Shake Test Simulation Using MSC/NASTRAN

Randal H. Visintainer
Ford Motor Company
Dearborn, Michigan

Farhang Aslani
Automated Analysis Corporation
Ann Arbor, Michigan

Abstract

Advances in Computer-Aided Testing technology have led to the simulation of service operation and test track loads in the laboratory. The controlled environment of the lab creates an ideal situation for attempts to simulate the lab test. Furthermore, the concept of up-front engineering using analytical prototype has pushed the demands on the CAE analysts to develop methodology to simulate operating conditions, durability events, and laboratory tests. To address this need, the frequency response solution (SOL 111) of MSC/NASTRAN is used to simulate the vehicle shake test of a heavy truck. The displacement-controlled lab test was performed between the frequency range of 20 to 50 Hertz (Hz). The MSC/NASTRAN simulation of the test and the correlation between the test and analysis results revealed several important points that should be considered in a computer simulation using the analytical prototype.

Introduction

This study was conducted to simulate the events of a lab test that a heavy truck cab is subjected to validate a new design. The primary objective of this study was to establish the methodology by which a cab shake test can be simulated through finite element analysis (FEA) using MSC/NASTRAN. The input for the shake test consisted of a frequency sweep from 20 to 50 Hz with an enforced displacement amplitude of 0.02 inch. Strains in the cab floor were measured from strain gages with a rosette configuration. Modal frequency response (SOL111) of MSC/NASTRAN was used to simulate the shake test.

Finite Element Model

Figure 1 shows the FE model of the truck system used in the computer simulation. The FE model consisted of two tip superelements and a residual. The first tip superelement was a plate model of the cab. The model included the cab sheet metal, doors, windshield, rear window, and the side glass. Rigid elements (RBE2) were used to connect these components to the cab sheet metal.

The second tip superelement represented the vehicle model (Figure 1) including the frame, driveline, payload, fuel tank, battery, axles, as well as the front and rear suspension. The cab isolators and the loading points on the leaf spring were included in the residual superelement. This arrangement allowed for relatively inexpensive re-analysis with various inputs to the truck as well as different cab mount stiffness and damping rate.

MSC/NASTRAN Features

The superelement capability of MSC/NASTRAN with split data base was used. We also used the data base split feature for upstream and downstream data of tip superelements. These features allowed us to efficiently use the disk space and CPU time of the CRAY C90 used to perform the analyses. For example, the several residual runs that were submitted did not have to bring all the data bases on line. This saved us the time that we would have had to spend waiting in the queue for a large database with all the unnecessary upstream data blocks to be put on line. This was particularly helpful since we had to perform several runs for pre-test planning as will be discussed later.

Structured Solution Sequence 103 (SOL103) was used to extract and save the component modes using the Lanczos method with sparse solver. To increase the accuracy of the system modes, the residual structure modes were extracted with the Modified Givens (MGIV) method without dynamic reduction. Modal frequency Solution Sequence 111 (SOL111) was used as a restart to SOL103 to compute the frequency response to the enforced displacement. The frequency response analysis was performed on the residual structure. A restart for data recovery for the tip superelement of the cab was performed to recover the stress and strain under the enforced diplacement at the center of the front suspensions.

Test Set-Up

Figure 2 shows the typical test set-up used for testing the truck used in this study. A test fixture shown in Figure 3 connects the center of the front leaf springs to the hydraulic actuators. For the purpose of the FEA correlation study, hydraulic actuators applied a vertical steady state displacement to their two connection points to the test fixture, which was assumed to be sufficiently rigid to transfer the displacement to the test vehicle without any deflection in the test fixture.

The displacement-controlled test was performed in the frequency range of 0-50 Hz. Figure 4.a shows the enforced displacement recorded during the test. The intent of the test was to enforce a 0.02 inch displacement at the center of the front leaf springs. However, due to a resonance condition at 39 and 44 Hz, the enforced displacement was higher than 0.02 inch. The resonance condition will be discussed in later sections. Figure 4.b shows the displacement used in the computer simulation to duplicate the enforced displacement recorded from the test.

Pre-Test Planning

Prior to the test, finite element analysis was conducted for pre-test planning purposes. Unit diplacement was enforced at the center of the front leaf springs in a frequency response analysis that covered the 20 to 50 Hz range. Dynamic post-processing software developed at Automated Analysis Corporation (AAC) and Ford Heavy Truck (FHT) was used to perform the following :

1. To identify the element and frequencies at which maximum stresses occur. A typical output of the DYNASORT software is presented in Figure 5.a.
2. To generate PATRAN/PPLOT input file to get X-Y plots of stress versus frequency for particular elements identified to have high stresses.
3. To generate PATRAN stress files to get stress contour plots of all elements of a panel for the frequencies at which maximum stress occurred. A typical stress contour plot for floor panel is shown in Figure 5.b.

The stress contour plots of the floor, back panel, and roof were used to identify high stress locations and come up with maps similar to that shown in Figure 6. The maps were provided to the test engineer for strain gage installation.

Results

Stress Correlation

Figure 7 shows the equivalent Von Mises stress recorded by strain gauge Number 9 at the tip of the floor panel shown in Figure 6. The stress response has peaks at several frequencies including 27, 29.5, 33.5, 42, 43.5, and 45.5 Hz. Figure 8 shows the same stress response predicted by the computer model for 3 different modal damping ratios between 0% to 5%. The computer simulation indicates peak stress response at 27, 30, 36, 43.5, and 48.5 Hz. Two conclusions can be reached by comparison of the stresses from the test and computer simulation :

1. The amplitude response from the computer simulations is very sensitive to the amount of damping in the model. Figure 8 shows that 2.5% damping results in a 42% drop in the peak stress amplitude at 27 Hz.
2. Some of the peak stresses recorded in the test were not predicted by the computer simulation. This can be attributed to two problems.
   - The test set-up including the test fixture has some flexibility which contributes to the peak responses of the vehicle. However, the computer model did not include the test fixture.
   - The cab FE model needs to be modified to correlate with simple tests such as bending and resonance frequency analyses. Such correlation effort was not undertaken in this study since it was not the primary objective of the study.

<u>Test Fixture Flexibility</u>

In a vehicle shake test, the test stand is designed not to have flexible deformation under the test condition. Therefore, the test fixture was not included in the computer model under the assumption that it was rigid. Since some stress peaks recorded during the test were not predicted by the computer model, it was thought that the rigidity assumption for the test fixture might have been invalid.

Therefore, we investigated the test fixture flexibility by computing its resonance frequencies in the range of 0 to 50 Hertz (Hz). SOL 103 of MSC/NASTRAN was used to identify the resonance frequencies of the test fixture under the test condition. The results indicate that the test fixture under the truck mass has two resonance frequencies of 16 and 40 Hz (Figure 9). The effects of these resonances are discussed in the Recommendation section.

Conclusions and Recommendations

- The test fixture used in the test set-up of the truck had 2 resonance frequencies below 50 Hertz (Hz). A finite element pre-test study should be conducted to determine resonance frequencies of the test set-up for two reasons:

  1. The Remote Parameter Control (RPC) software from MTS does not compensate for the resonance modes of the test set-up that have resonance frequency above the control frequency of the RPC software. This results in the amplification of the test vehicle response under the following condition :

     RPC control frequency < Test Set-Up Resonance Frequency < Maximum Excitation Frequency

  2. The FE correlation study, which does not include the test set-up, will not correctly predict the vehicle response if the test set-up resonance mode is excited during the test.

- Dynamic test simulation should concentrate on acceleration response and not on stress response. Acceleration is a direct physical measurement whose accuracy is influenced by a smaller number of factors compared to stresses which can not be measured directly.

