

Design, Analysis and Optimization of Heavy Vehicle Chassis Using Finite Element Analysis

Abdulkadir Yasar (Corresponding author)
Cukurova University, Ceyhan Engineering Faculty, Department of Mechanical Engineering,
P.O. Box 01950, Ceyhan, Adana, Turkey, E-mail: ayasar@cu.edu.tr

Durmus Ali Bircan
Cukurova University, Department of Mechanical Engineering, PO Box 01460,
Sarıcam, Adana, Turkey

Abstract

The most prominent challenge in today's vehicle industry is how to cope with the accretive demand for ever higher performance, longer life, safety and lower weight of components so that fuel economy requirements can be achieved at realistic cost levels. Chassis of a truck is the skeleton for a vehicle providing structural strength and mounting points for other components. In this study, a truck ladder chassis was 3D- modelled using Computer Aided Designed (CAD) method via CATIA^{TM} software. Finite Element Analyses (FEA) were conducted on the truck ladder chassis by varying materials such as AISI 1006 Structural Steel and ASTM Class 30 Cast Iron using ANSYS software. The analyses were executed by considering various materials, number of wheels, type of profiles, and supports used. The optimum levels of the control factors for minimizing the deformation using S/N rates were determined for a wheel size of 6x4, I profile, and supports used. The optimization was finally carried out using Taguchi method in the Minitab software.

Keywords: Vehicle Ladder Chassis, Computer Aided Design, FEA, Taguchi method.

1. Introduction

Chassis is one of the important parts used in automotive industry. Automotive chassis is a frame just like a skeleton which was expected to fulfill functions such as (i) providing mounting points for the suspensions, tires, axle assemblies, brakes, the steering mechanism, the engine and gearbox, the final drive, the fuel tank, and the seating for the occupants, (ii) providing rigidity for accurate handling, and (iii) protecting the occupants against external impacts [1]. The chassis provides strength and stability to the vehicle under different conditions. The main function of the chassis is not only to support the components and payload mounted upon it including engine, body, passengers and luggage, but also to maintain the desired relationship between mounting points of the suspension and steering mechanism. The energy absorbed by the chassis under impact conditions lower the energy levels transmitted to a vehicle's occupants and surroundings lowering the chances of injury. The chassis is subjected to stress, bending moments and vibrations due to road roughness, weather and components mounted on it when the truck travels along the road. The stress acting on the chassis varies with displacement and each part on the car chassis [2]. For a chassis to fulfill these functions, it ought to be low weight in order to lessen inertia and allow for satisfactory performance. It must be sturdy enough to endure the loads generated by the interactions between the driver, engine, power transmission, and road conditions [3]. It must also be sufficiently rigid to tolerate shocks, twists, vibrations, and other stresses. In conjunction with strength, a significant focus in chassis design is the accommodation of adequate bending stiffness for superior handling characteristics. Therefore, maximum stress, maximum equilateral stress and deflection properties constitute essential criteria for the design of the chassis [2]. There are several types of automotive chassis which include ladder chassis, backbone chassis, monocoque chassis, and tubular space frame chassis. Ladder chassis is considered to be one of the earliest forms of automotive chassis still in use by most of the SUVs today

Demand is rising for the overall features of trucks, not just improved cost and weight. Consequently, an increasing focus on optimization and modularization has evolved together with numerous vehicle variants which altogether require the employment of efficient analysis methods. Finite element models in chassis design process are generally capable of incorporating almost all the characteristics and properties by changing geometrical dimensional details and their properties, loading conditions, different types of solver based on the definition of problems and its applications.

1. 1 Basic Concept of FEA

FEA has now become an integral part of Computer Aided Engineering (CAE) and is being extensively used in the analysis of many tedious real time engineering problems [4]. The field of FEA has developed a lot and this analysis greatly depends on rigorous mathematical foundation. Many powerful software tools and packages are available promoting its widespread use in industries [5]. Finite Element Method (FEM) is a computational technique which is employed for achieving approximate solutions for boundary value problems in engineering. Concisely stated, a boundary value problem is a mathematical problem that requires the satisfaction of a differential equation everywhere within a known domain of independent variables and also the specific conditions on the boundary of the domain via one or more dependent variables. A plain description of the FE method is that it simply involves slicing a structure into several elemental pieces, then reconnecting these elements at nodes; these nodes can be conceptualized as pins or drops of glue that hold the elements together [6].

