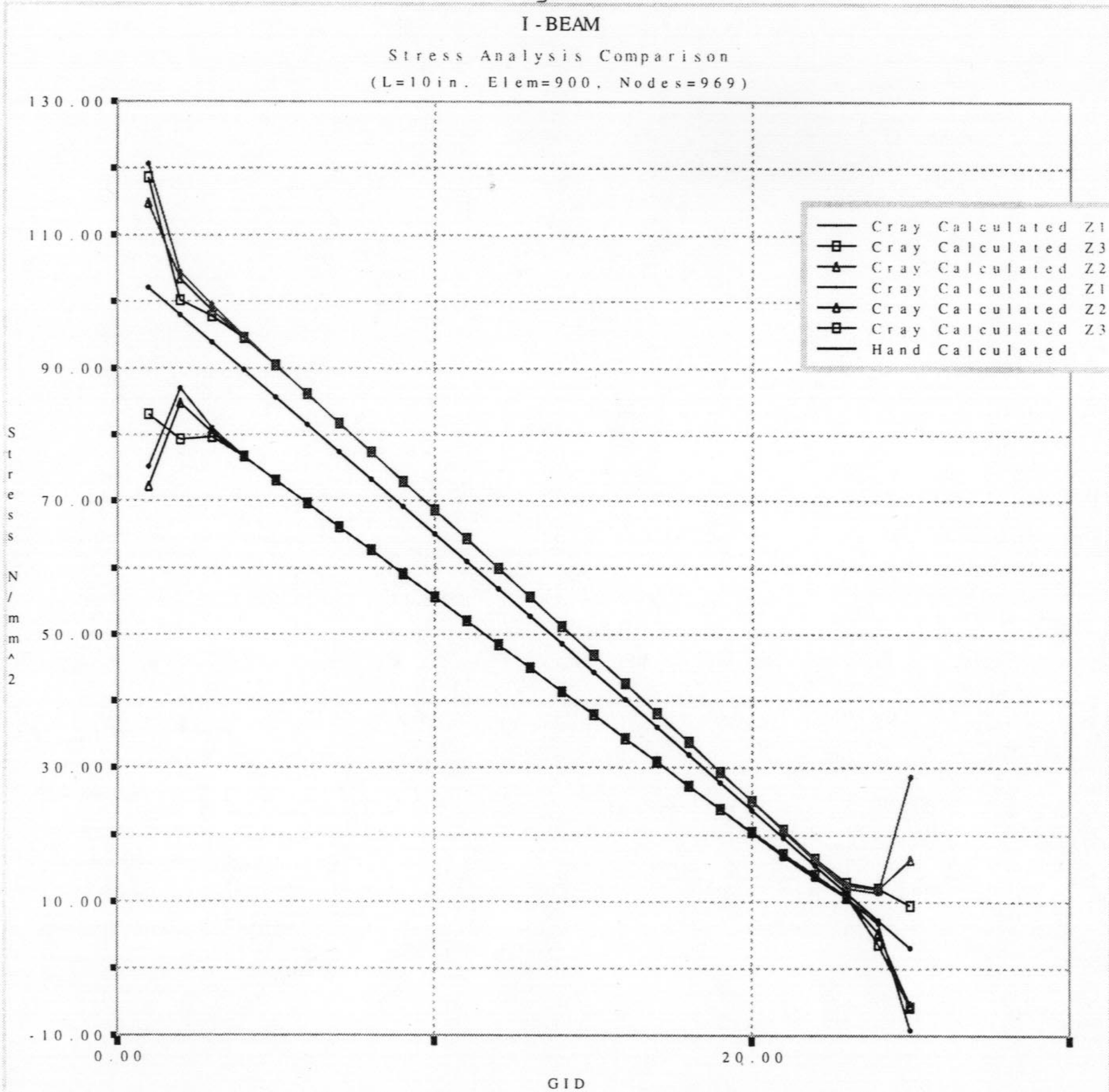

I-BEAM
Stress Analysis Comparison
(L=10in. Elem=900, Nodes=969)
130.00
110.00
90.00
70.00
50.00
30.00
10.00
-10.00
Stress N/mm^2
0.00
20.00
GID
Cray Calculated Z1
Cray Calculated Z3
Cray Calculated Z2
Cray Calculated Z1
Cray Calculated Z2
Cray Calculated Z3
Hand Calculated

