Nonlinear Analysis of the Terex Scraper Roll-Over Protective Cab

Charles J. Hunckler[*] and Randy D. Austin[**]

ABSTRACT

A rollover test of the Terex Roll-Over Protective Cab has been conducted using finite element analysis. MSC/NASTRAN Version 61B, which has the capability of including geometric nonlinearities and material nonlinearities, was used. The analysis predicted the extent of permanent deformation of the compartment and energy absorbed during the load test.

The analysis was done at lower cost than hardware testing of a prototype would involve. In the future, this nonlinear finite element analysis can be used to finalize operator compartment designs before hardware tests are conducted. The method reduces the number of hardware tests and hardware changes required, and, therefore, reduces the cost and time to develop the product.

[*] President, Engineering Methods, Inc., Lafayette, Indiana

[**] Supervisor, Computer-Aided Engineering, Terex Corporation, Hudson, Ohio

1. INTRODUCTION

Most construction equipment manufactured in the United States has been equipped with roll-over protective structures seeking to protect the operator in the unlikely event a vehicle may roll over. The Society of Automotive Engineers (SAE) has established some basic recommended standards for the performance of roll-over protective structures (ROPS). OSHA and The Bureau of Mines have required users of earthmoving equipment to install roll-over protective structures and have used SAE Procedure J1040C or its predecessors to define the performance requirements of the ROPS. International Standard ISO 3471 and Canadian Standard CSA B352 are other documents that have similar requirements to SAE J1040C.

The objective of SAE Procedure J1040C "Performance Criteria for Roll-Over Protective Structures (ROPS) for Construction, Earthmoving, Forestry, and Mining Machines" was to establish a consistent and repeatable method of evaluating force-deflection and load carrying characteristics under static loading. Although the SAE procedure does not duplicate structural deformations due to an actual roll, it is expected by the Society of Automotive Engineers that a ROPS structure capable of withstanding these static loads will give crush protection when the vehicle is operating between 0 and 10 mph on a hard clay surface of 30 degrees maximum slope, and the vehicle undergoes a 360 degree roll about the machine's longitudinal axis without the vehicle losing contact with the slope.[1]

The ROPS procedure undergoes constant review by the Society of Automotive Engineers and is subject to revision based on data obtained from actual vehicle roll-overs. Currently the Society of Automotive Engineers is reviewing the potential need for a vehicle to undergo more than one roll. This review may result in future revisions to the ROPS test procedure.

The first step of the SAE J1040C ROPS test places a horizontal load on the side of the cab with requirements of not encroaching on a defined operator zone while attaining a horizontal force requirement and a horizontal energy requirement. The horizontal force requirement is intended by SAE to ensure sufficient strength to penetrate unfrozen soil, thereby giving braking action to discourage further rolling of the vehicle. For prime movers:

$$F\ (LB_F) = 8270 \left[\frac{M\ (LB)}{10000}\right]^{1.20}$$

where M = mass of the prime mover (scraper mass with kingpins and hitches excluded)

F = horizontal force applied to the cab

There is a horizontal energy requirement intended to assure the ROPS will deflect when it impacts a surface that will not significantly deform such as frozen ground, concrete, or rock. For prime movers:

$$U\ (IN{*}LB_F) = 65800 \left[\frac{M\ (LB)}{10000}\right]^{1.25}$$

where U = horizontal energy absorbed

After the cab has been subjected to the lateral load which typically plastically deforms the cab, a vertical load is then applied with a similar requirement of not encroaching on the defined operator zone. The vertical force requirement is intended to assure the deformed ROPS will support the vehicle in an upside down attitude. For all vehicles covered by J1040C:

$$F\ (LB_F) = 2.0\ (\ M(LB)\ )$$

Terex has certified conformance to SAE J1040C by physically testing cabs and canopies. The laboratory technique involves destructively testing a cab or canopy and is a very costly process. The results of the test are pass or fail and provide little guidance on how to most effectively utilize the cab geometry and material.

Nonlinear finite element analysis offers a method to greatly improve the design process. The cab designer is faced with the challenge of providing sufficient stiffness to attain the force requirement while also providing sufficient flexibility to meet the energy absorbtion requirement. The finite element technique can be used to optimize the balance between the stiffness and the flexibility required. The complexity of construction equipment cabs allows only basic sizing calculations to be made with the textbook approach. There are multiple load paths present in a cab due to the number of internal reinforcements used and the number of load carrying columns. Thus the designer is forced to rely most heavily on past experience with similar cabs to predict the success of a new design.

The designer is also frequently under pressure to have a production cab ready in a very tight time frame. The failure of a ROPS test could force rescheduling of a production release date. The fabrication and installation of another pilot cab and retesting may require one to three months in addition to design and drafting time. Likewise, the fabrication and testing costs would be almost doubled.