FEA consists of three main steps, namely, pre-processing, solution, and post processing. Pre-processing (i.e. model definition) includes definition of the geometric domain of the problem, the element type(s) to be employed, the material properties of the elements, the geometric properties of the elements (length, area, etc.), the element connectivity (mesh of the model), the physical constraints (boundary conditions), and the loadings [6]. The solution comprises the governing algebraic equations in matrix form, computes the unknown values of the primary field variable(s), and gathers the findings. The computed results are then employed to determine the additional and the derived variables such as reaction forces, element stresses, and heat flow with the help of a back substitution step. In post processing, analysis and evaluation of the results are carried out stepwise.

1.2 Taguchi method

Taguchi robust parameter design has been extensively employed over the past ten years for solving numerous single response process parameter designs. The Taguchi method, utilizing Orthogonal Arrays (OAs) to design an experiment and Signal-To-Noise Ratio (S/NR) to evaluate response performance of the experimental runs, has been employed in many single response applications in order to determine the optimal parameters/levels combinations with the aim of reducing response variation and simultaneously relocating the mean to the desired value [7].

The Taguchi method includes reduction of variation via a process developed through robust experimental design. Taguchi developed a method for designing experiments in order to investigate how the different parameters influence the mean and how the variance of a process performance characteristic defines the degree of wellness for the process's functioning. The experimental design proposed by Taguchi includes employment of orthogonal arrays for organizing the parameters that influence the process and the levels at which they ought to be varied. Instead of conducting compulsory tests for all of the possible combinations in the way that the factorial design employs, the Taguchi method tests the pairs of the combinations. This enables compilation of the necessary data for determining which factors are of utmost influence to the product quality with a minimum amount of experimentation, consequently saving time and resources [8]. Proposed Taguchi design procedure can be seen in Fig. 1.

The need of the present work is to contribute to the understanding of FE simulation of chassis design and analysis. Although a lot of research work has been carried out with regard to FEA of chassis, only few studies have attempted to simulate the chassis design. The present research work is different from the literature in respect to modeling, design, and analysis of heavy vehicle chassis for various effects of stress distribution. In this study, a truck ladder chassis was designed and a 3D CAD model was created using CATIA software. Structural analyses were then conducted on the truck ladder chassis by varying materials (AISI 1006 Structural Steel and ASTM Class 30 Gray Cast Iron) using ANSYS. The analysis was carried out by considering materials, number of wheels, type of profiles, and supports used. Finally, the results were analysed and optimized using Taguchi method in the Minitab program.

