NOVATEUR PUBLICATIONS
INTERNATIONAL JOURNAL OF INNOVATIONS IN ENGINEERING RESEARCH AND TECHNOLOGY [IJIERT]
ISSN: 2394-3696
VOLUME 2, ISSUE 3 MARCH2015

FEA Based Validation of Weld Joint Used In Chassis of Light Commercial Vehicles (LCV) In Tensile and Shear Conditions

Mr.Yadav (Khot) Rahul Shivaji
Research Scholar K.L.Universiity Vaddeshwaram A.P.

Mr. Pankaj Bhakare
Research Scholar K.L.Universiity Vaddeshwaram A.P.

Abstract

Generally automotive systems are subjected to dynamic and static loads, due to hives driving and adverse conditions of the road surface, thus it causes cracks, noise, vibration and failure in an automobile, since in general the total effect of work for automobile system is lowered. Such effect is arising due to behaviour in types of loading, construction and condition of the work where the automobile system is and reducing its strength. Therefore however most of the Light Commercial Vehicles (LCV) chassis are manufactured by welding to reduce the weight of the chassis. Hence there is a more stress concentration occurs at a welded joint region. The fundamental task is to develop a model of welded joint to sustain various loads of LCV chassis. Effect of different design parameters, properties of material are observed with respect to the strength of the weld and location of stress concentrations are identified. During the design process model parameters are often altered to evaluate alternative welding joining choices, to reduce weight of the system to sustain varying loading conditions without failure.

Keywords: Light commercial vehicles (LCV), Welded joints, Finite Element Analysis (FEA)

Introduction

In current industrial practice, welds and welded joints are an integral part of many complex load- carrying structures. Unfortunately, welds are often the weakest portions of these structures and their quality directly affects the integrity of the structure. Failure strength is believed to have a close relation to the precise geometrical discontinuity of the welded joint. The ultimate goal to produce welds of suitable strength and at a reasonable cost. The major challenge in today's ground vehicle industry is to overcome the increasing demands for higher performance, lower weight and longer life of the component, all this at a reasonable cost & in a short period of the time. The chassis is the backbone of the vehicles and integrates the main truck component system such as the axle’s suspension, power train, cabs and trailer. Thus chassis is a major component of the vehicle system.

The fundamental task is to develop a model of welded joint to sustain various loads of LCV chassis. Effect of different design parameters, properties of material are observed with respect to the strength of the weld and location of stress concentration are identified. During the design process model parameters are often altered to evaluate alternative welding joining choices, to reduce weight of the system to sustain varying loading conditions without failure. A better approach to the prediction of welding deformation is using the combined technologies of experiments with calculation. With modern computing facilities, the Finite

Element (FE) technique has become an effective method for prediction and assessment of welding failure and distortions various factors, the quantitative prediction and the control of welding deformation especially for a large and complex welded structure is extremely difficult.

Literature Survey

Ashutosh Dubey and Vivek Trivedi [4] made finite element model for vehicle chassis. In Finite element model shell elements have been used for the longitudinal members & cross members of the chassis. The advantage of using shell element is that the stress details can be obtained over the subsections of the chassis as well as over the complete section of the chassis. Beam elements have been used to simulate various attachments over the chassis, like fuel tank mountings, engine mountings, transmission mounting, etc. Spring elements have been used for suspension stiffness of the vehicle. The vehicle model is fixed at the wheels. The model is tested with the experimental results determined for the opposite wheels at the bumps. The diagonally opposite wheels of the vehicle are lifted to full deflection of suspension & the stress is measured at six locations. The measured stresses & the stresses calculated in ANSYS for the vehicle model at these six locations are almost similar. Impact loads have been measured (in terms of .g.) experimentally by using accelerometers on the front & rear axles. Input spectrum for Power Spectrum Analysis has been obtained by usingFFTAnalyzer for the secondary roads at the driving speed of 30 kmph. The results of finite element analysis have been checked by experimental methods too, & very good resemblance has been found between both the methods.

M. Xiea, J.C. Chapman [1] has suggested that Friction-welded bar–plate connections are a basic structural component of Bi-Steel steel–concrete–steel sandwich construction. In Bi-Steel members, the bar–plate connections, embedded in concrete, are subject to tension, shear and bending. They describe experimental and numerical studies on the static and fatigue strength of the friction-welded connections with the bar loaded in tension. Finite element analysis is carried out to examine the effects of plate thickness, the collar (flash) formed after friction welding, and possible initial defects or fatigue induced cracks. It is found that except for 6 mm plate specimens, the static tensile capacity of the embedded connections is governed by the tensile strength of the bar connectors. In the fatigue tests, single fracture and double fracture mechanisms were observed. Some of the researchers address about the welding distortion considering residual stresses for a flat plate. Also, the mathematical behaviour of welded specimen is considered as a multisport welded structure is eventually determined by the strength of each single spot weld. Also existence of crack like imperfection in the welded joint is normally considered to eliminate and so called crack initialization stage of fatigue life. As such no certain guidelines or set of formulation is available for curved plate lap joint regarding overlap.