The negative consequences of having to revise and re-test a cab orients the designer toward a conservative design strategy. The most typical conservative design strategy is to increase stiffness and strength. This approach can be counter productive in ROPS design where increasing the stiffness may reduce the energy absorption capability of the cab.

Many design considerations contributed to Terex interest in nonlinear finite element analysis and can be summarized as follows: roll-over protective cab design is a complex process requiring a proper balance between stiffness and flexibility. The current practice of physical testing is a very effective pass/fail technique for certification. However, the use of physical testing as a development tool is costly. The use of nonlinear finite element analysis offers a cost effective alternative to developmental physical testing. When many design iterations are anticipated, there can be a substantial savings realized by using finite element techniques.

2. ENGINEERING APPROACH

The Version 61B of NASTRAN includes geometrically nonlinear and materially nonlinear capabilities. NASTRAN, of course, is a well known, widely used, general purpose finite element program which was first developed in the 1960's primarily for application in the Apollo space project. The MacNeal-Schwendler Corporation has greatly enhanced the original capabilities of the program.

The finite element model used for this project employed on the order of 3,000 nodes and 3,000 shell elements, with approximately 16,000 degrees of freedom. This can be categorized as a large finite element model.

The data storage needs for the present case were larger than many nonlinear NASTRAN problems because of no substructuring (i.e., the superelement included all degrees of freedom), and the full set of nodal displacements was calculated at each point in the nonlinear iterative analysis.

Based on the observation of earlier hardware tests, it was assumed that no elastic buckling of the structure of the operator's compartment would occur. Buckling can be predicted and analyzed by means of this version of NASTRAN; however, the emphasis of this analysis is on predicting the yielding of material.

One other simplifying assumption has been that the deflections of the structure are of small rotations. Again, the version of NASTRAN used is capable of handling large rotations, but for the present problem it was not necessary.

3. ANALYSIS AND DISCUSSION

An Example Problem

The example problem included the analytical features pertinent to the subsequent analysis of the Universal Operator's Compartment. CQUAD4 is a NASTRAN shell finite element and was used in this example analysis and later was used in the compartment analysis. The load-displacement curve shown in Figure 1 closely follows the published results of the reference given. The example problem exercise had helped in the development of appropriate Job Control data, as well as NASTRAN Executive and Case Control formats which later helped expedite the full analysis of the Universal Operator's Compartment.

A Linear Analysis Step

The finite element model of the Universal Operator's Compartment was subjected to force and boundary conditions which represented the horizontal side load portion of the laboratory test. An arbitrary load value of 10,000 lbs. was applied and a linear NASTRAN analysis was performed in order to obtain predicted stress levels throughout the finite element model. From this it was extrapolated that first yielding of any material in the compartment structure would occur when a side force of 70,000 lbs. was applied.

The Iterative NASTRAN Analysis

NASTRAN approaches the nonlinear problem in piece-wise linear steps. That is, the boundary conditions, material properties, and changes in stiffness due to the deflection are calculated after each step, and the revised stiffness is used to extrapolate some further increment towards the solution. The selection of these increments and iteration points is left primarily to the engineer doing the finite element analysis. The objective is to obtain an accurate solution, which is certainly obtained with a large number of steps and increments, and yet at the same time to obtain the solution in a cost effective manner. Obviously, cost increases as the number of solution increments increases. In general, it can be said that when the structure is linear or nearly linear, large incremental steps, i.e., large changes in force may be specified. As the slope of the load-deflection curve becomes shallow it is necessary to make smaller changes and to take smaller increment steps. Of particular concern is the point at which the load reaches its maximum value. The finite element solution procedure requires very small load changes in order for a numerically stable solution to occur. In Figures 2 and 3 the load-deflection relationship of the analysis results is shown.

It was further found that, with the large number of degrees of freedom involved in this analysis, the numerical stability is more difficult to obtain than for smaller size problems. The slope of the stress-strain curve above yielding was originally allowed to be zero. This caused severe numerical problems, and it was subsequently recommended by MacNeal-Schwendler Corporation (the present source for NASTRAN) that an arbitrarily small positive slope be introduced. The value chosen for the present analysis was a slope of 3.0×10^6 psi for the material with a slope in its linear range of 30.0×10^6 psi. This approach can introduce errors because the actual material property may not be so well behaved, but it is thought that judicious use of this slope adjusting can allow a reasonable solution to be obtained. The extent of the differences can be seen in Figure 1. There are two curves presented, one for perfect yielding and one for an arbitrary stiffness above the yield point. There is no return curve to the zero force condition for the case with perfect yielding because the above mentioned numerical problems prevented that solution from being obtained.