Figure 5

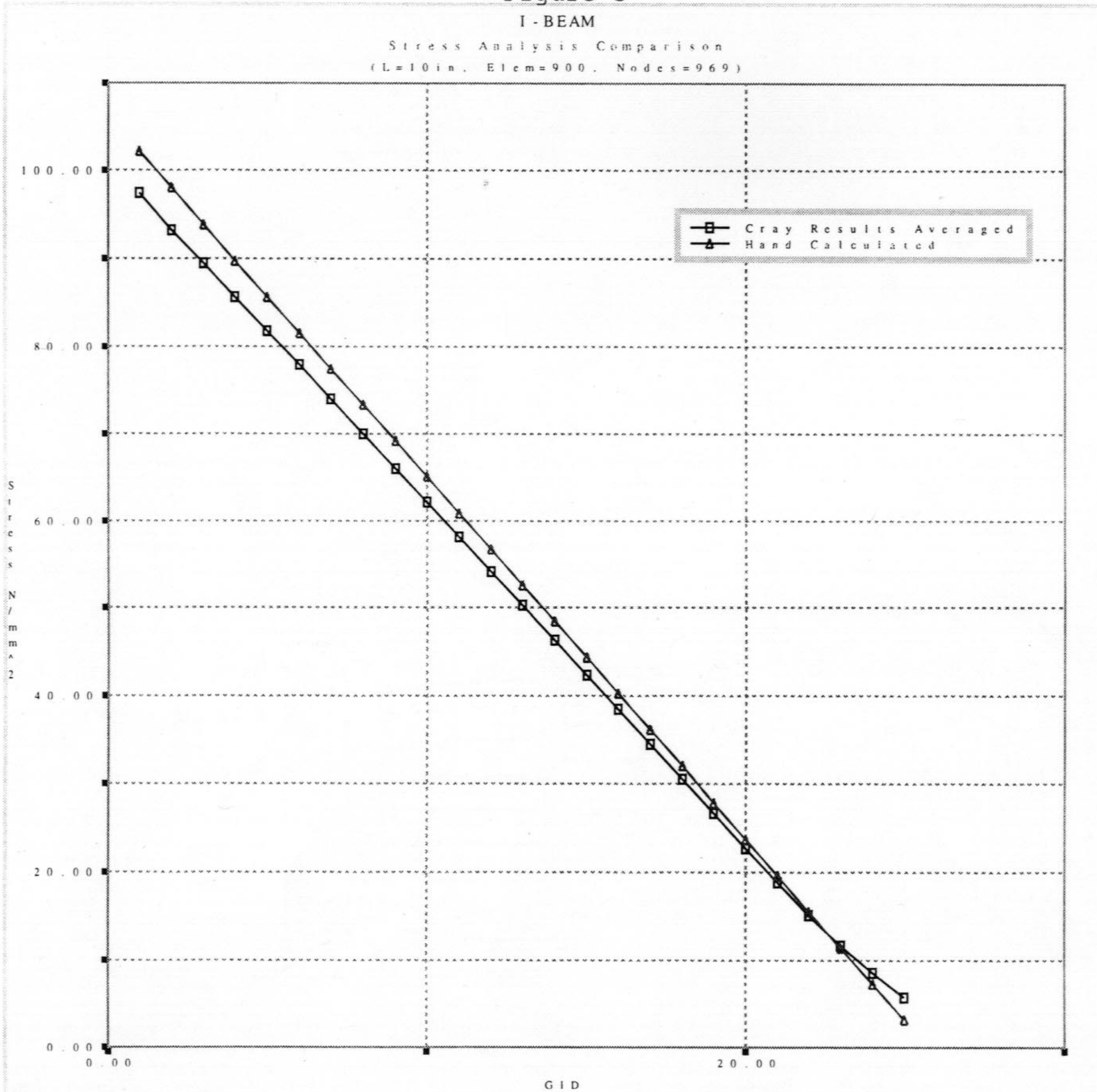