Experimentation and FEA with results

An experiment is a methodical trial and error procedure carried out with the goal of verifying or establishing the validity of a hypothesis. The mechanized the LCV chassis Electrical arc welding is used for the purpose of taking advantage of the heat developed by the electric arc that forms between an electrode (metal filler) and the material to be welded. Universal Testing Machine (UTM) is used for testing the body under tensile and shear loading

[3]. The component considers for tensile testing and shear testing and analysis purpose is as shown in fig. 1.Similar component is used for testing in FEA.

Fig. 1 Component used for testing purpose on Universal Testing Machine (UTM)

The results of stresses and deformations which are getting from UTM and FEA from Ansys Software are tabulated in Table No. 1 and which is a tensile loading. And in Table no. 2 shear loading is applied and corresponding values of stresses and deformations are tabulated.

Table No. 1 Comparative statement of Experimental and Finite Element Analysis (FEA) results for case of a tensile load

Sr.No.	LOAD (N)	Expt. Stress (MPa)	Expt. Deformation (mm)	FEA stress (MPa)	FEA Deformation (mm)
1	14560	404.44	1.4	454.58	0.825
2	13000	361.11	1.23	395.21	0.669
3	12500	347.22	1.03	376	0.551
4	12000	333.33	0.92	357.12	0.433
5	11000	305.55	0.85	319.02	0.397
6	10000	277.77	0.6	280.9	0.361

Table No.2 Comparative statement of Experimental and Finite Element Analysis (FEA) results when shear loading is applied

Sr. No.	LOAD (N)	Expt. Shear stress (MPa)	Expt. Deformation (mm)	FEA shear stress (Mpa)	FEA Deformation (mm)
1	13500	97.82	4.6	112.21	3.1
2	12000	86.95	4.3	96.61	2.6
3	11500	83.33	3.9	92.5	2.5
4	11000	79.71	3.7	88.57	2.4
5	10500	76.08	3.2	84.5	2.3
6	10000	72.46	2.9	80.51	2.2
7	9500	68.84	2.7	77.52	2

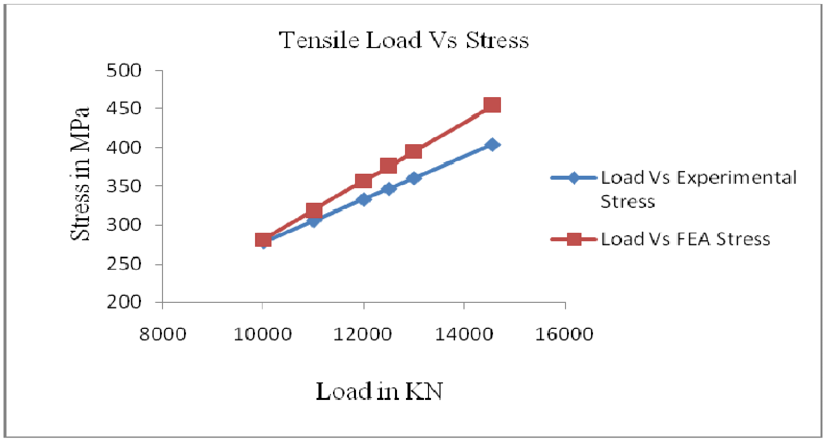

Chart No. 1 Variation of Experimental and FEA Stress with respect to tensile load

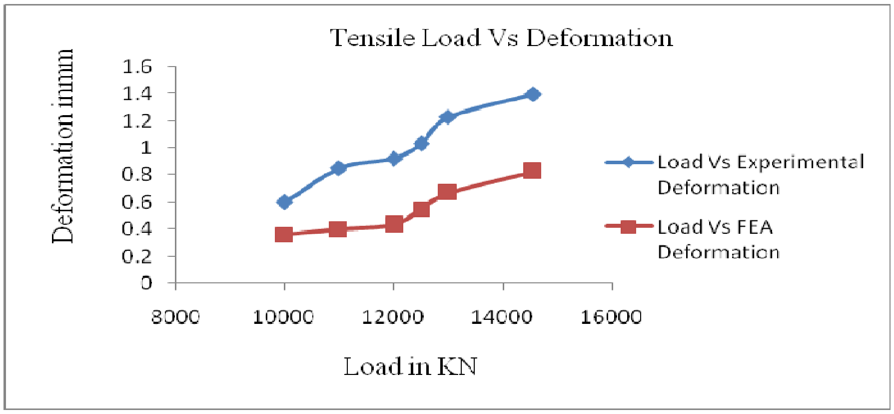

Chart No. 2 Variation of Experimental and FEA Deformation with respect to tensile load