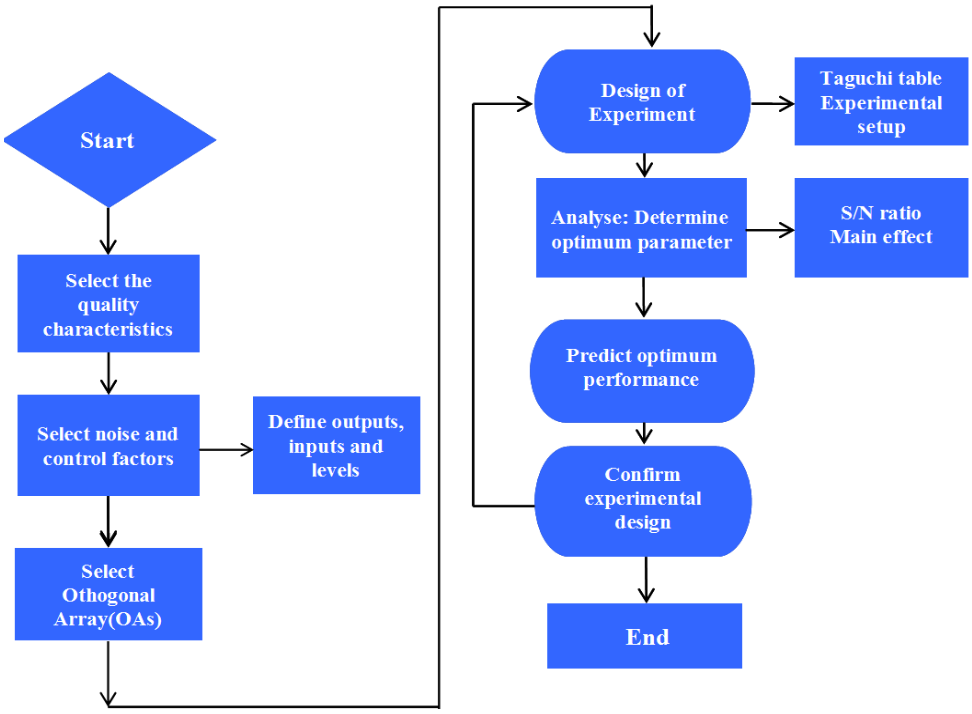

Fig. 1 Proposed Taguchi design procedure

2. Literature Review

FE models of vehicles and vehicle components are being progressively applied in preliminary design analysis, vehicle crashworthiness evaluation, and component design. Because of the importance of the chassis in vehicle design and crash simulation, it has been the object of numerous design investigations [9]. Karaoglu and Kuralay [10] conducted a stress analysis on a truck chassis by using FEM. The analysis revealed that increasing the side member thickness could alleviate the stresses on the joint areas; however, it was necessary to take into consideration that the overall weight of the chassis frame was also increased. Using local plates only in the joint areas could also increase side member thickness. Thus, it was possible to prevent an excessive increase in the chassis frame weight.

Nor et al. [11] analyzed stresses on a low loader structure design via FEA. FEM, simulations and analysis were carried out using CATIA V5R18 software. Computed results were compared to the analytical calculation and it was revealed that the location of maximum deflection was consistent with the theoretical approximation, although with variation of the magnitude. A safety factor for the low loader structure was calculated as well. Patel and Patel [12] carried out static structural analysis of a truck chassis. It was possible to easily analyze the structural systems of the chassis using FEA techniques. Accordingly, a proper FEM of the chassis was developed. The chassis was modeled in PRO-E environment. FEA was performed on the modeled chassis using ANSYS Workbench. The highest stress produced was registered as 106.08 MPa by FE analysis. The calculated maximum shear stress was registered as 95.43 MPa. The result of the FE analysis was 10% greater than the result of the analytical calculation. The maximum displacement of the numerical simulation result was 3.0294 mm. This result of the numerical simulation was 5.92 % greater than the result of the analytical calculation, which was 2.85 mm. The difference between the analytical and the numerical calculations occurred because of the simplification of the model and the various uncertainties of the numerical calculation. Sanjay et al. [13] performed the modal and static structural analysis for TATA 407 fire truck chassis with steel frame as well as carbon fibers. Using results of the analyses, stress, strain, and total deformation values were compared for both materials.

Structural optimization utilizing computational tools has become a major research field over the last years. Methods commonly used in structural analysis and optimization may require considerable computational cost depending on the complexity of the problem. Among the available methods, Design of Experiment (DOE) may be combined with classic analysis for reducing computational effort without compromising on the final solution quality [14].

A review of the literature indicates that the Taguchi method is the best available option for design of experiments when a vast number of process parameters are involved. The Taguchi approach is quite suitable in experimental design for designing and developing robust products or processes irrespective of variations in the process parameters and/or variations in the environmental conditions ions [15]. Hsua et al. [16] employed a FEM-based Taguchi method for examining the effects of various factors for attaining robust design of a body cage. The FEM-based Taguchi methods competently decreased the time and effort required for evaluating the design variables of implants and assessed the contribution of each design variable fairly.

3. Material and Methods

3.1 Design Calculations for Truck Ladder

The analytical calculations for the considered truck ladder were done using the basic design parameters. The pre-required data considered are as follows:

Materials: Chassis must be rigid and strong enough to absorb the vibrations produced by engine, suspension, and drive line. The most commonly used materials for chassis are AISI 1006 steel and ASTM Class 30 cast iron (Material properties can be seen in Table 1).

Wheel: There are several types of truck chassis connected to wheel such as; 4x2, 6x4, 6x2/4 and 8x6/4. In this analysis; 4x2 (2 front wheels, 2 rear wheels, and 2 rear wheel driven truck) and 6x4 types were used.

Profile: There are not many known profiles used in the design of chassis. The most commonly used types are U and I profiles. Thus, these profiles were selected in the analysis.

Support: Supports are used to decrease total deformation of chassis while increasing the strength of the system. In this analysis, the supports were used at the upper side of the body.