The material stress-strain properties do need to be well defined for this type of analysis. They can make a significant difference in the deflection which occurs for a given force, as is shown in Figure 1. The main purpose of the present analysis was to develop the procedures and approach so that this method can be more routinely applied to structural designs before the hardware tests occur. It was felt that further investigation of items of this type might be better made in a more specific application.

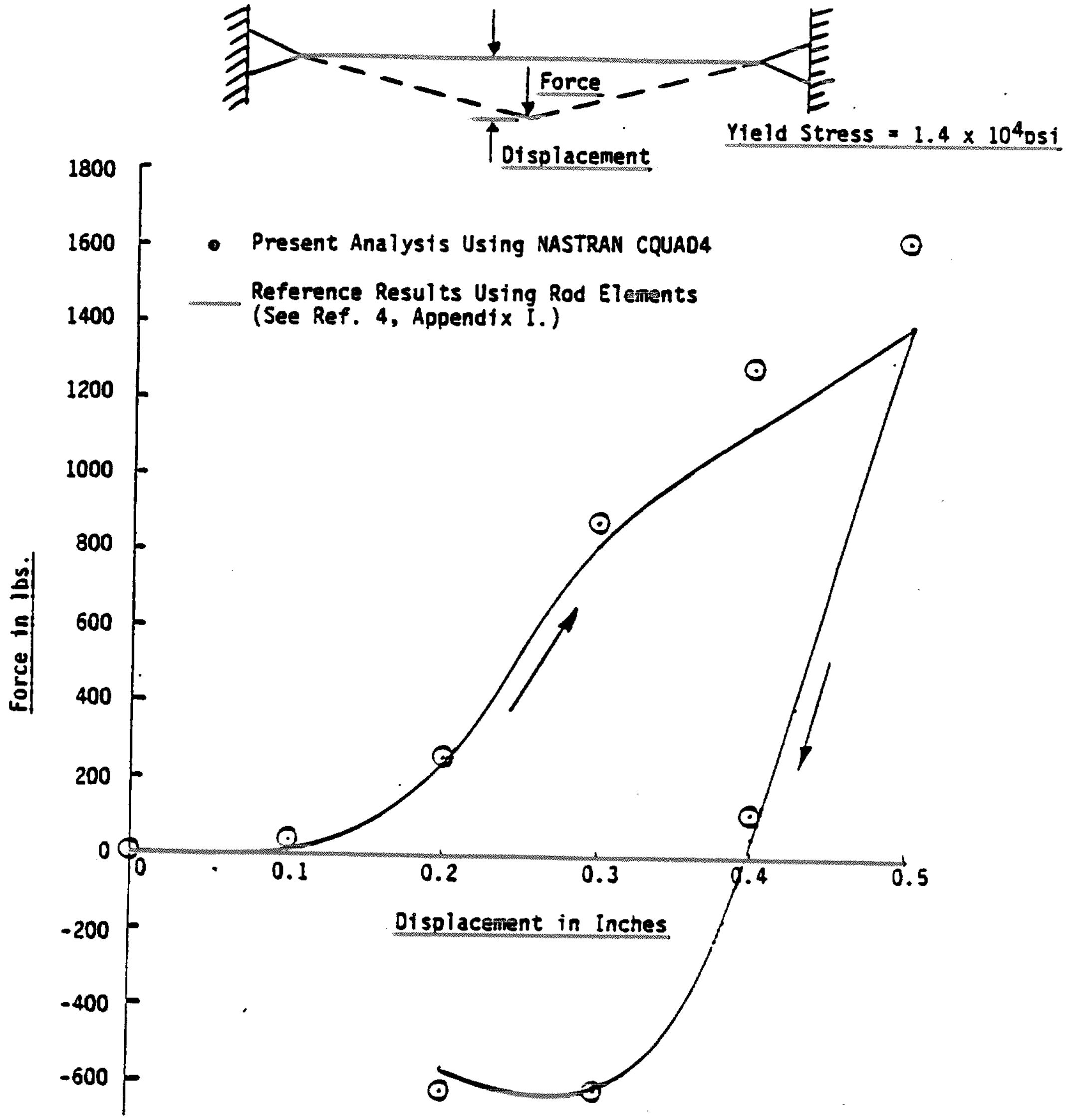


Figure 1. Example Problem Results Compared to a Published Solution

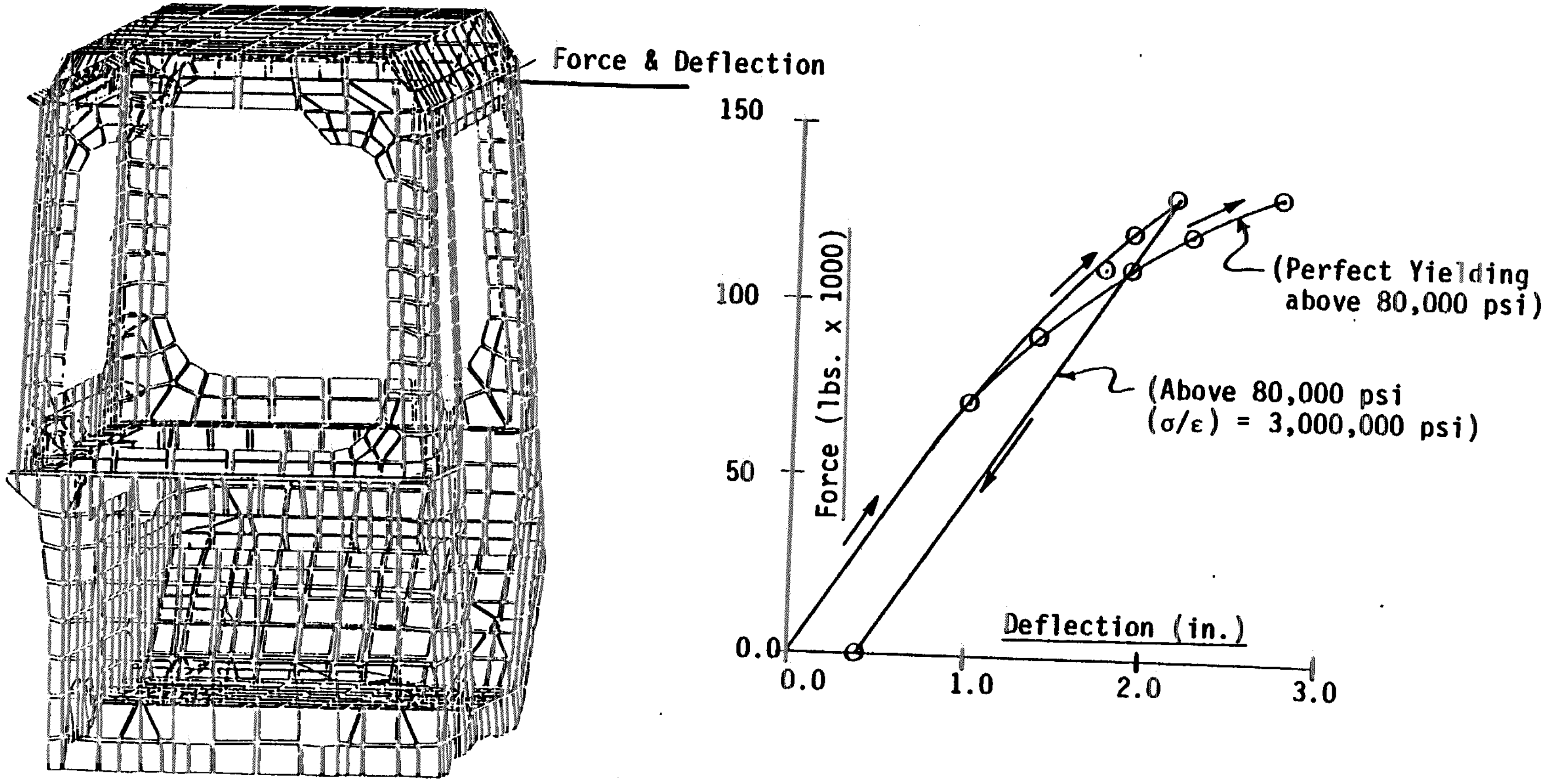


Figure 2. Horizontal Force-Deflection Analysis Results

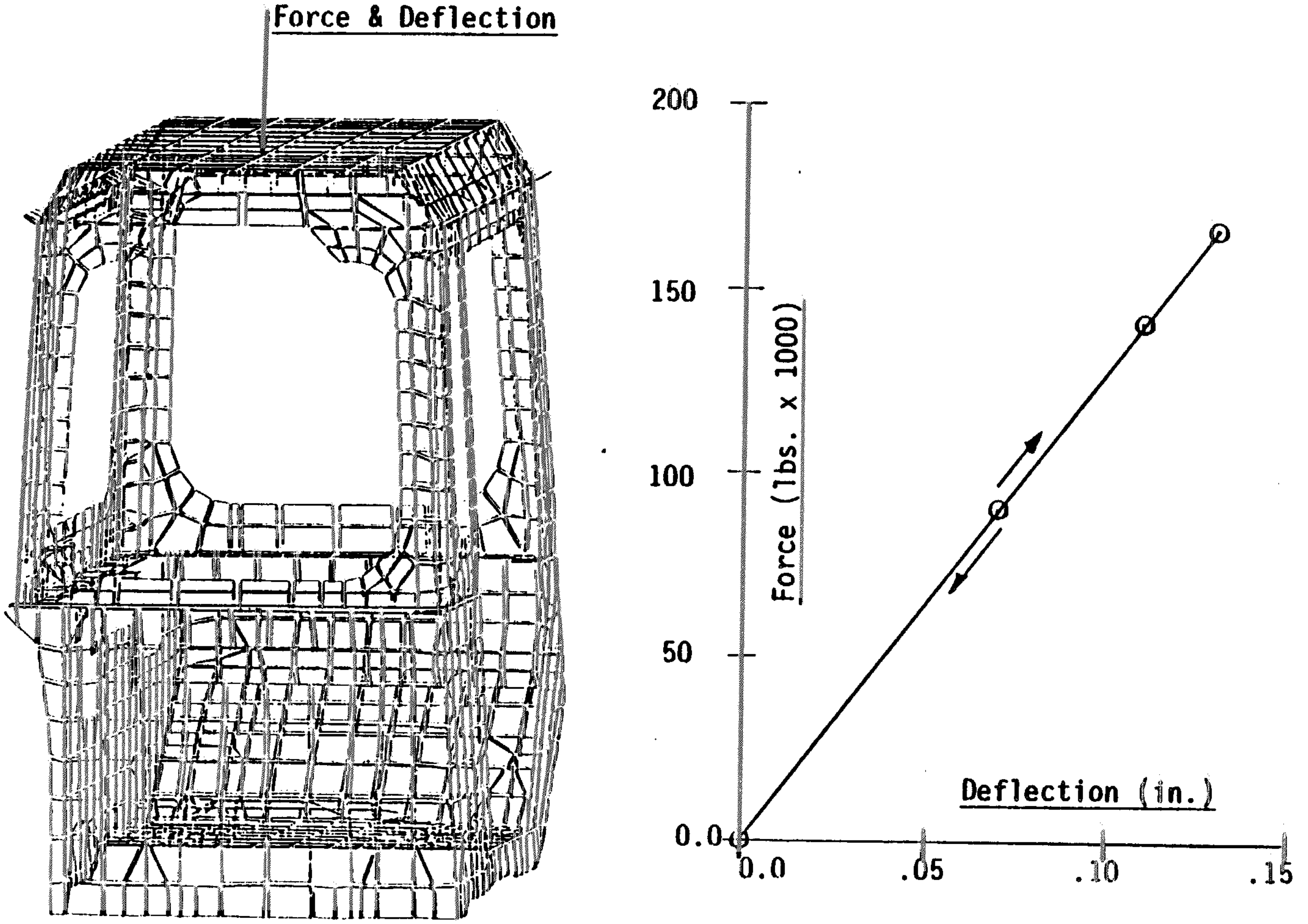