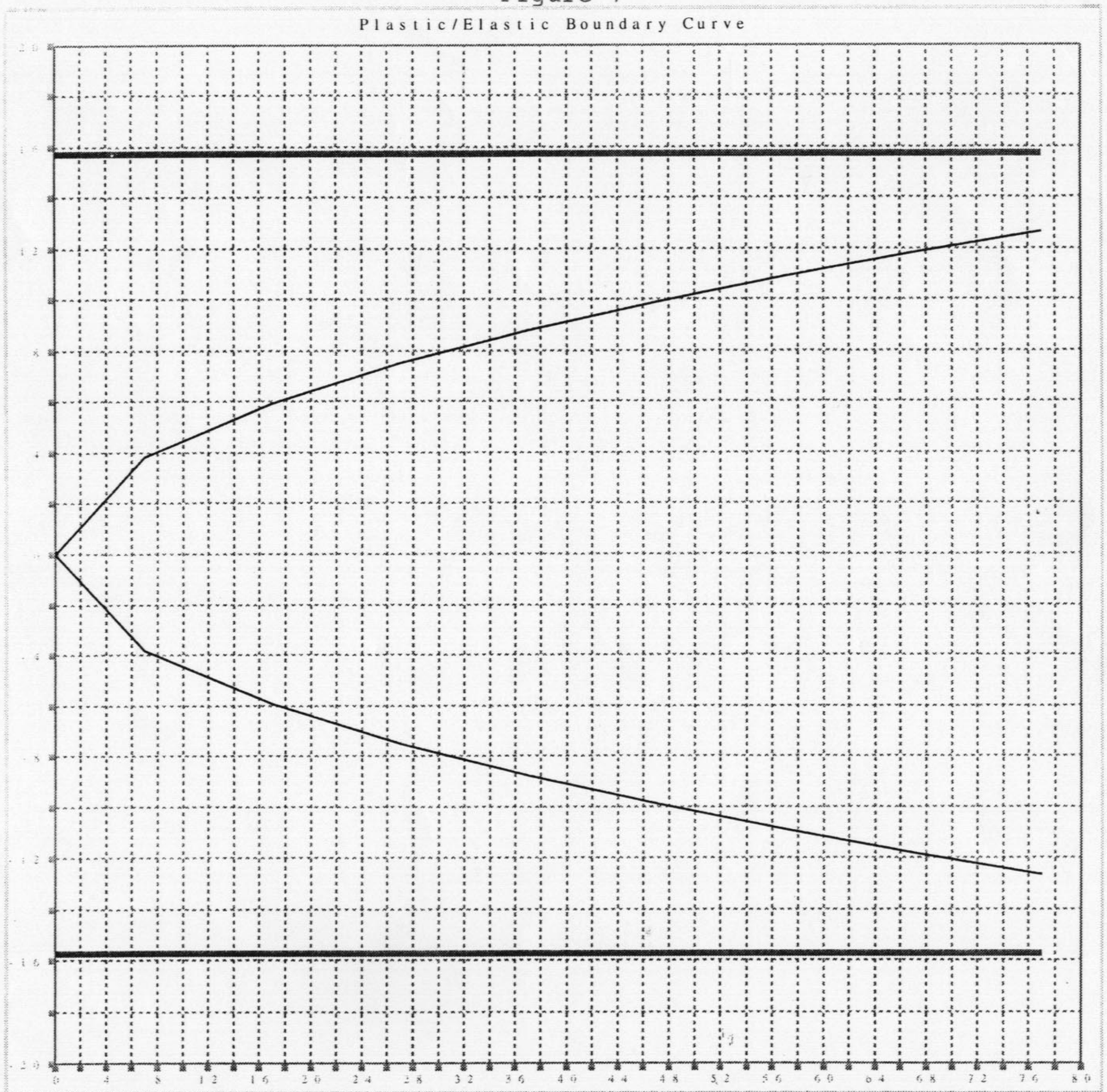