Table 1 Material properties of AISI 1006 Cold Drawn Structural Steel and ASTM Class 30 Gray Cast Iron [17]

Properties	**AISI 1006 Steel, Cold drawn**	**ASTM Class 30, Gray Cast Iron**
Density	7.872 g/cm³	7.15 g/ cm³
Hardness, Brinell	95	210
Hardness, Vickers	98	220
Tensile Strength, Ultimate	330 MPa	214 MPa
Tensile Strength, Yield	285 MPa	-
Modulus of Elasticity	205 GPa	110 GPa
Poisson's Ratio	0.29	0.28
Shear Modulus	80 GPa	36 - 45GPa

3.2 Design of Truck Ladder

The Truck Ladder was designed in CATIA software as seen in Fig. 2. CATIA, a feature based parametric solid modeling program, is the standard in 3D product design, featuring industry-leading productivity tools that promote best practices in design. The design procedure was based on creating a model, viewing it, assembling parts as required, and then generating any drawings which are required.

3D CAD models created via CATIA were transferred to the ANSYS program to be ready for FEA. The selection of the element type in the mathematical model, creating the mesh form, determining the contact areas, boundary conditions, environmental and material properties, and the type of analysis were performed in the program interface. The Taguchi parameter design phase serves towards the objective of determining the optimal truck ladder parameters in order to achieve the lowest deformation. The relationship between the control factors (materials, profiles, number of wheels, supports used) and output response factors (deformation) and the optimal conditions of the parameters of truck ladder matters were considered in this study.

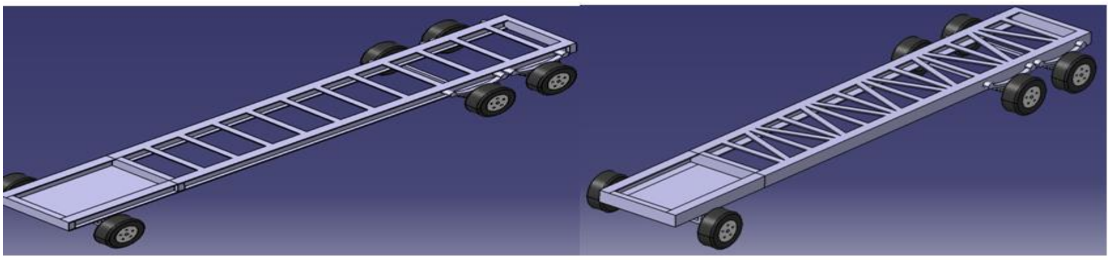

Fig. 2 Designed Truck Ladder using CATIA 6x4 wheel type with and without supports.

To accommodate four control factors into the experimental study, a standardized Taguchi-based experimental design L16 (2^4) was chosen to be used in this study. There were 16 experimental runs that need to be conducted with the combination of varying levels of each control factor (A-D). The selected parameters were displayed in Table 2 with their codes and values.

Table 2 Parameters and levels used for orthogonal array

Control Factors	Level 1	Level 2
Materials	AISI 1006 Steel	ASTM Class 30 Cast Iron
Number of wheels	4x2	6x4
Profiles	U	I
Supports	Used	Not Used

4. Results and Discussion

4.1 Results of Finite Element Analysis

CAD model of truck ladder was created and imported into ANSYS for FEA. In order to calculate the total deformation, two different forces were individually applied to the rear of the body. The first was 100 kN which was equivalent to load of nearly 10 tones. The second was 150 kN equivalent to 15 tones approximately. In order to account for the cabinet and engine weights, another force was applied to the front of the body. At all designs, the force was applied at a constant value of 30 kN equivalent to a load of approximately 3 tones. By taking these into consideration, total deformations in the truck ladder were obtained as shown in Fig. 3-6.