Figure 3. Vertical Force-Deflection Analysis Results

Direction	Subcase	Load-lbs.	Deflection-in.	
Horizontal	1	70 (x1000)	- .9839	Perfect Yielding Above 80,000 psi
	2A	90	-1.369	
	2B	110	-1.902	
	3	120	-2.258	
	4	130	-2.741	
Horizontal	1	70 (x1000)	- .9839	Above 80,000 psi (σ/ε) = 3,000,000 psi
	2	110	-1.682	
	3	120	-1.899	
	4	130	-2.141	
	5	129.870	-2.139	
	6	110	-1.872	
	7	-0-	- .395	
Vertical	11	90,026	- .06928	
	12	141,469	- .10959	
	13	154,330	- .11974	
	14	167,191	- .12990	
	15	167,024	- .12978	
	16	(No Printout Requested)		
	17	141,469	- .1099	
	18	-0-	~0	

Table 1. Tabulated Loads & Deflections

4. RESULTS

The deflections under the various load conditions are shown in Figure 4. The line drawings are connections of selected points for which the displacements are being shown, and the Universal Operator's Compartment is being viewed from the left front corner of the vehicle. The black lines show the compartment shape before forces are applied. The red lines show the distortion under a maximum horizontal side force of 130,000 lbs. Upon being unloaded the compartment returned to the position indicated by the blue lines. And, of course, the difference between the blue and the black shapes is a result of yielding within the compartment. Then a maximum vertical force was applied and that deflected shape is shown in green. Some yielding did occur for the vertical load condition, but the resulting permanent deformation was not visible at the scale of this figure. The blue frame is a good representation of the unloaded compartment at the completion of the test.