I-BEAM
Stress Analysis Comparison
(L=10in, Elem=900, Nodes=969)
Cray Results Averaged
Hand Calculated
100.00
80.00
60.00
40.00
20.00
0.00
Stress N/mm^2
0.00
20.00
GID

Figure 6

Beam Normal Modes Analysis

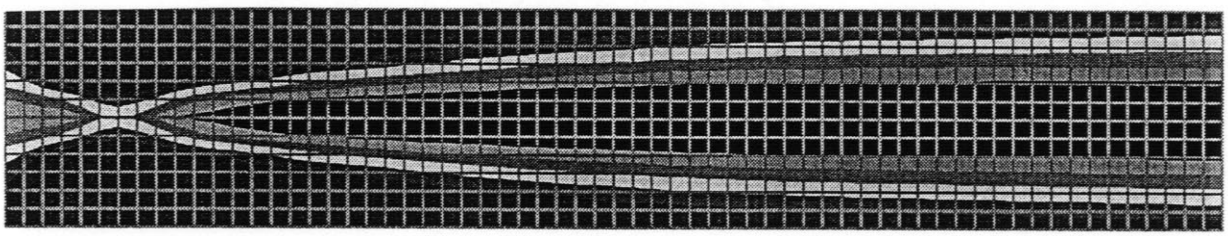
DESCRIPTION					1st VERTICAL BENDING MODE			2nd VERTICAL BENDING MODE			3rd VERTICAL BENDING MODE		
TYPE	MODEL	LENGTH	ELEMENT TYPE	THICKNESS (mm)	HAND (hz)	NASTRAN (hz)	ERROR (%)	HAND (hz)	NASTRAN (hz)	ERROR (%)	HAND (hz)	NASTRAN (hz)	ERROR (%)
CBAR	CBAR (ELEM=25)	254.0	1st Order	DNA	550.66	552.02	+.2470	3450.22	3453.17	+.0855	9662.80	9653.18	-.0996
SHELL	I-BEAM (Close Sec.) (ELEM=1500)	254.0	1st Order	3.0	475.43	475.55	+.0252	2979.51	2658.94	-10.76	8342.78	6506.76	-22.00
	I-BEAM (Close Sec.) (ELEM=7500)	1270.0	2nd Order	3.0	19.02	19.39	+0.518	119.18	120.83	-0.074	333.71	335.49	-0.910
	I-BEAM (Close Sec.) (ELEM=7500)	1270.0	2nd Order	0.5	19.29	19.36	+0.518	120.92	120.67	-0.207	338.57	334.98	-1.060
	I-BEAM (ELEM=150)	254.0	1st Order	3.0	550.66	531.15	-3.544	3450.22	2663.26	-22.693	9662.80	5808.08	-39.892
	I-BEAM (ELEM=750)	1270.0	1st Order	3.0	22.10	22.19	+0.407	138.48	137.42	-0.765	387.75	377.80	-2.566
	I-BEAM (ELEM=750)	1270.0	2nd Order	3.0	22.10	22.14	+0.181	138.48	137.14	-0.968	387.75	376.99	-2.775
	I-BEAM (ELEM=750)	1270.0	1st Order	0.5	22.12	22.15	+0.135	138.62	137.09	-1.104	388.15	374.79	-3.442
	I-BEAM (ELEM=750)	1270.0	2nd Order	0.5	22.12	22.11	-0.045	138.62	136.84	-1.284	388.15	374.78	-3.445
	I-BEAM (ELEM=900)	254.0	1st Order	3.0	550.66	530.08	-3.737	3450.22	2667.09	-22.698	9662.80	5930.52	-38.625
	I-BEAM (ELEM=4500)	1270.0	2nd Order	3.0	22.10	22.14	+0.181	138.48	137.14	-0.968	387.75	377.00	-2.772
	I-BEAM (ELEM=4500)	1270.0	2nd Order	0.5	22.12	22.11	-0.045	138.62	136.84	-1.284	388.15	374.92	-3.408
	I-BEAM (ELEM=3600)	254.0	1st Order	3.0	550.66	529.93	-3.765	3450.22	2667.51	-22.686	9662.80	5941.22	-38.515
	I-BEAM (ELEM=18000)	1270.0	2nd Order	3.0	22.10	22.14	+0.181	138.48	137.13	-0.975	387.75	377.00	-2.772
	I-BEAM (ELEM=18000)	1270.0	2nd Order	0.5	22.12	22.11	-0.045	138.62	136.84	-1.284	388.15	374.94	-3.403
SOLID	I-BEAM (ELEM=2550)	254.0	1st Order	3.0	550.66	531.33	-3.510	3450.22	2726.11	-20.987	9662.80	6163.90	-36.217
	I-BEAM (ELEM=2550)	254.0	2nd Order	3.0	550.66	530.77	-3.612	3450.22	2716.55	-21.264	9662.80	6132.05	-36.540
	RECTANGLE ELEM=400	254.0	1st Order	DNA	405.02	402.75	-.5604	2538.25	2365.73	-6.797	7107.22	6085.43	-14.377
	RECTANGLE ELEM=800	254.0	1st Order	DNA	405.02	402.25	-.6839	2538.25	2363.34	-6.891	7107.22	6078.36	-14.476

Figure 7



Plastic/Elastic Boundary Curve
20
16
12
8
4
0
-4
-8
-12
-16
-20
0 4 8 12 16 20 24 28 32 36 40 44 48 52 56 60 64 68 72 76 80

Figure 8



REFERENCES

[1] Fertis, D. G. 1995. *Mechanical and Structural Vibrations*, John Wiley and Sons, Inc., NY, pp. 95-101.

[2] Hibbeler, R. C. 1994. *Mechanics of Materials*, 2nd ed., Macmillan College Publishing Co., NY, pp. 271, 286-293.

[3] Inman, D. J. 1994. *Engineering Vibration*, Prentice-Hall, Inc., NJ, pp. 329-336

[4] Timoshenko, S. 1956. *Strength of Materials*, Prt. 1, Krieoer

[5] Timoshenko, S. 1956. *Strength of Materials*, Prt. 2, Krieoer

[6] Timoshenko, S. 1974. *Vibration Problems in Engineering*, 4th ed., John Wiley and Sons, Inc., NY