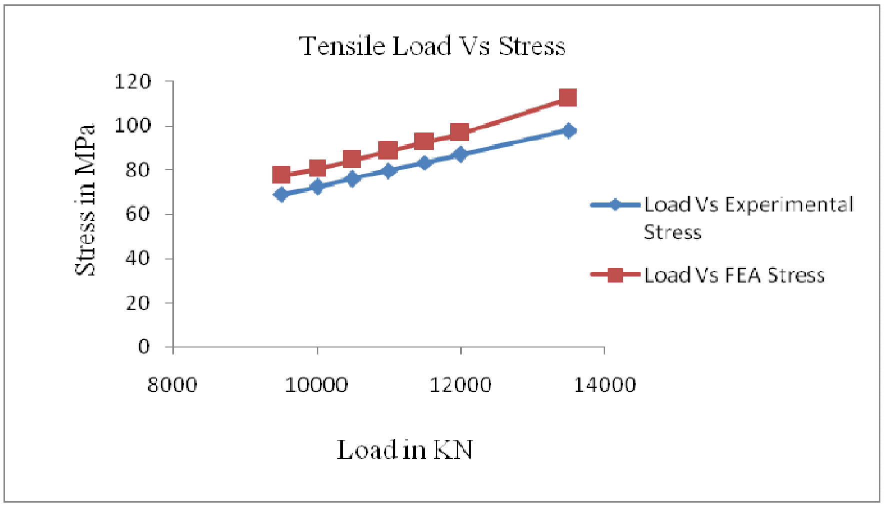

Chart No. 3 Variation of Experimental and FEA Stress with respect to shear load

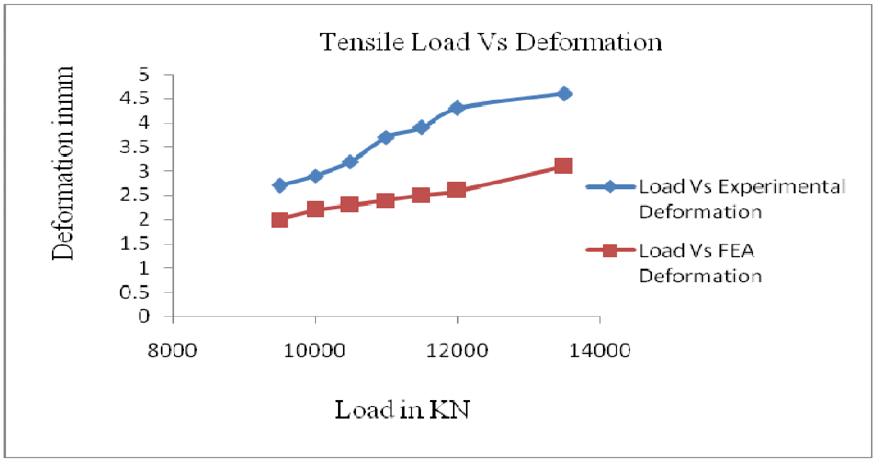

Chart No. 4 Variation of Experimental and FEA Deformation with respect to shear load

During the FEA and Experimental analysis it observed that failure is occurred at reduced i.e. welded section. From above graph it is cleared that as load applied on the chassis is increased the stress developed in structure increases also deformation changes. Graph no.1 and graph no.2 represent the relationship between tensile load acting on chassis with respect to stress and deformation respectively. As load increases from 12000N the deviation between the FEA results and experiment result also increases. The pattern of curve traced in graph no 2 is nearly same. Similarly when shear load is applied on the chesses corresponding stresses and deformations are plotted in graph no.3 and graph no. 4.