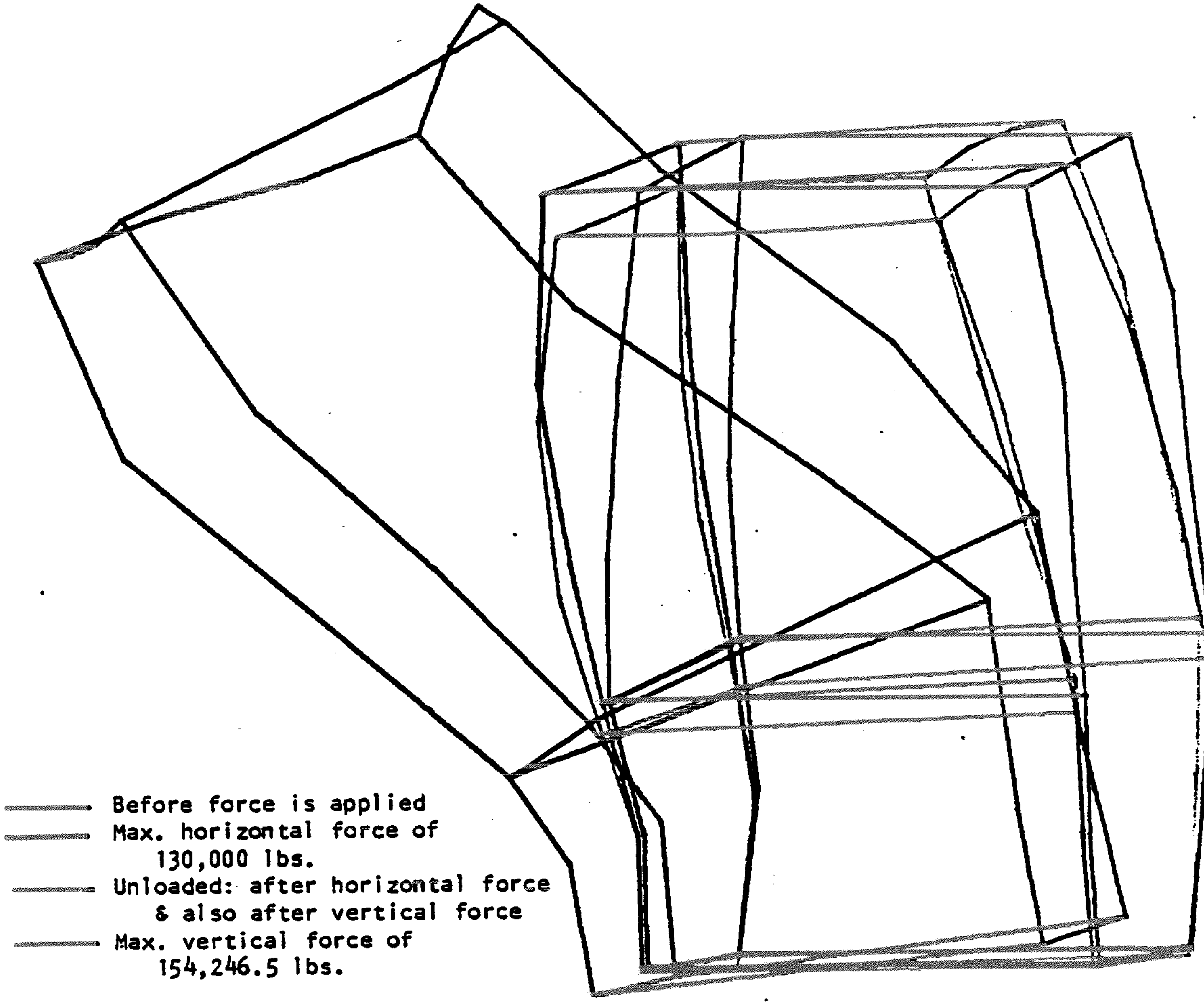
The stresses displayed in Figures 5 and 6 are Von Mises equivalent stresses for either the top or bottom surface, whichever had the greatest equivalent stress value.

As mentioned earlier, this finite element analysis is of the compartment alone. The laboratory tests were for the compartment and the vehicle frame together. In fact, the laboratory tests showed most of the permanent deflection to occur in the frame for this Universal Operator's Compartment. Therefore, this finite element analysis is predicting fairly small permanent deformations.

