Defence Science Journal, Vol 47, No 2, April 1997, pp. 259-264

Structural Analysis of a Tracked Vehicle Hull

M. Mala
Combat Vehicles Research & Development Establishment, Madras-600 054.

and

N. Siva Prasad
Indian Institute of Technology, Madras-600 036.

ABSTRACT

The hull of a tracked military vehicle is complex in geometry and loading pattern. Analytical studies were carried out using numerically integrated elements for system analysis (NISA), a general finite element programme developed by the Engineering Mechanics Research Corporation (EMRC), USA. Structural changes in the initial design were made to bring deflection within acceptable limits. Dynamic stress levels for the hull structure were determined from strain gauge measurements. The resultant stresses were obtained adding the static and dynamic values. Finite element analysis was found to be very useful to check the rigidity of the structure at design stage and to suggest suitable modifications.

1. INTRODUCTION

Hull is the chassis of a tracked military vehicle, designed to carry firing equipment and to accommodate power plant, firing controls and other systems. It is made of armour plates of different thickness for providing protection against small arms fire. The requirement to counter the latest antitank weapons and high mobility in all-terrain conditions has made it necessary that the hull of the tracked vehicle should have sufficient strength to withstand various loadings. The hull of a military tracked vehicle has been analysed here to examine the design adequacy of the hull structure and to estimate the natural frequency which determines safe operating speeds. Narasimhan and Siva Prasad[1] used finite element analysis for gun turret. James, *et al*[2]. explained the vibration of mechanical systems with microcomputers. Dally and Riley[3] have given the details of the experimental stress analysis.

2. FINITE ELEMENT ANALYSIS

The hull structure with plates of different thicknesses is complex in configuration. The overall view of the hull is shown in Fig. 1. The main part of the hull is composed of roof plate,

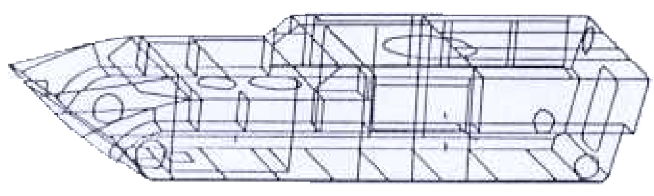


Figure 1. Overall view of the hull

Received 23 February 1996, revised 24 December 1996

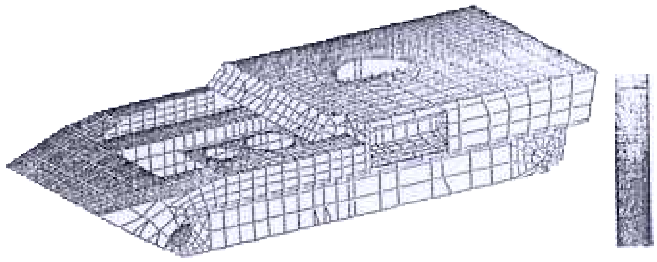


Figure 2. Finite element model of the hull

sponson side plate, sponson bottom plate, hull side plate and floor plate. Stiffeners are welded to the hull plates at various locations to provide rigidity to the structure. The vehicle is required to run under various terrain conditions and the loading pattern is complex. Conventional methods are inadequate to solve problems of this kind. Finite element analysis and economical analysis methods are reliable and well-suited for the development and optimisation of automotive structures. Numerically integrated elements for system analysis (NISA) – a general purpose finite element program[4] has been used for stress analysis of the hull structure.

2.1 Discretisation

The hull structure is discretised as shown in Fig. 2. Both quadrilateral and triangular shell elements are used in the model. The stiffeners are modelled using three-dimensional (3-D) beam elements.

No. of nodes	=	2111
No. of quadrilateral shell elements	=	1921
No. of triangular shell elements	=	125
No. of 3-D beam elements	=	374
No. of elements	=	2420
Total degree of freedom	=	12582

The model has been checked for element connectivity, element distortion, element normal distortion and boundary check.

2.2 Analysis

As the tracked vehicle is supported on torsion bars, it is assumed that the hull is fixed at torsion bar anchor points. This will be the extreme case of suspension locking when all the load will be taken by the hull structure alone. Two load cases are considered in the analysis.

(i) Case 1: Dead loads and weight of the vehicle.

(ii) Case 2: Dynamic loads due to track tension, vehicle acceleration and road undulations in addition to dead loads and weight of the vehicle.