Future Work

The primary objective of this study was to establish a methodology by which cab shake test can be simulated through finite element analysis using MSC/NASTRAN. In a cab shake test, the input to the vehicle is the response of the vehicle that is collected during vehicle operation on specific routes representing the operational conditions that the vehicle will experience during its life [1]. The lessons learned in this study will be used in an ongoing project in which random vibration analysis capabilities of MSC/NASTRAN in SOL 111 [2] will be used to predict the Power Spectral Density (PSD) of the stress of different panels.

The number of zero-crossings from the analysis will be used to statistically calculate the fatigue life of the cab structure [3].

Acknowledgment

The test described here was coordinated by the Ford Heavy Truck Development Department. The coordination effort by Richard J. Bond is especially appreciated. The support and encouragement from Ford Heavy Truck management was also crucial to the initiation and completion of this study.

Reference

1. Fash, J.W., Goode, J.G. , Brown. R. C., "Advanced Simulation Testing Capabilities", SAE No. 921066, 1989, Proceedings of the Eighth International Conference on Vehicle Structural Mechanics and CAE.
2. MSC/NASTRAN Dynamic Analysis Handbook.
3. S. H. Crandall and W.D. Mark, "Random Vibration in Mechanical Systems", Academic Press, 1963.

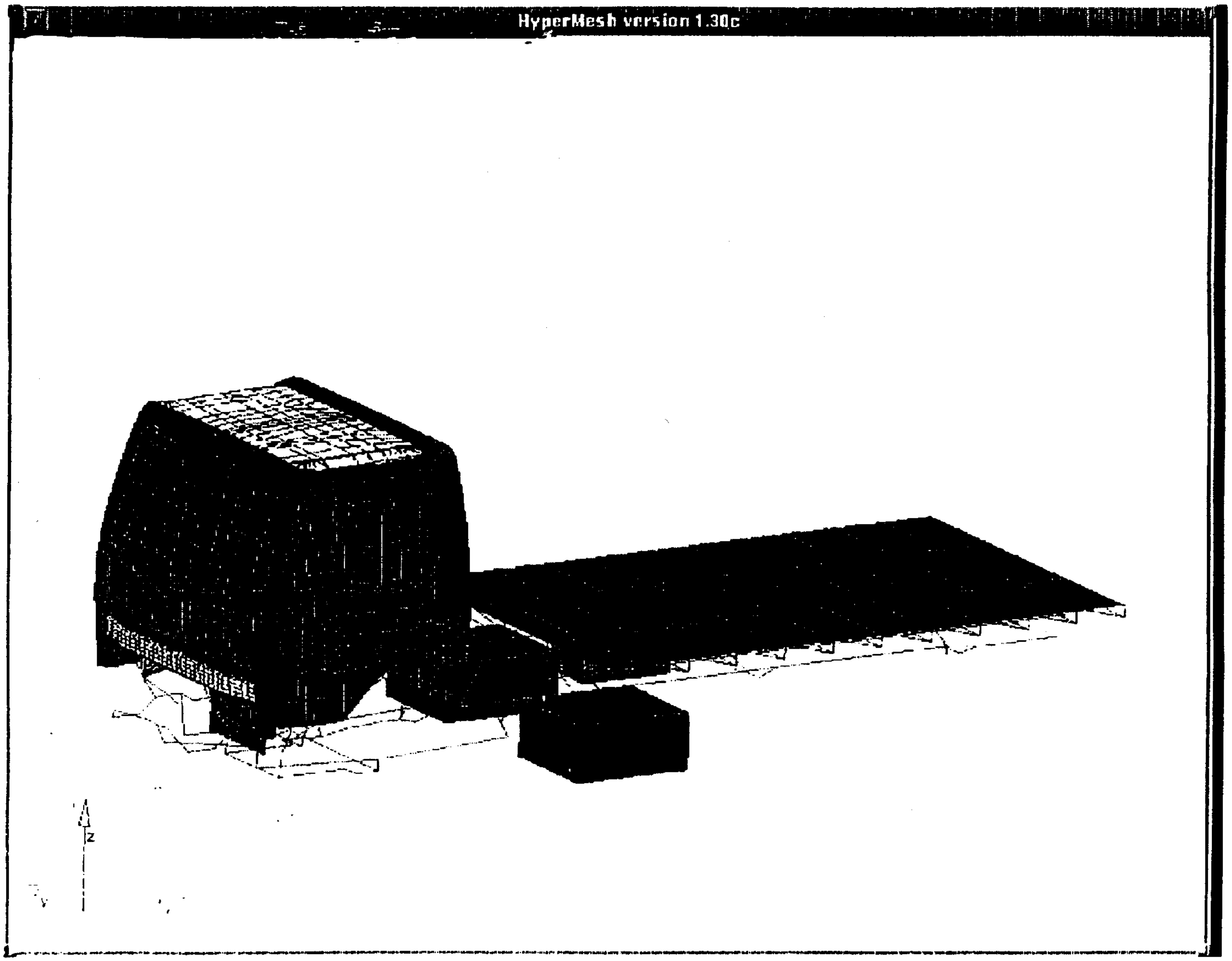

Figure 1 - Finite Element Model of the Truck System

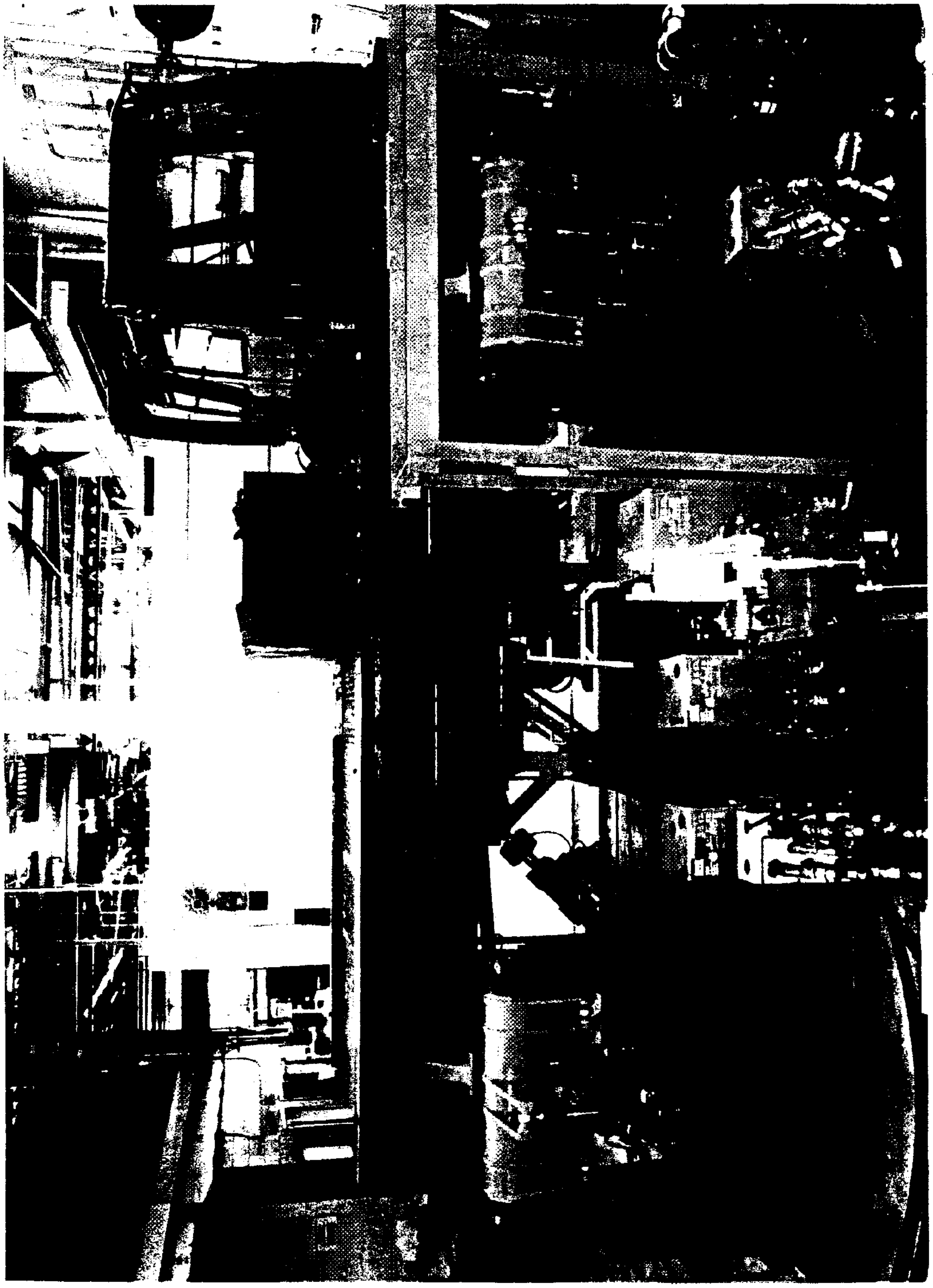

Figure 2 - Typical Test Set-Up for Testing a Truck

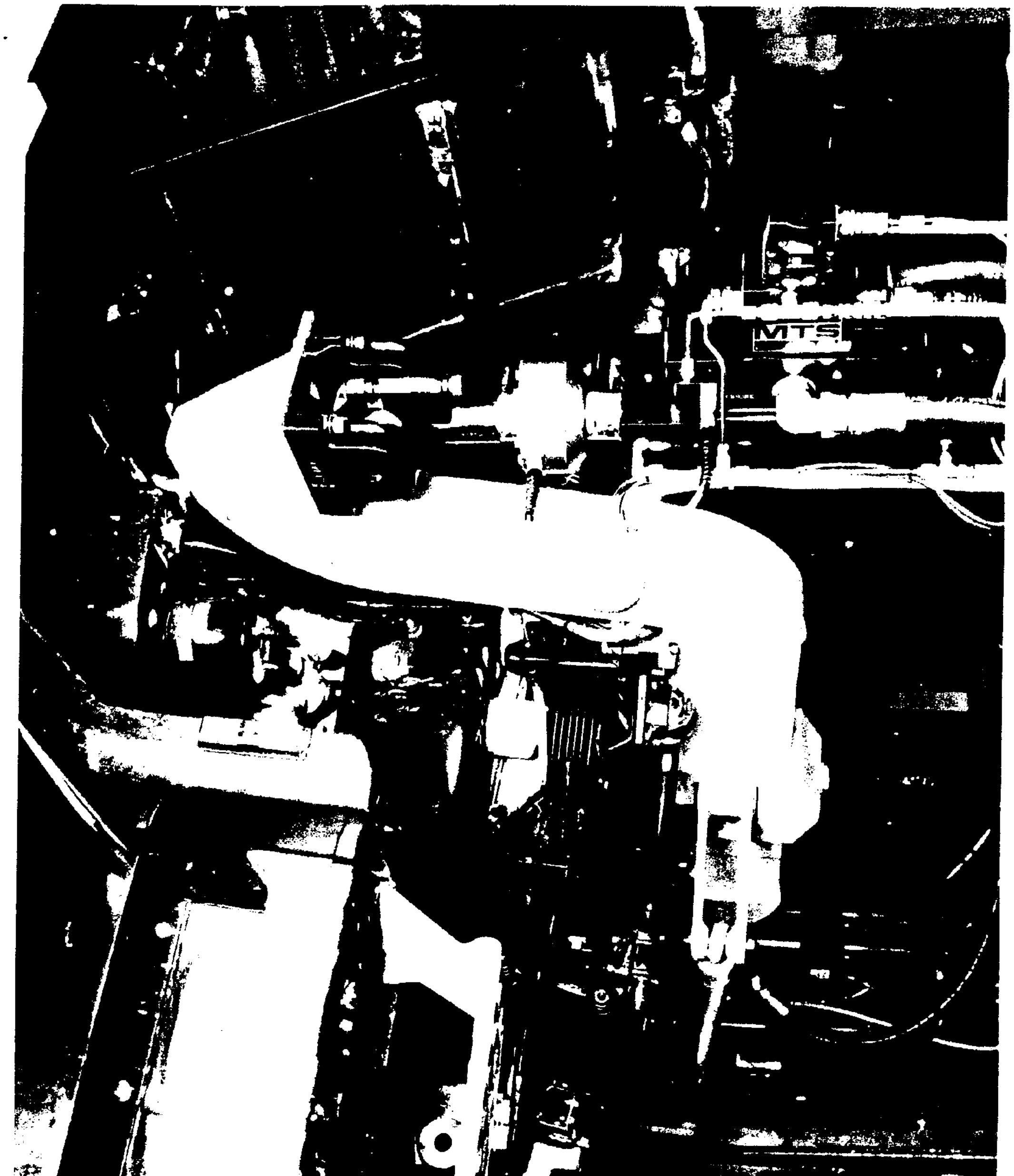

Figure 3 - Connection of the Truck to Test Fixture

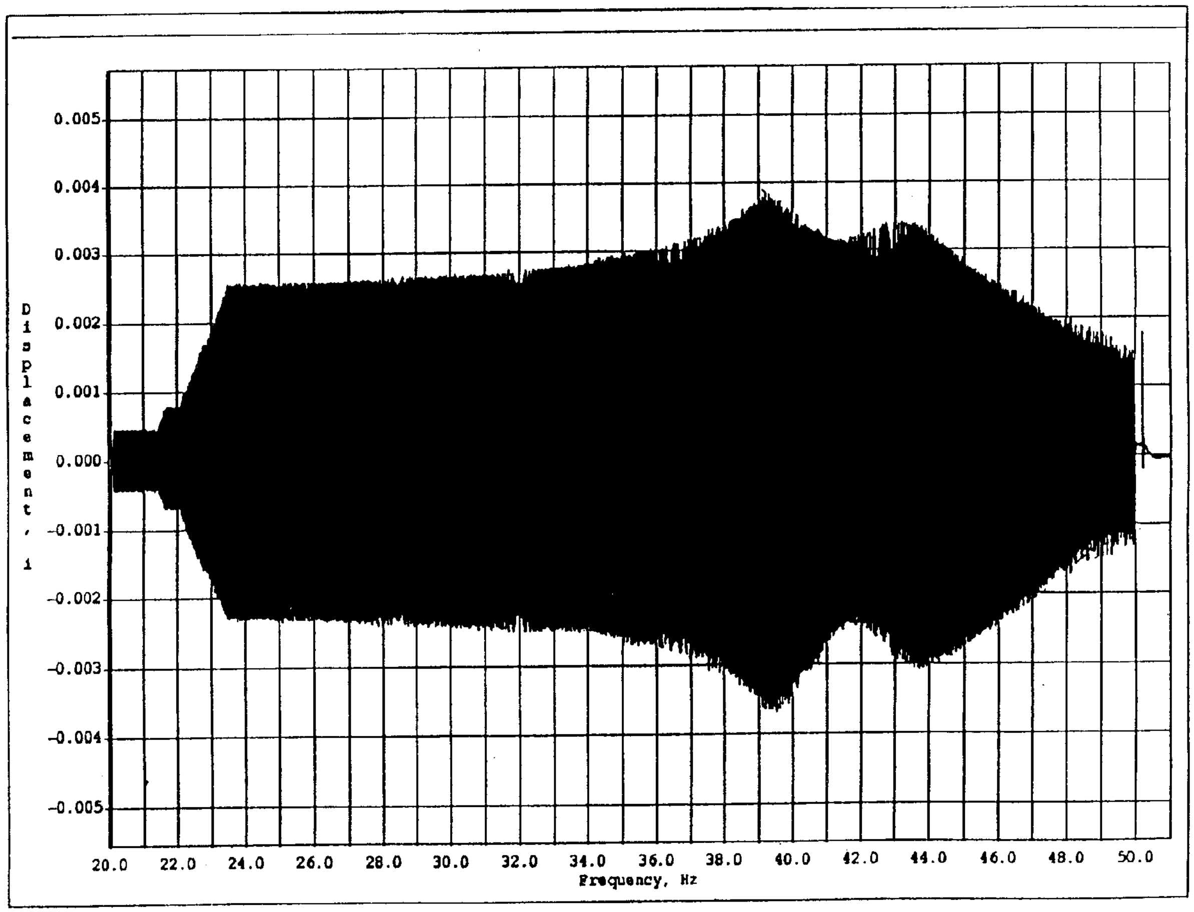

Figure 4.a - Enforced Displacement Recorded During the Test

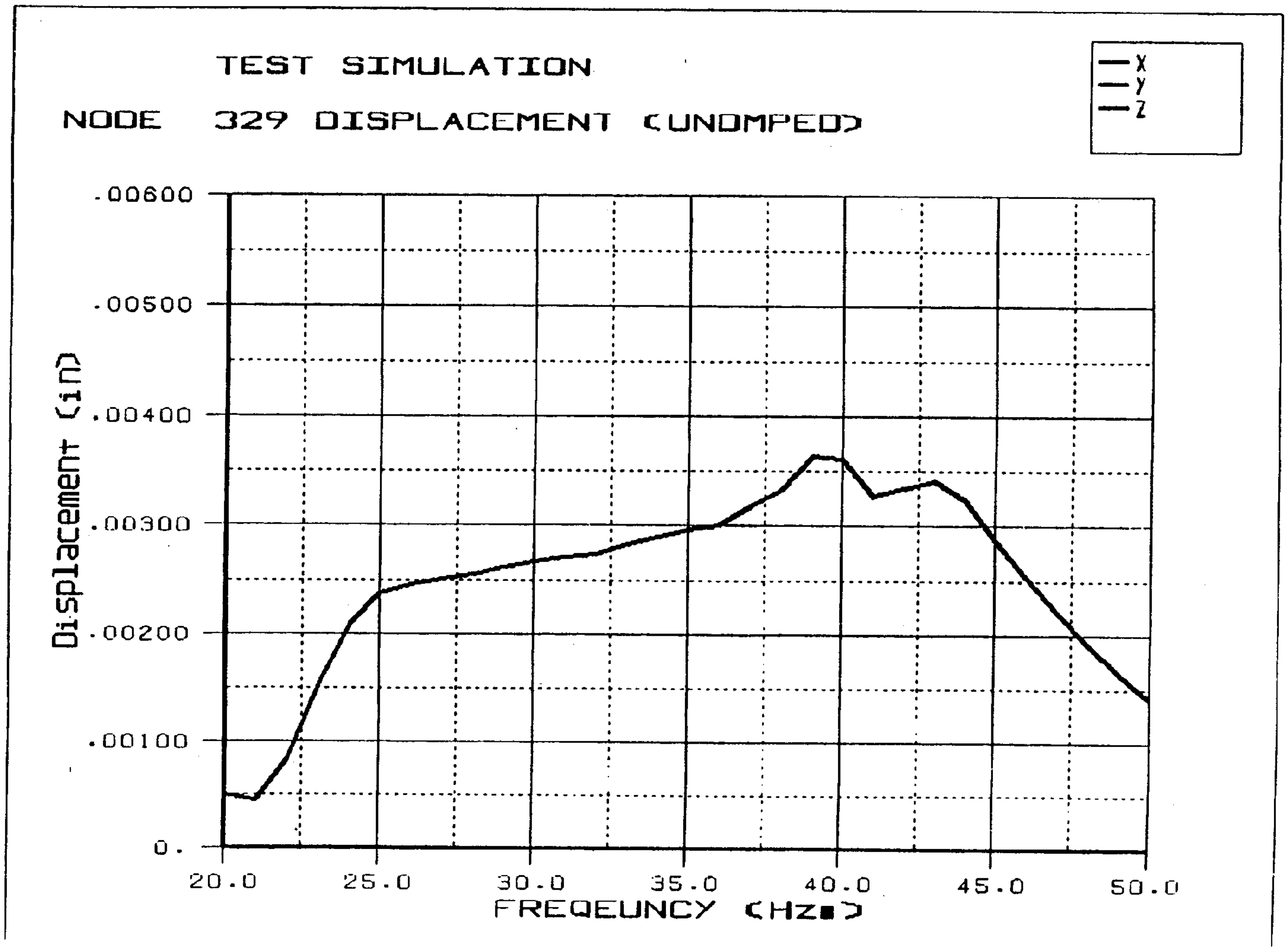

Figure 4.b - Enforced Displacement Used in Computer Simulation

AUTOMATED ANALYSIS CORPORATION (A A C)
OUTPUT FORM PROGRAM D Y N A S O R T

JOB TITLE : UNDER IN-PHASE FRONT TIRE UNIT ACCEL
DYNAMIC ANALYSIS TYPE : FREQUENCY RESPONSE
SORTED RESPONSE : STRESS
SORTED COMPONENT : VON MISES
SCALE FACTOR : 1.0000000000

ELEMENT	MAX STRESS	FREQ
43637	0.220439738E+01	0.183400000E+02
43637	0.836019515E+00	0.303800000E+02
43637	0.677368964E+00	0.274000000E+02
43637	0.676485220E+00	0.280000000E+02
43637	0.563982982E+00	0.254000000E+02
41292	0.516244071E+00	0.183400000E+02
41402	0.510633633E+00	0.183400000E+02
41293	0.468148393E+00	0.183400000E+02