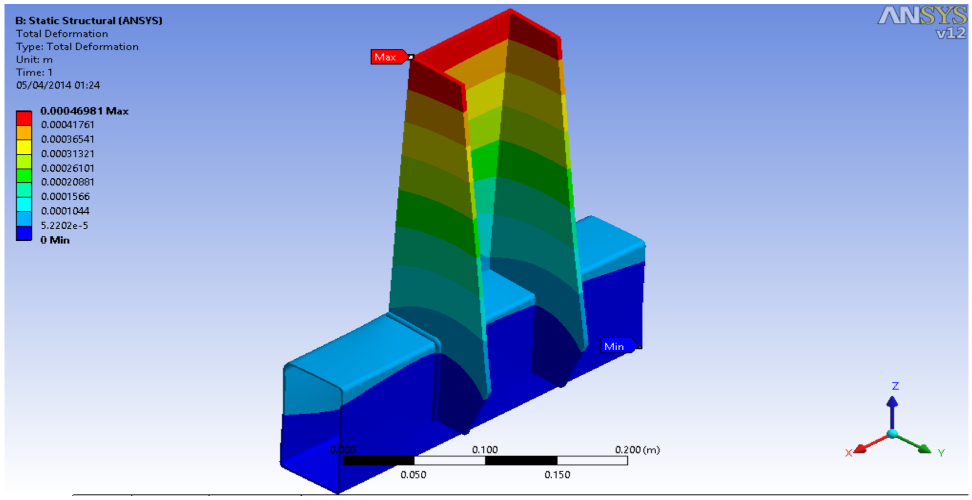

Fig.No.2 Variations in the deformation when analysis the LCV

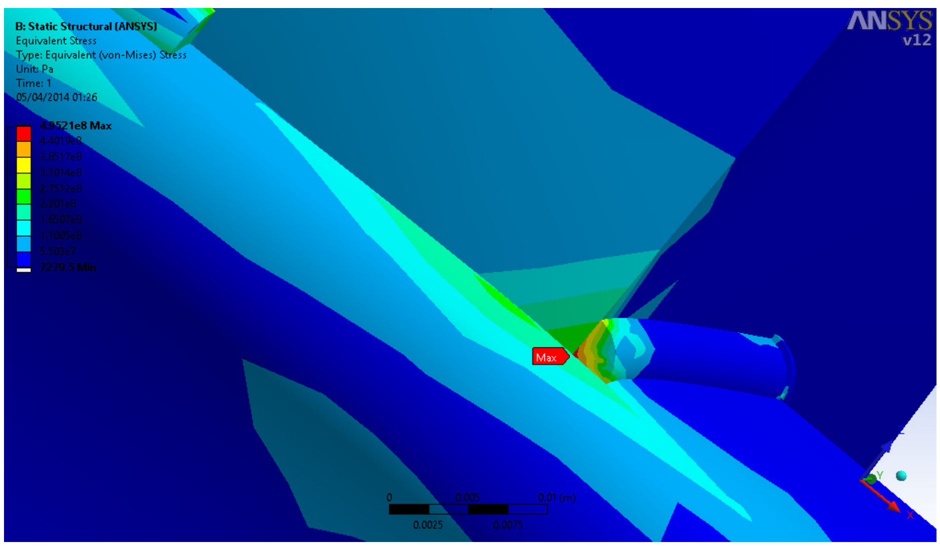

Fig.No.3 Magnifying view of location of stresses induced in chasses of LCV.

Conclusion

We have observed that, there is good agreement of FEA analysis & Experimental Analysis. Actually the Load acting on the chassis Specimen is 5000N but after the Analysis we have observed that it carry the load up to 10000 to 12000N i.e. it sustaining the load double than the actual.

In case of the compressive test, the buckling of the chassis specimen starts without failure of the welded joint after the load of 80 KN. Actual size of the weld throat is 5mm for the tested specimen for which the failure is occurs at 14560 N, if we will reduce the size of weld from 5mm to 4 mm weld size throat it will also sustain the load more than 5000 N, hence by reducing the size of weld ultimately we will reduce the cost of welding

Acknowledgment
The authors would like to thank to the Marathwada Mitra Mandal College of Engineering for providing helps in analysing the results by using softwares in this study.

References

1. M. Xie,∗, J.C. Chapman ―*Static and fatigue tensile strength of friction-welded bar–plate connections embedded in concrete‖ published in Journal of Constructional Steel Research 61 (2005) 651–673*
2. Dean Deng, Hidekazu Murakawa, Wei Liang ―*Prediction of Welding distortion in a curved plate structure by means of elastic finite element method Published in journal of Material Processing Technology 203(2008) page no. 252-266*
3. Rahul S. Khot,Dr. S.S.Gawade, Vinaay Patil (2012*). FEA based validation of Weld joints to determine effect of overlap length of weld strength in OCT-NOV, 2012, Vol. – I, Issue-III srjis, Pune.*
4. Ashutosh Dubey and Vivek Dwivedi . *The Vehicle Chassis Analysis by Load Cases & Boundary Conditions for Stress Analysis.2003*
5. Balbirsingh R.Guron, Dr. D.V. Bhope, Prof. Y. L. Yenarkar, "*Finite Element Analysis of Cross Member Bracket of Truck Chassis IOSR Journal of Engineering (IOSRJEN) ISSN: 2250- 3021, p-ISSN: 2278 – 8719 Vol. 3, Issue 3 (Mar. 2013), ||V 3 || PP 10 -16*