The forces due to the recoil after firing have not been considered as external loads in the present analysis, as the missiles are turned by 90° from the initial axial position and the forces are transmitted

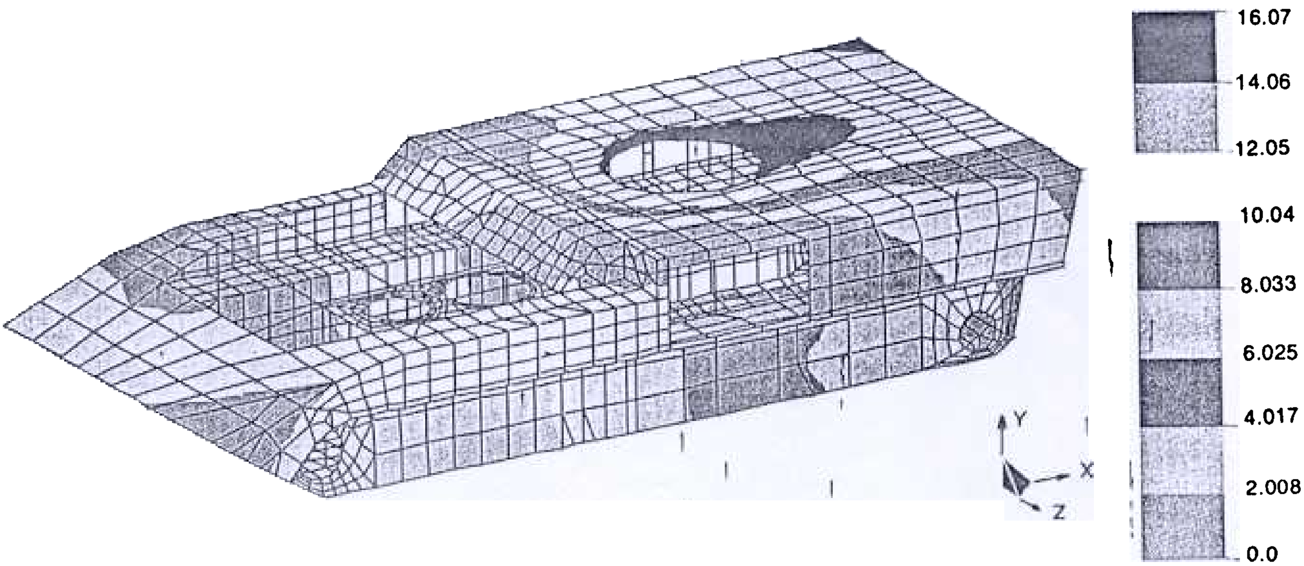


Figure 3. Resultant displacement contour load case-1

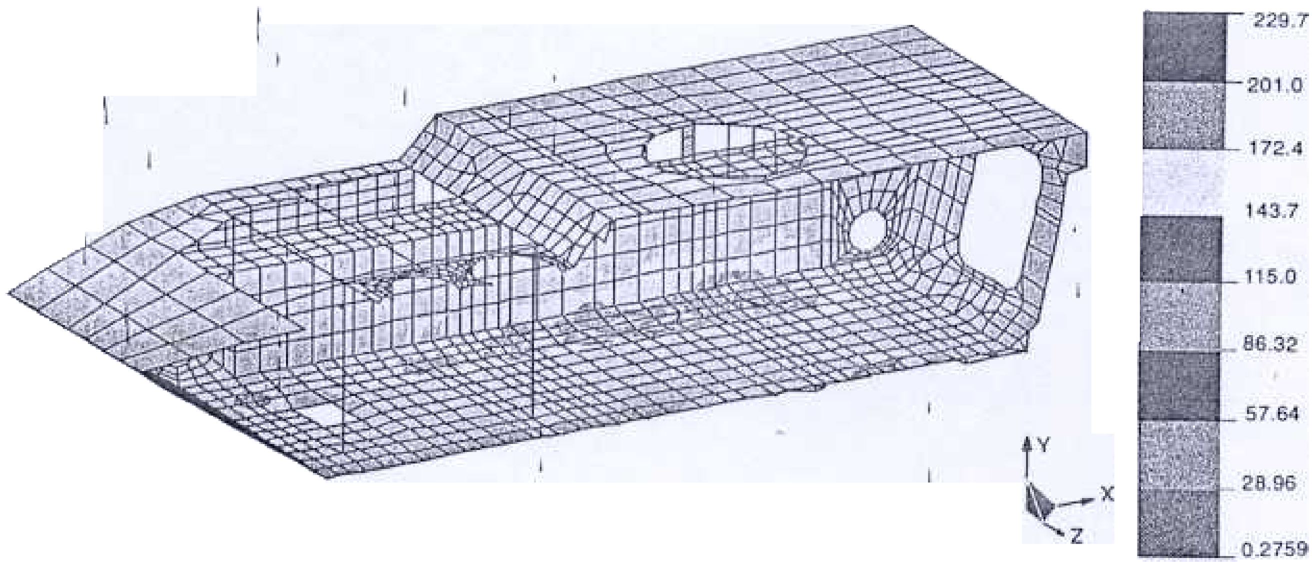


Figure 4. VON-Mises stress contour — load case-1

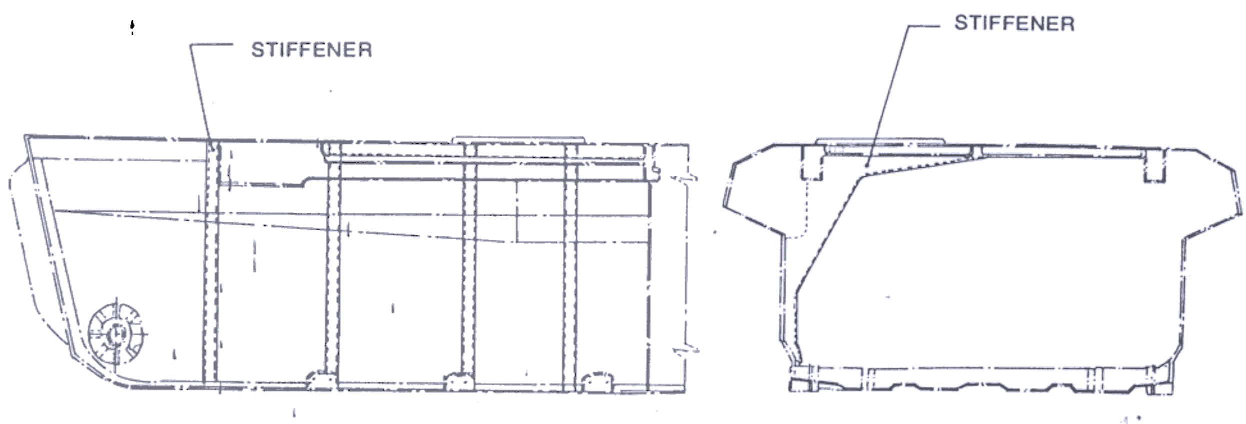


Figure 5. Modified geometry of the hull

to the ground directly by anchoring. The hull is analysed for stress and deflection at various points for load Cases 1 and 2. Resultant displacement contours and Von-Mises stress contours for load Case 1 are given in Figs 3 and 4.

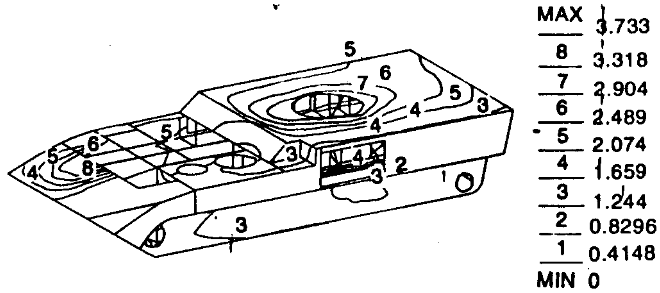
3. STRUCTURAL OPTIMISATION

Based on the results obtained from static analysis of the hull structure, it was found that deflection of the roof plate was above the permissible limits. This has to be minimised for proper operation of any firing equipment. Different geometries were studied to reduce the deflection of the roof plate of the hull. One such geometry is shown in Fig. 5. Stiffeners with varying cross-sections are introduced between the top plate and the bottom plate replacing the original stiffeners and additional stiffeners extending from the bottom plate to the side plate. Deflections from the static analysis of this geometry are shown in Fig. 6. It is seen that the deflection of the top plate is well within the permissible limits.

4. VIBRATION ANALYSIS

4.1 Natural Frequencies & Mode Shapes

An n degree-of-freedom system has *n* natural frequencies and for each natural frequency there is a corresponding normal mode shape that defines a distinct relationship between the amplitudes of



Figrue 6. Resultant displacement contour for modified structure — load case-1

generalised coordinates for that mode. The squares of the natural frequencies and the corresponding sets of coordinate values describing the normal mode shapes are referred to as eigen values and eigen vectors respectively and they are of fundamental importance in the analysis of free or forced vibration of multi-degree-of-freedom system. Many real systems have damping of less than 20 per cent and for this reason, the eigen values and eigen vectors of multi-degree-of-freedom systems are usually determined. Considering undamped free vibration, eigen values are the frequencies to be avoided in actual operation. Besides these eigen values, associated eigen vectors are needed to estimate the dynamic response of a multi-degree-of-freedom system by modal superposition technique.