43636	0.458208108E+00	0.183400000E+02
43634	0.416103892E+00	0.183400000E+02
41402	0.395129219E+00	0.280000000E+02
41402	0.391792764E+00	0.274000000E+02
43632	0.373734375E+00	0.183400000E+02
41402	0.362489807E+00	0.303800000E+02
43515	0.342861348E+00	0.183400000E+02
41289	0.326910756E+00	0.183400000E+02
46295	0.319675801E+00	0.183400000E+02
43371	0.309281232E+00	0.183400000E+02
41402	0.298362763E+00	0.254000000E+02
41291	0.294904731E+00	0.183400000E+02
50069	0.292737208E+00	0.183400000E+02
41204	0.279920865E+00	0.183400000E+02
50739	0.277522403E+00	0.183400000E+02
45530	0.271848982E+00	0.183400000E+02
41398	0.264509581E+00	0.183400000E+02
45636	0.256047400E+00	0.183400000E+02
50258	0.253988159E+00	0.183400000E+02
50751	0.253196513E+00	0.183400000E+02
43633	0.242904349E+00	0.183400000E+02
41382	0.242683398E+00	0.183400000E+02
41223	0.242395961E+00	0.183400000E+02
41671	0.241443136E+00	0.274000000E+02
41202	0.237406791E+00	0.183400000E+02
41205	0.236813203E+00	0.183400000E+02
50752	0.233903778E+00	0.183400000E+02
51130	0.232905800E+00	0.183400000E+02
40865	0.230830711E+00	0.183400000E+02
40867	0.230167669E+00	0.183400000E+02
41671	0.228842691E+00	0.280000000E+02
41292	0.225315272E+00	0.280000000E+02
41200	0.222312779E+00	0.183400000E+02