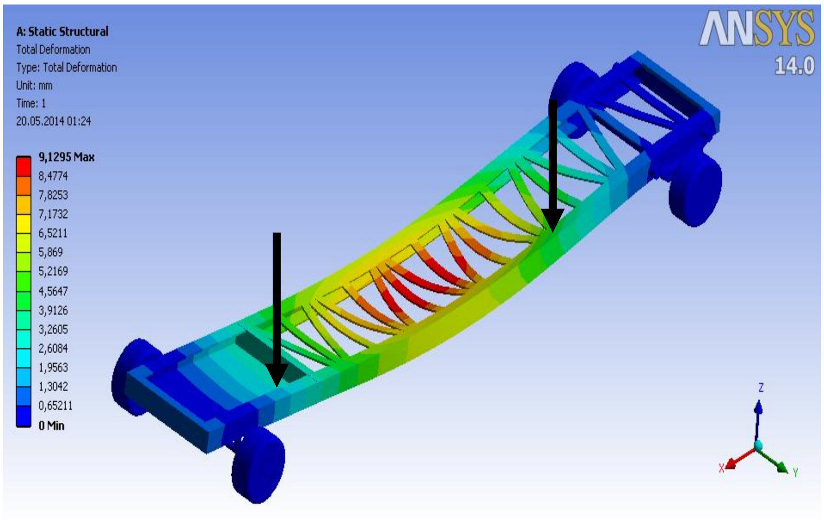

Fig. 3 Total deformation for L16-1 under applied load of 100 kN

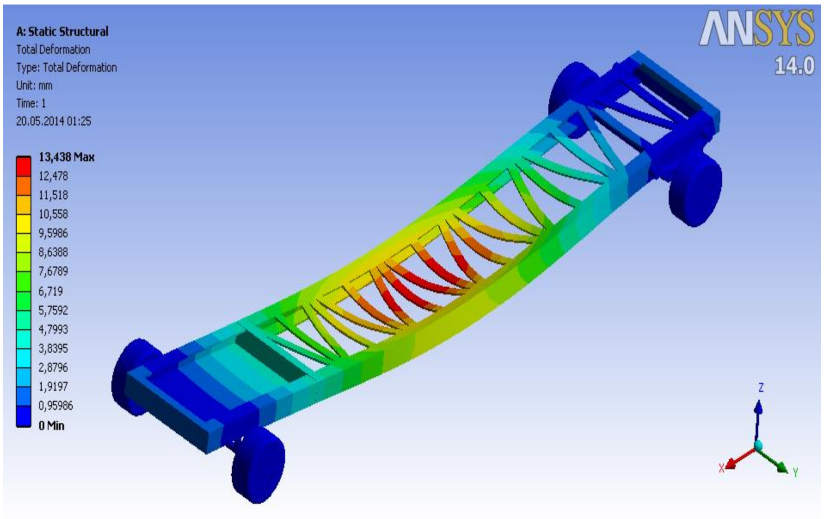

Fig. 4 Total deformation for L16-1 under applied load of 150 kN

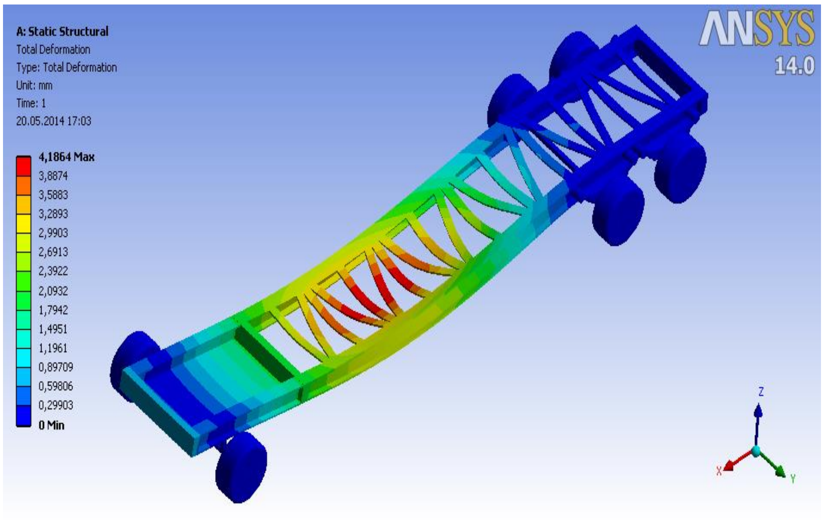

Fig. 5 Total deformation for L16-5 under applied load of 100 kN

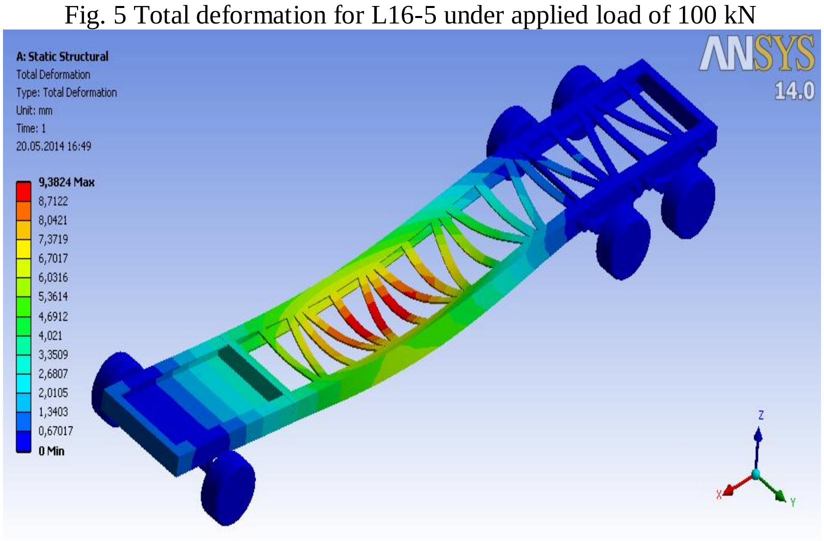

Fig. 6 Total deformation for L16-15 under applied load of 100 kN

4.2 Transferring the results into Minitab

The previously obtained ANSYS total deformation results were later transferred into Minitab software that was created during design stage as can be seen in Table 3. The ANSYS results were used as input values for Taguchi L16 (2^4) orthogonal array. The collected experimental data were then analyzed by using Minitab software to determine the effects of each parameter (materials, number of wheels, type of profiles, and supports used) on truck ladder chassis. The optimum combination was obtained by choosing the level with the lowest S/N ratio for each control factor. The level 1 of parameter A, C, D and level 2 of parameter B were the optimum levels based on statistical analysis for design, respectively. Therefore, the optimal combination was A1B2C1D1 (denoted as L16-5). The larger wheel size of 6x4 was declared as optimum design due to better resistance of the deformation. Meanwhile, I profile and the supports used cases were projected as the best response given under the ideal condition of minimum total deformation value. When the resultant stress existing due to forces were analyzed lower stresses were observed at critical locations shown