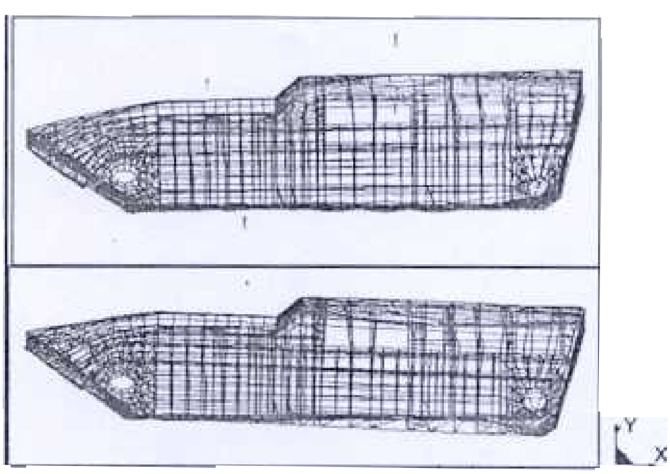


Figure 7. Free vibration analysis of the hull structure : mode 1 - 3.77 Hz, mode 2 - 4.64 Hz.

4.2 Modelling & Analysis

Various loads acting on the hull structure are modelled as lumped masses and undamped free vibration analysis is carried out for the hull structure. Lanczos' method has been employed for solving the problem, as this method permits considerable saving in labour for finding out the first few eigen values of a large size matrix. This is especially true when the eigen values are closely spaced, as in automobile bodies. The first two eigen values and mode shapes are shown in Fig. 7.

5. EXPERIMENTAL STRESS ANALYSIS

5.1 Strain Gauge Installation & Instrumentation

There has been a need to evaluate the structural rigidity of the vehicle and the adequacy of the stiffeners under severe dynamic excitations due to terrain undulations. The dynamic stress analysis is carried out using strain gauges. Four salient locations have been identified for installing strain gauges. The locations of strain gauges are shown in Fig. 8. Rosette strain gauges at 45° have been chosen for measurement at points 1 and 2, as the stress fields at these points are unknown. Linear weldable gauges are chosen for points 3 and 4, as these are located on stiffener beams and are inaccessible for gauge bonding. For dynamic strain measurement in field trials, the strain gauges are connected to the signal conditioning amplifier and the output of the amplifier has been recorded using a magnetic tape recorder.

5.2 Field Trials

The vehicle was run on high frequency sinusoidal terrain (± 125 mm amplitude, 4000 mm wavelength), cross-country and Aberdeen Proving Ground (APG) terrains. The recorder was played back, and typical waveforms indicating the strain variations were stored on a four-channel storage oscilloscope, digitised and transferred to a personal computer. The waveforms for the strain variations at location 1, when the vehicle was negotiating sinusoidal terrain and cross-country, are given in Figs 9 and 10, respectively. The strain and stress values were calculated and it was found that the maximum dynamic stresses at measurement points are well within limits.

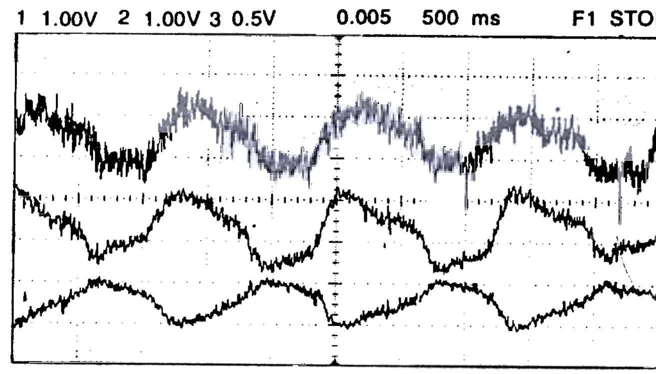


Figure 9. Strain variation at location 1 when negotiating sinusoidal terrain.