Figure 5.a - A typical Output for Sorted Stresses from DYNASORT Program

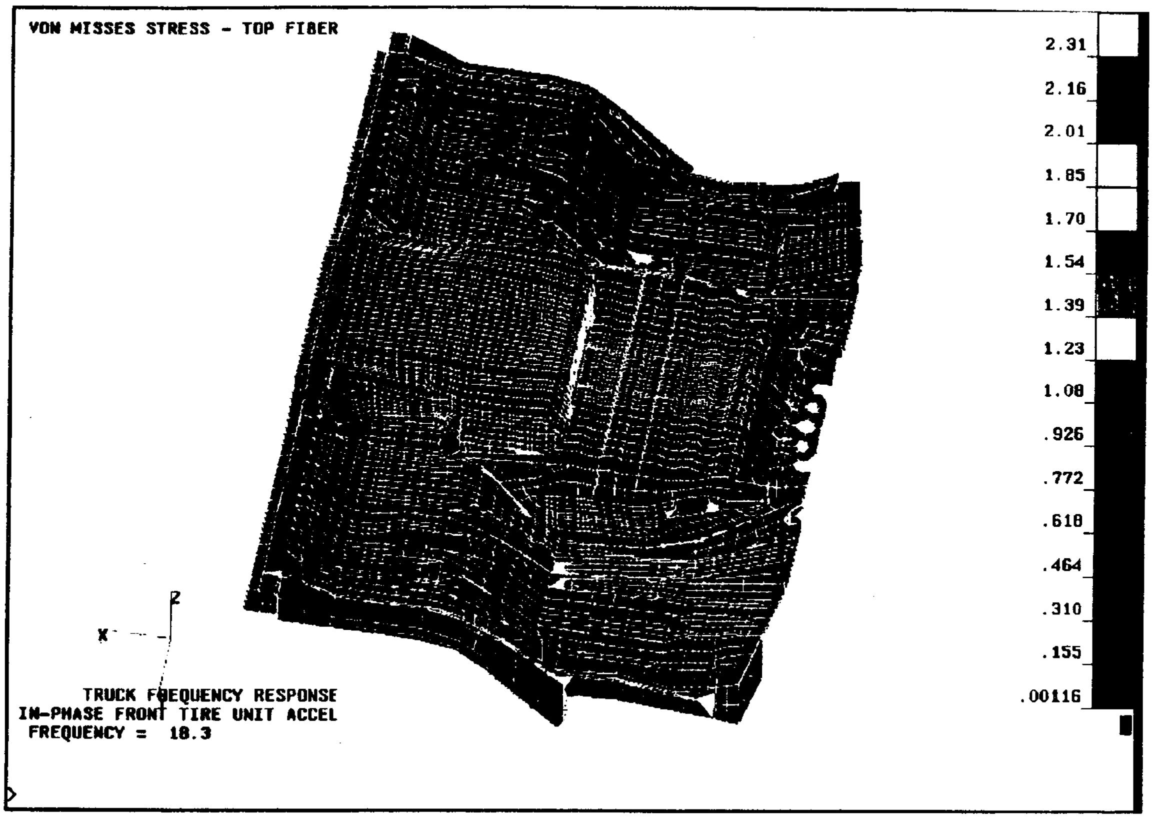

Figure 5.b - A typical Stress Contour for Cab Floor

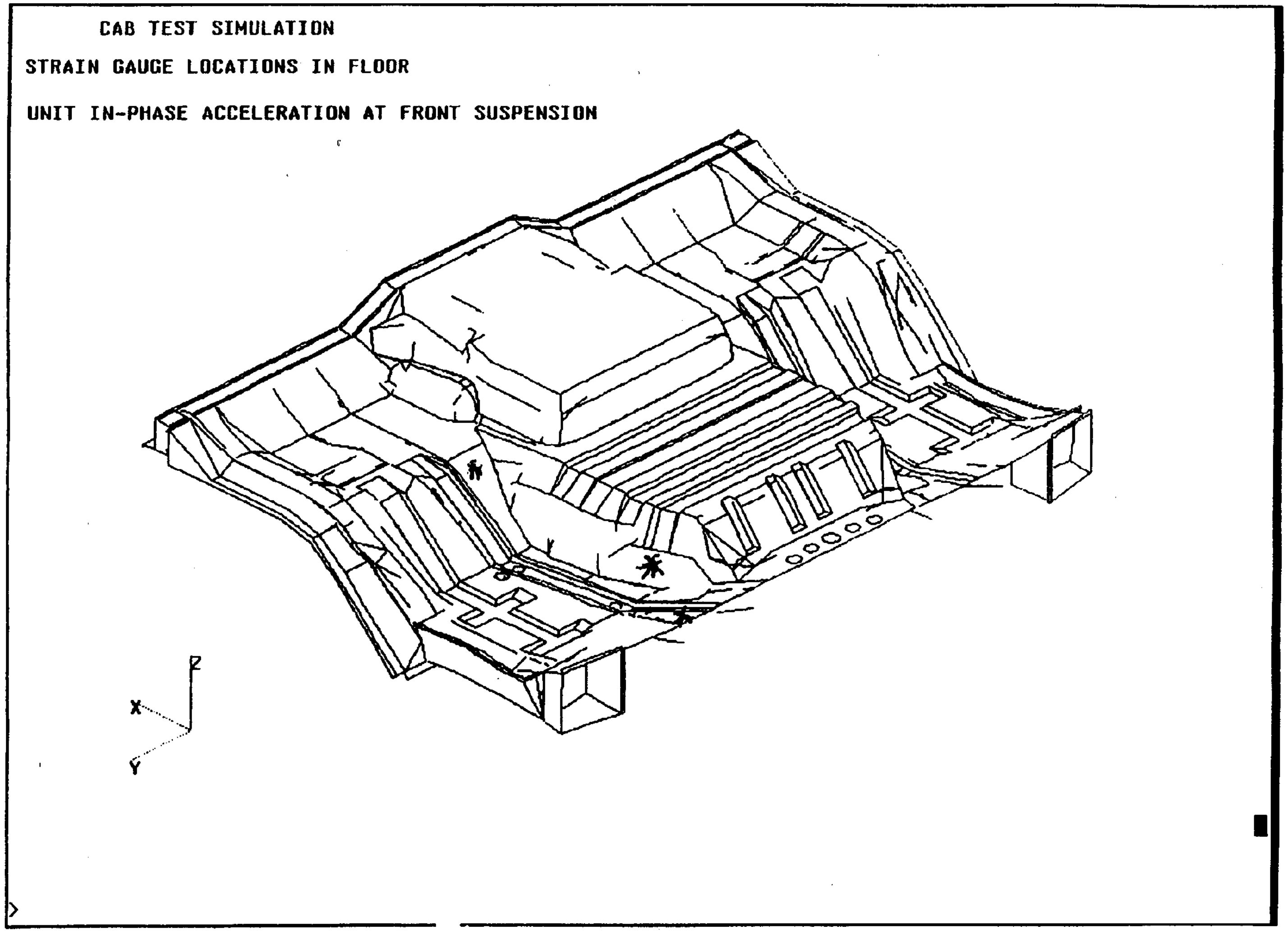

Figure 6 - Strain Gauge Locations in Cab Floor

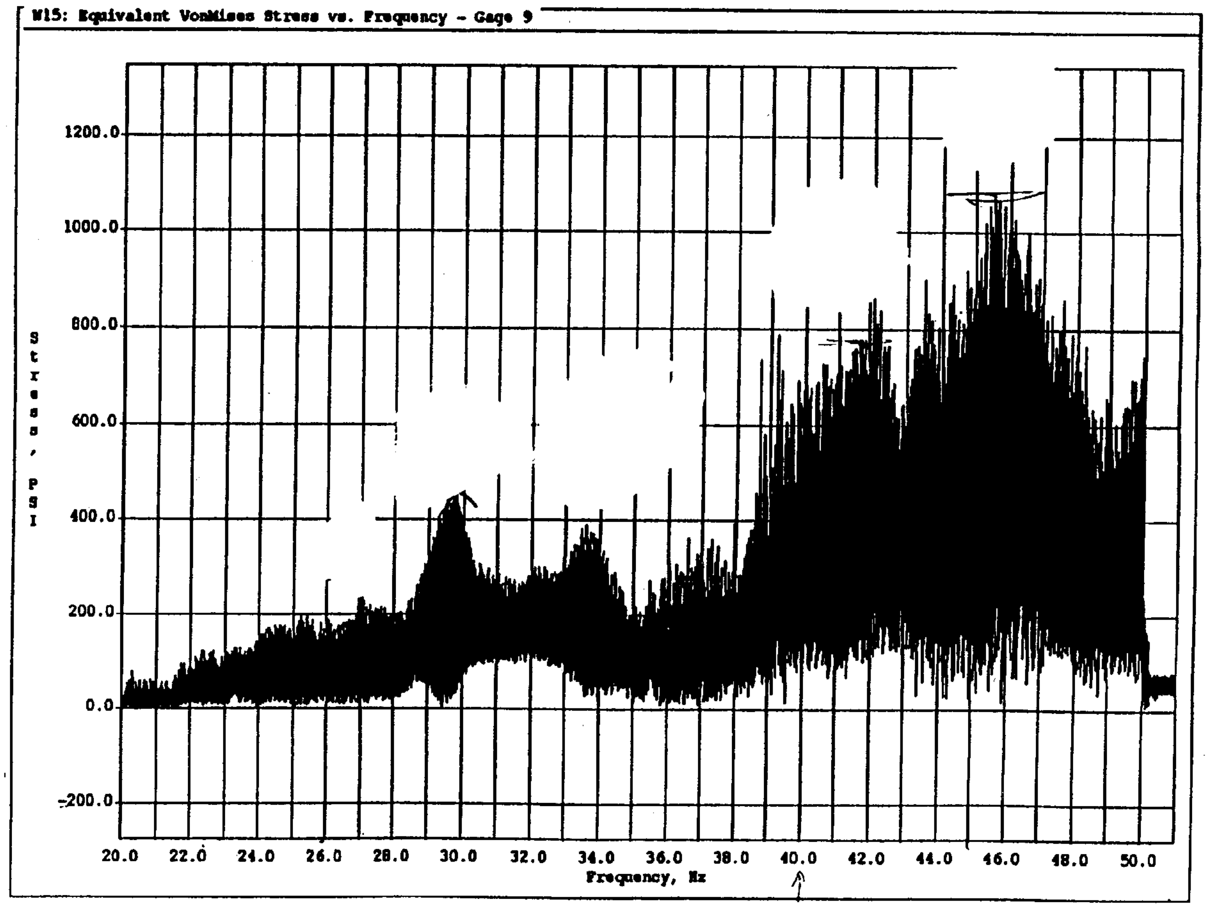