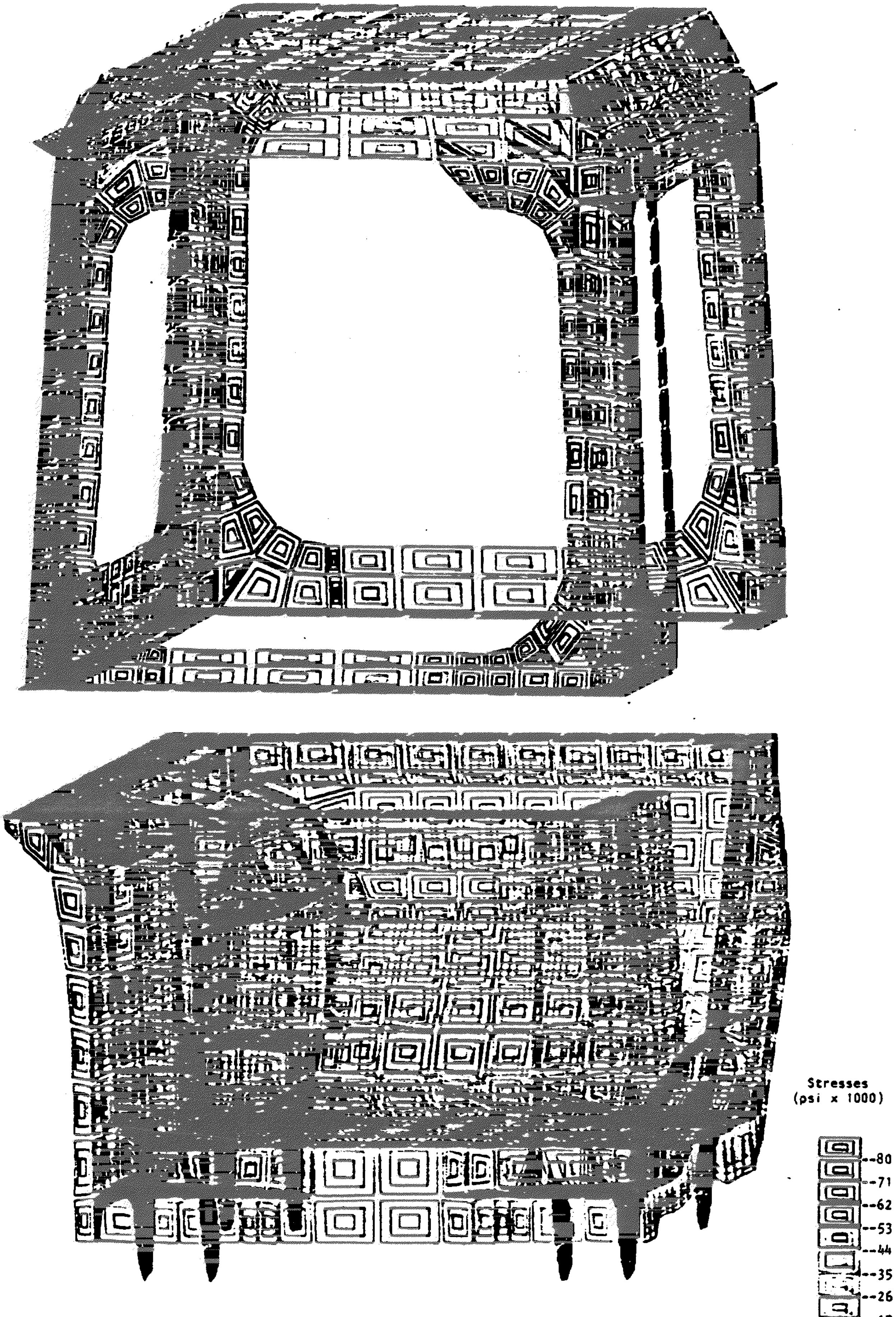
Figure 4. Deflections at Various Load Conditions
(Plotting Software by Engineering Methods, Inc.)

Figure 5. Stresses for the Maximum Horizontal Load
(Plotting Software by Engineering Methods, Inc.)