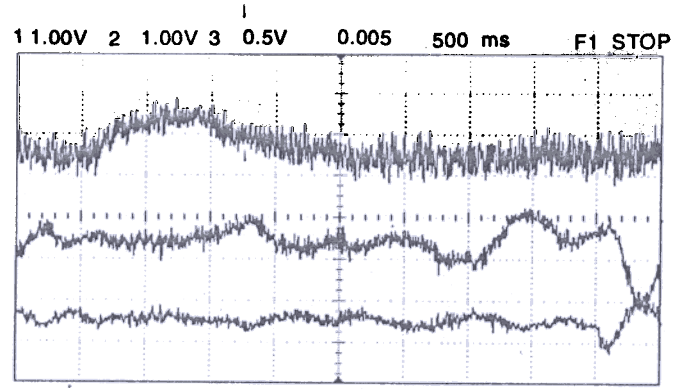
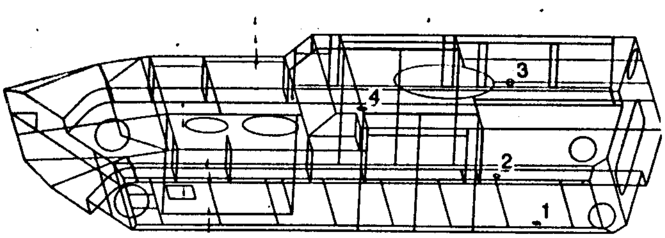
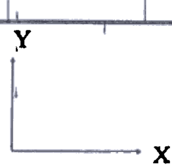


Figure 10. Strain variation at location 1 when negotiating cross-country terrain.



Location details	Rosette orientation	Angle of the gages i, j & k with respect to global axes	Channel Nos. corresponding to gages i, j & k
Bottom plate near 7th stn. torsion bar (left side of hull)	i j k	45, 0, -45	1, 2, 3
Side plate near 7th stn. torsion bar (right side of hull)	i j k	45, 0,-45	3, 2, 1
Top plate near the slewing ring (centre of hull)		Along X axis	1
Top plate near the slewing ring (left side of hull)		Along Y axis	2



6. RESULTS & DISCUSSION

During field trials, strain gauges were balanced for zero strain at the static position of the vehicle. This does not mean that the stresses in the vehicle are zero at static position. To measure the static stress, the vehicle has to be built with the strain gauge mounted at the location. The strain gauge instruments have to be balanced and after the structure is built, variations in the strains have to be noted. This process is lengthy and difficult. The static stresses on the vehicle can be ascertained through finite element analysis. The static stresses at each measurement point added to the dynamic stresses give the total stresses on the vehicle. These are given in Table 1. It is seen that the total stresses at chosen locations are well within limits.

Table 1. Stresses at measurement points

Measurement location	Dynamic stress from measurement (N/mm^2)	Static stress from finite element model (N/mm^2)	Total stress (N/mm^2)
	σ_1 = 13.02	σ_1 = 20.3	σ_1 = 33.32
	σ_2 = -113.29	σ_2 = 31.01	σ_2 = -82.28
2	σ_1 = 91.46	σ_1 = -0.15	σ_1 = 91.31
	σ_2 = -45.91	σ_2 = -43.24	σ_2 = -89.15
	σ_x = 29.97	σ_x = 32.18	σ_x = 62.15
4	σ_z = -19.96	σ_z = -34.61	σ_z = -54.57

7. CONCLUSION

The finite element model for the tracked vehicle hull structure has been developed and analysed. Dynamic stress levels for the hull structure have been determined from strain gauge measurements. As the hull structure will be used for variants of the tracked vehicle system, this model will be very useful for checking the rigidity of the structure at the design stage itself. Hence, the model developed has proved to be an important design tool.

ACKNOWLEDGEMENTS

The authors are grateful to Shri M. Natarajan, Director, Combat Vehicles Research & Development Establishment (CVRDE), Madras, for extending all facilities and granting permission to publish this paper. The authors also acknowledge the useful suggestions made by Shri P.L. Sathyanarayanan, Project Director, and Shri M. Paramasivan, Dy Director, during the course of this work.

REFERENCES

1. Narasimhan, G. & Siva Prasad, N. FEM analysis of gun tank turret. *Def. Sci.J.*, 1989, **39**, 257-68.
2. James, M.L.; Smith, G.M.; Wolford, J.C. & Whaley, P.W. Vibration of mechanical and structural systems with microcomputer applications, Harper & Row, Singapore, 1989.
3. Dally James, W. & William F. Riley Experimental stress analysis, McGraw Hill Singapore, 1991.
4. Engineering Mechanics Research Corporation : NISA II/DISPLAY III Technical manuals, USA, 1994.

Contributor

Ms Mala obtained her BE (Mechanical Engineering) from College of Engineering, Guindy, in 1982 and MTech (Machine Design) from Indian Institute of Technology (IIT), Madras, in 1996. She has been working as Scientist at the Combat Vehicles Research and Development Establishment (CVRDE), Madras.