Figure 7 - Equivalent Von Mises Stress at the Location of Strain Gauge Number 9

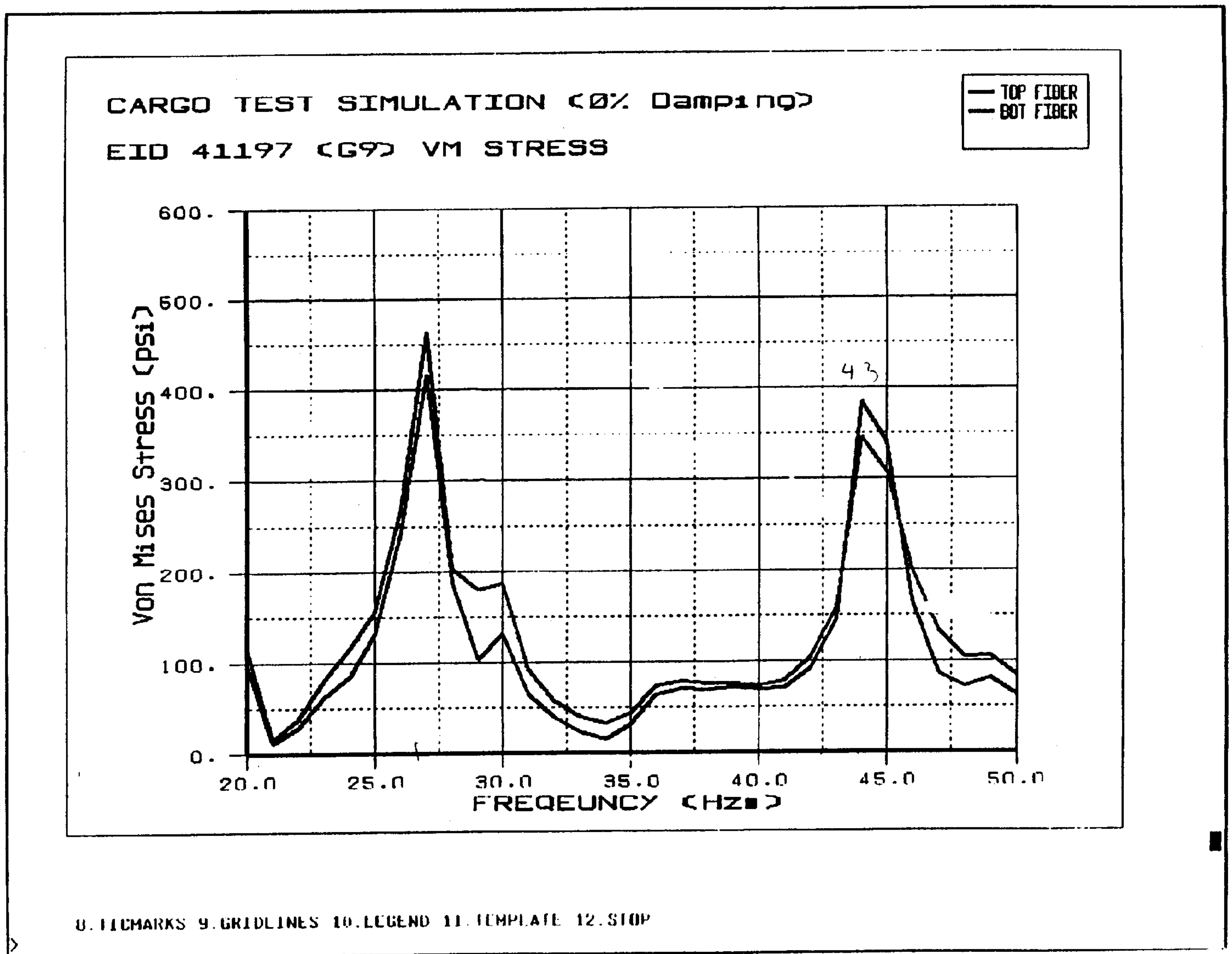

Figure 8a - Computer Prediction of the Equivalent Von Mises Stress at the Tip of the Floor Panel with 0% Modal Damping (Location of Strain Gauge 9)