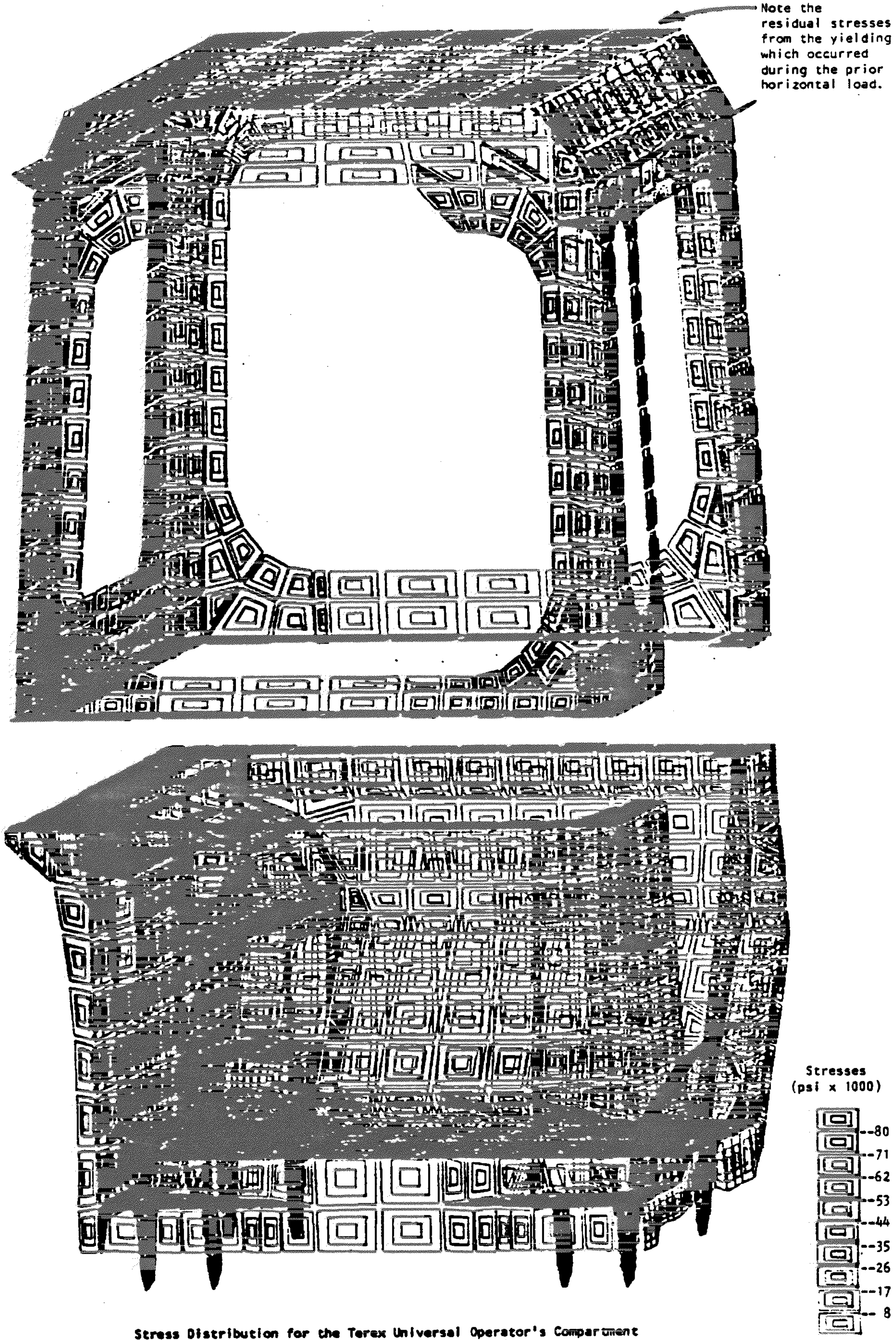
Figure 6. Stresses for the Maximum Vertical Load;
Includes Residual Stresses from the Horizontal Load
(Plotting Software by Engineering Methods, Inc.)



The Exagerated Deflections
for the Terex Universal Operator's Compartment
at the Various Load Cases



Stress Distribution for the Terex Universal Operator's Compartment at the Maximum Horizontal Load of 130,000 lbs.



Note the residual stresses from the yielding which occurred during the prior horizontal load.

Stress Distribution for the Terex Universal Operator's Compartment at the Maximum Vertical Load of 154,246.5 lbs.

5. CONCLUSIONS AND RECOMMENDATIONS

The laboratory test results were obtained for the Universal Operator's Compartment and the vehicle frame together. The present analysis results were for the Universal Operator's Compartment alone. Therefore, the deflections obtained were different, and laboratory measurements of more detail will be needed to compare these analysis results to lab results.

The cost of the present project compares very favorably with the cost of fabricating and testing hardware. This is especially true if one looks at the long term costs of maintaining a certain quantity of test facilities in the future five to ten year time frame. The total number of tests being performed could be minimized if the present finite element approach was implemented on a widespread basis.

Analysis procedures of this type can reduce the number of hardware test and refinement cycles required for the development of a production design. In a five to ten year time period that can reduce or minimize the demand for hardware test equipment, space, and other associated costs. It is not recommended that testing be eliminated, of course, but the number of tests performed in order to reach a good design can be very significantly reduced.

1. 1983 SAE Handbook, Volume 4, "On-Highway Vehicles and Off-Highway Machinery," pp. 40.235 - 40.243.

2. MSC/NASTRAN, Application Manual, Vol. 1, CDC Edition, MacNeal-Schwendler Corporation, Los Angeles, California, May/June, 1981.

3. MSC/NASTRAN User's Manual, MSR-39, MacNeal-Schwendler Corporation, Los Angeles, California, May, 1976, Revised May, 1979.

4. Nonlinear Statics in MSC/NASTRAN, SA109, A. O. Smith Corporation, Milwaukee, Wisconsin, September 23-25, 1981.

5. Nonlinear Statics in MSC/NASTRAN, SA109, SOL 64,66 References, A. O. Smith Corporation, Milwaukee, Wisconsin.

6. Finite Element Analysis Fundamentals, Richard H. Gallagher, Dept. of Structural Engineering, Cornell University, Prentice-Hall, 1975.