in red color in Fig. 3-6. Thus, the control factor of support (C) at Level 5 (100kN) demonstrated the optimum results. The prediction results were compared with the experimental results obtained with all materials used for truck ladder chassis. Both training and test data were summarized in Table 3. Subsequently, AISI 1006 steel showed better characteristic for smaller resultant deformation value, which would be the optimum condition.

Table 3 Optimum design results

L16	Material	Wheel	Profile	Support	Total Deformation for 100kN (mm)	Total Deformation for 150kN (mm)	S/N Ratio	Mean
1	1	1	1	1	7.9909	11.684	19.39571051	9.83745
2	1	1	1	2	8.1187	11.858	19.5305229	9.98835
3	1	1	2	1	9.1295	13.438	20.57108423	11.28375
4	1	1	2	2	9.7592	14.5	21.17594617	12.1296
5	1	2	1	1	4.1864	6.1516	13.79427779	5.169
6	1	2	1	2	4.2892	6.2992	14.00348035	5.2942
7	1	2	2	1	5.1579	7.6336	15.62650844	6.39575
8	1	2	2	2	5.5473	8.2181	16.26139753	6.8827
9	2	1	1	1	14.544	21.266	24.59751274	17.905
10	2	1	1	2	14.763	21.562	24.72421821	18.1625
11	2	1	2	1	16.616	24.458	25.77271707	20.537
12	2	1	2	2	17.725	26.335	26.35934487	22.03
13	2	2	1	1	7.6171	11.193	18.99331652	9.40505
14	2	2	1	2	7.7989	11.453	19.1964745	9.62595
15	2	2	2	1	9.3824	13.891	20.82435108	11.6367
16	2	2	2	2	10.072	14.92	21.44184927	12.496

5. Conclusions

In this study, the Taguchi method was used to determine the optimal design parameters for the heavy vehicle chassis made of AISI 1006 Steel and ASTM Class 30 Gray Cast Iron materials with different number of wheels, type of profiles, and supports used. The experimental results were evaluated using FEM analysis with ANSYS.