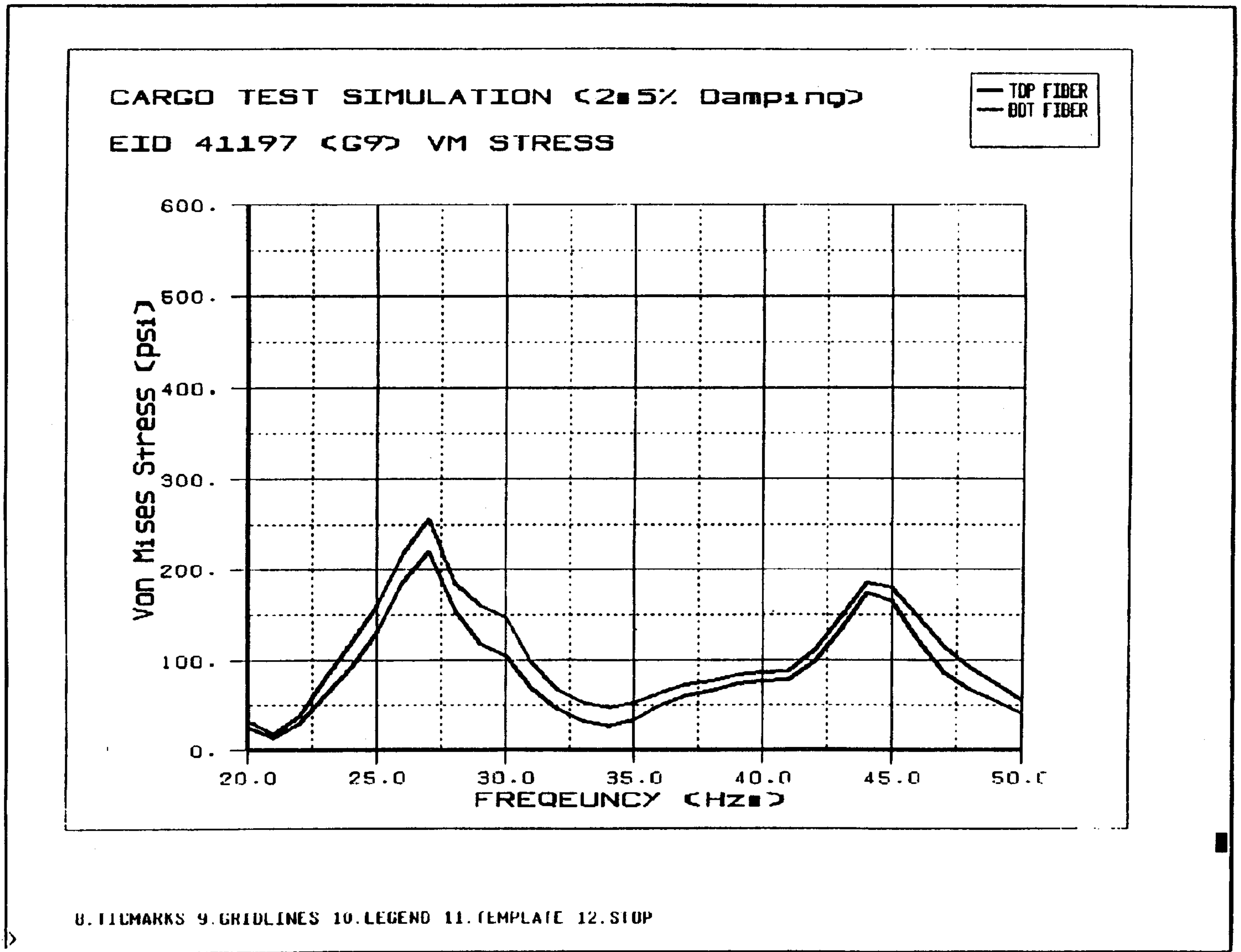

Figure 8b - Computer Prediction of the Equivalent Von Mises Stress at the Tip of the Floor Panel with 2.5% Modal Damping (Location of Strain Gauge 9)

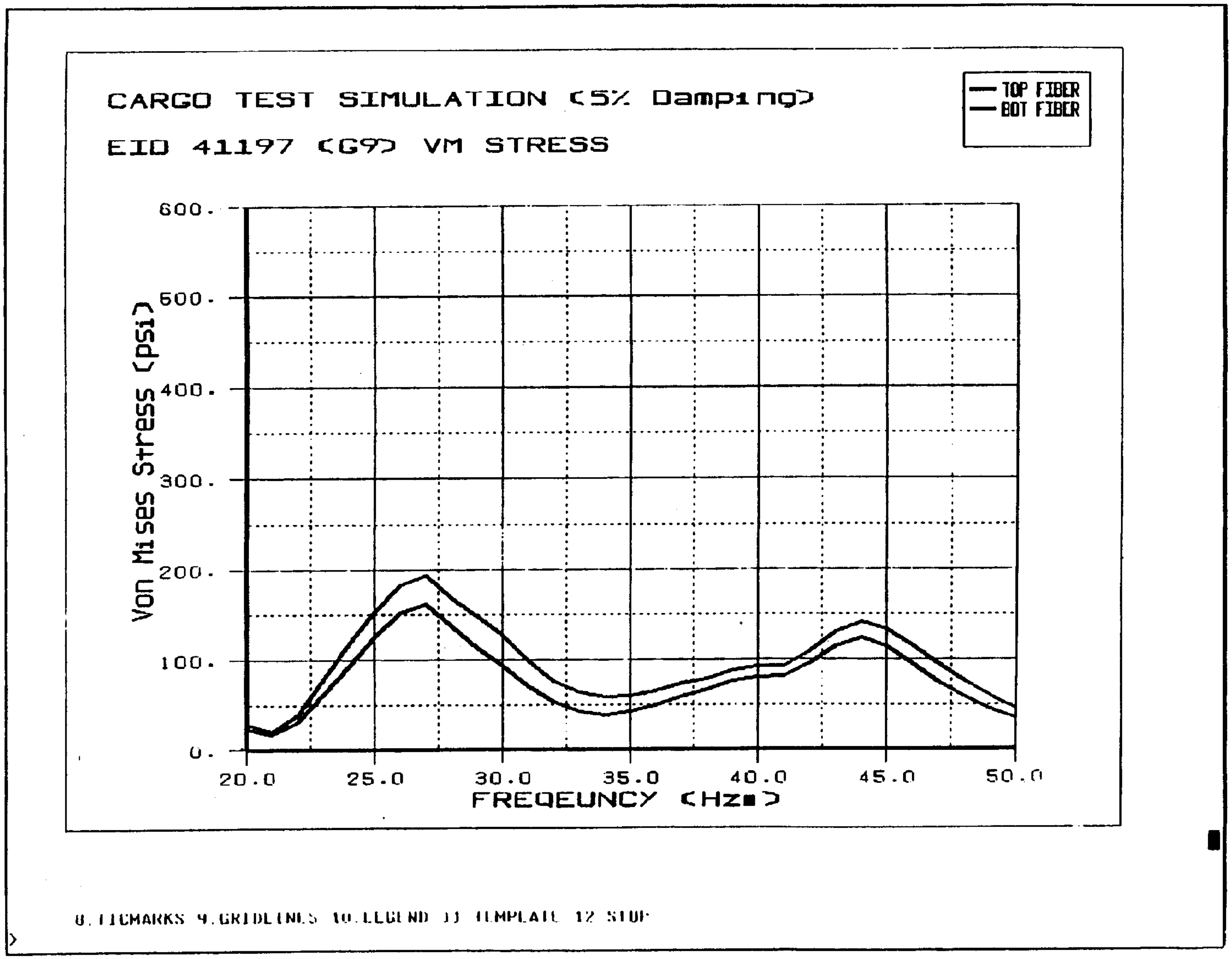

Figure 8c - Computer Prediction of the Equivalent Von Mises Stress at the Tip of the Floor Panel with 5% Modal Damping (Location of Strain Gauge 9)

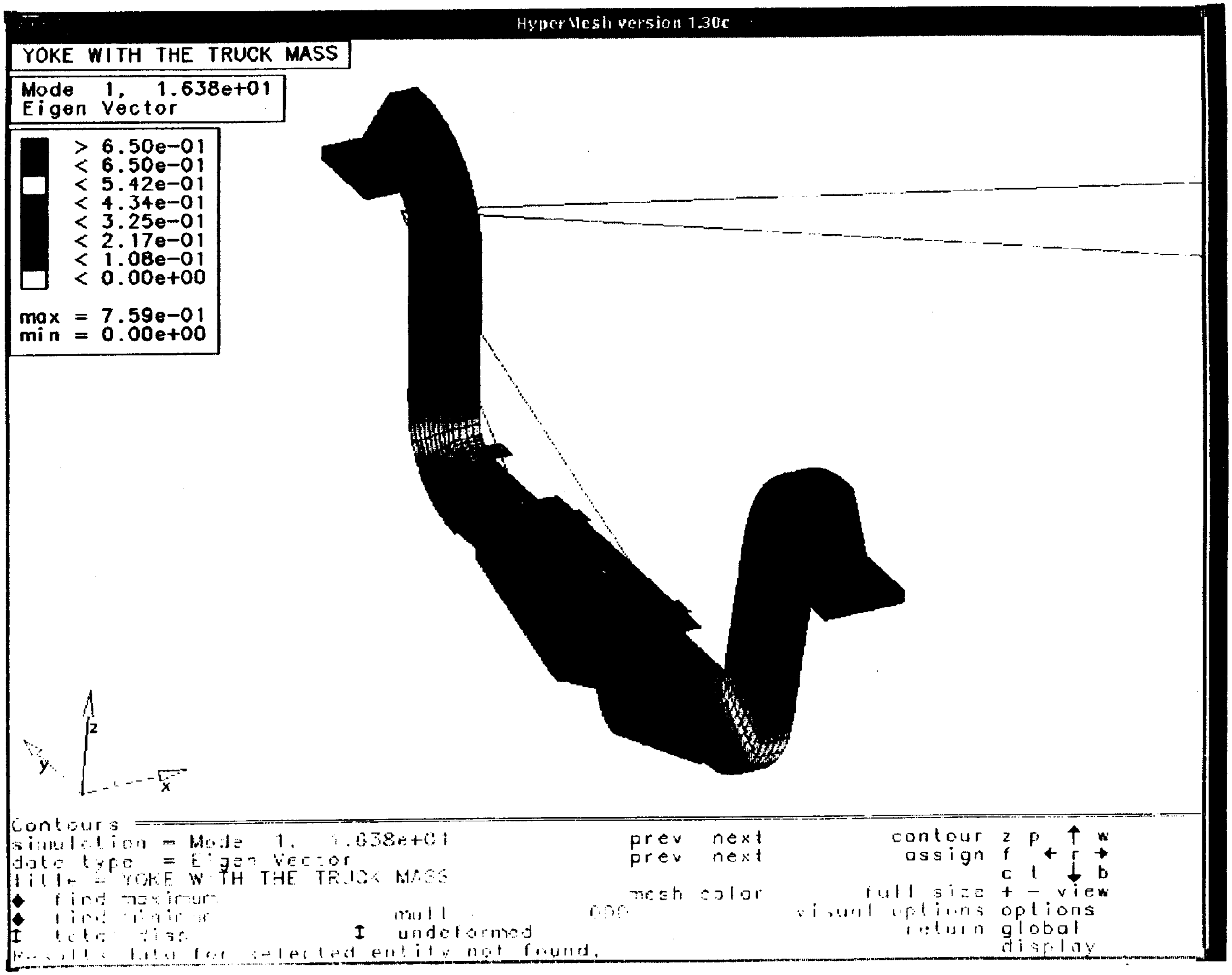

Figure 9.a - Contour Plot of Test Fixture Resonance Mode 1

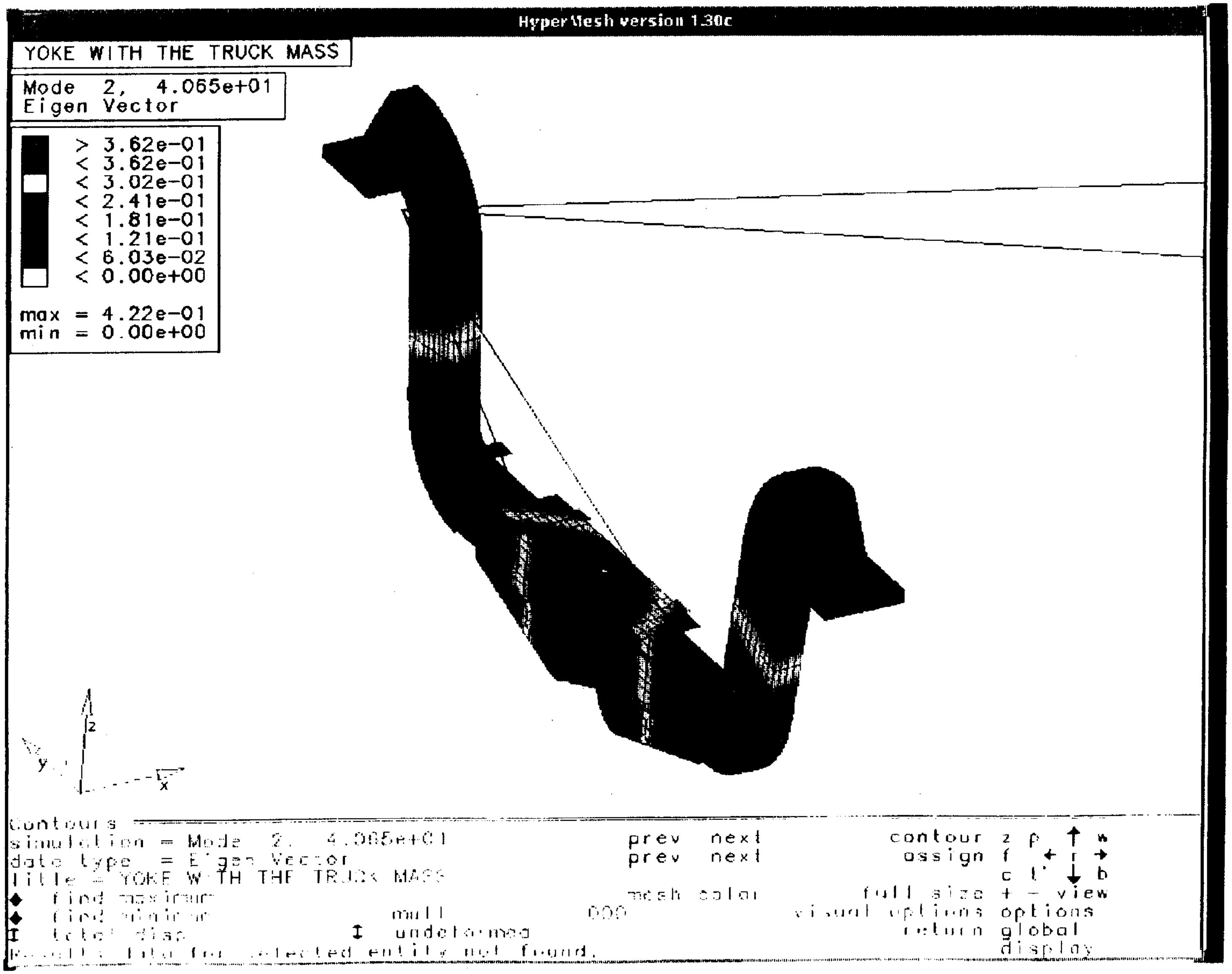

Figure 9.b - Contour Plot of Test Fixture Resonance Mode 2