Heavy vehicle chassis was designed by using 3D CAD parametric software CATIA and optimum levels of the control factors to minimize the deformation using S/N rates were determined. The optimal conditions were found at A1B2C1D1 coded as L16-5 (wheel size of 6x4, I profile, supports used and AISI 1006 steel material, respectively). According to the results of the statistical analyses, the most significant parameters were found to be the profile type and supports used.

The lowest S/N ratio corresponds to a better performance, regardless of the category of the control parameters. Therefore, lowest level of S/N ratio (13.79) and Mean (5.17) was obtained from the minimum deformation values.

6. References

[1] Linton, W. (2002). Analysis of Torsional Stiffness and Design Improvement Study of a Kit Car Prototype, MSc Thesis, Cranfield University.

[2] Patel, H., Panchal, K. C., Jadav, C. S. (2013). Structural Analysis of Truck Chassis Frame and Design Optimization for Weight Reduction. *International Journal of Engineering and Advanced Technology (IJEAT)*, vol. 2, no. 4, pp. 665-668.

[3] Agrawal, M. S. and Razik, M.D. (2013). Finite Element Analysis of Truck Chassis. *International Journal of Engineering Sciences & Research Technology*, vol. 2, no. 12, pp. 3432-3438.

[4] Vinodh, S. and Ravikumar, R. (2012). Application of probabilistic finite element analysis for crane hook design. *Journal of Engineering, Design and Technology*, vol. 10, no. 2, pp. 255- 275.

[5] Seshu, P. (2009). Textbook of Finite Element Analysis, PHI Learning Pvt. Ltd.,

[6] Hutton, D. V. (2004). Fundamental of Finite Element Analysis, Mc Graw Hill, New York.

[7] Tatjana, V., Vidosav, S. and Majstonovic, D. (2010). An integrated approach to optimize parameter design of multi-response processes based on Taguchi method and artificial intelligence. *Journal of Intelligent Manufacturing*, vol. 23, no. 5, pp. 1511-1528.

[8] Fraley, S., Oom, M., Terrien, B.and Zalewski, J. Design of experiments via taguchi methods: orthogonal arrays, available from < https://controls.engin.umich.edu/ >, (Accessed on 15.11.2014).

[9] Liu, Y. (2010). Development and evaluation of a finite element truck chassis crash model, *International Journal of Crashworthiness*, vol. 15, no. 1, pp. 107-113.

[10] Karaoglu, C. and Kuralay, N.S. (2002). Stress analysis of a truck chassis with riveted joints. *Finite Element in Analysis and Design*, vol. 38, pp. 1115-1130.

[11] Nor, M.A.M., Rashid, H., Mahyuddin W.M.F.W., Azlan, M.A.M. and Mahmud, J. (2012). Stress Analysis of a Low Loader Chassis. *International Symposium on Robotics and Intelligent Sensors, Procedia Engineering*, vol. 41, pp. 995-1001.

[12] Patel, V.V. and Patel, R.I. (2012). Structural analysis of a ladder chassis frame. *World Journal of Science and Technology*, vol. 2, no. 4, pp. 05-08.

[13] Sanjay, S.G., Abhijeet, K., Pradeep, G.P. and Baskar, P. (2014). Finite Element Analysis of Fire Truck Chassis for Steel and Carbon Fiber Materials. *Int Journal of Engineering Research and Applications*, vol. 4, no. 7, pp. 69-74.

[14] Patel, T.M., Bhatt, M.G., Patel, H.K. (2013). Parametric Optimization of Eicher 11.10 Chassis Frame for Weight Reduction Using FEA-DOE Hybrid Modeling. *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, vol. 6, no. 2, pp. 92-100.

[15] Dabade, U.A. and Bhedasgaonkar, R.C. (2013). Casting Defect Analysis using Design of Experiments (DoE) and Computer Aided Casting Simulation Technique. *Procedia CIRP*, vol. 7, pp. 616- 621.

[16] Hsua, W.H., Chaoa, C.K., Hsub, H.C., Lin, J. and Hsua, C.C. (2009). Parametric study on the interface pull out strength of the vertebral body replacement cage using FEM-based Taguchi methods. *Medical Engineering & Physics*, vol. 31, pp. 287-294.

[17] ASM Metals Reference Book, 3rd Edition, Editor: Michael Bauccio.