



FINITE ELEMENT ANALYSES OF MINI COMBINED HARVESTER CHASSIS AND HITCH

Received: 26 December 2016 / Accepted: 18 March 2017

Abstract: The perennial problems associated with harvesting of agricultural products in sub-Sahara Africa are not unconnected with financial limitations of the farmers. The design of low cost mini combine harvester was aimed at ameliorating the challenges of agricultural products harvest in Nigeria. The work presented here was a detailed analysis of low cost mini combine harvester chassis and hitch. The need for cost effectiveness, affordability, durability and efficiency of the designs necessitated detail analysis of the design to achieve the above objectives. Solidworks Finite Element Analysis (FEA) software was employed in carrying out both static and fatigue analysis of a low-cost mini combine harvester chassis and hitch design. The results were compared and contrasted, with appreciable improvements on available existing data. The stresses, displacements and strains on the chassis were significantly low with factors of safety of 2.48 and 2.80 for chassis and hitch respectively.

Key words: analyses, chassis, factor of Safety, FEA, harvester, hitch

Analiza šasije i kuke mini kombajna za žetvu pomoću metode konačnih elemenata Višegodišnji problem u vezi sa berbom poljoprivrednih proizvoda u oblasti sub-Saharne Afrike nisu nepovezana sa finansijskim ograničenjima poljoprivrednika. Dizajn jeftinih mini kombajna je imao za cilj ublažavanje izazova žetve poljoprivrednih proizvođača u Nigeriji. Rad predstavljen ovde je detaljna analiza šasije i kuka jeftinih mini kombajna. Potreba za isplativost, dostupnost, trajnost i efikasnost dizajna zahtevala je detaljnu analizu dizajna za postizanje gore navedenih ciljeva. Solidworks softver metoda analize konačnih elemenata (FEA) je korišćena za obavljanju analize šasije kombajn i za kuke za vuču kako statičke tako i analize zamora jeftinog mini kombajna. Rezultati su dali značajnija poboljšanja u odnosu na postojeće podatke. Naprezanja, pomeranje i deformacije su značajno niži sa faktorima sigurnosti 2,48 i 2,80 za šasiju i vučnu kuku respektivno.

Ključne reči: analiza, šasija, faktor sigurnosti, FEA, kombajn, kuka

1. INTRODUCTION

The recent downward turn in the prices crude oil in international market has given a wake-up call to a number of governments especially in sub-Sahara Africa on the need to strengthen agriculture as a mainstay of economy. In Nigeria, despite the neglect agriculture had suffered over time due growing dependence on oil, Nigeria can still be considered an agrarian economy with sizeable number of Nigerian, particularly in the Northern part still rely on agriculture for livelihood and as Small and Medium Enterprise [1]. It is against this backdrop that successive governments particularly the current economic transformation agenda is being focused around agriculture and rural development with strategic planning and adequate policy measure focusing on improved productivity and value addition, to move the sector out of stagnation [2].

Mechanization has ever remains one of the biggest challenges facing average Nigerian farmer, and Lamidi and Akande [3] was of the opinion that, the main constraint to successful farm mechanization in Nigeria is non-affordable farm machineries to farmers. Hitherto, majority of farmers still employ hand tools for cultivation and harvesting processes [4]. This predicament is not unconnected with lack of access to loan facilities and higher interest rate. The high yield grains and seedling made available to farmers by

government must be complimented by affordable machinery especially for harvesting and transportation from farm to the markets. Abdulkareem [5] was however optimistic on the varieties of indigenous technological objects recently developed in various forms. Among these emerging technologies was the work of Olukunle [6] who developed an indigenous self-propelled combine harvester for cowpea and soya bean. The machine is capable of operating at 0.33 ha/h at estimated feed rate of 322.22 kg/h between 100 to 600 rpm. The performance of the machine in various operational conditions was satisfactory but the estimated costs of the machine at ₦450,000 and ₦1,200,000 for one and it full scale three rows capacity respectively were beyond the reach of small scale farmers. Hence the need for cheaper, cost effective and durable alternative combine harvester capable of serving the need of the local farmers, which is achievable through Finite Element Analysis (FEA).

1.1 Finite Element Analysis (FEA) on Frame

Modern engineering practices have been revolutionized by the rise of Finite Element Analysis. It is a computational technique used to obtain approximate solutions of boundary value problems in engineering [7]. The basic procedure for completing any basic FEA involves four major steps namely: assigning of geometry, application of boundary

constraints and loads, meshing of object, and running the solver. It is noteworthy that engineering success in innovative products in today's competitive environment requires FEA simulation power. SolidWorks simulation technology ensures the quality and performance of design before dabbling to production. Comprehensive analysis tools allow models to be tested digitally for valuable insight in the design process. The tools has capability of establishing leverage between various competing parameters including weight and cost reduction, improve durability and manufacturability, optimize margins, and compare design alternatives to meet specific customer requirements.

Wakeham [8] carried out study and presented a good knowledge on fundamental requirements of chassis, stating how to start and optimised design making use of FEA software. The study ascertains the performance requirements of the automotive chassis. Among others, the chassis holds firmly all components while transferring lateral and vertical loads on the chassis via the suspension and to the wheels.

The ladder frame which is the oldest and the simplest style of frame is still employed in the construction of modern vehicles. It was formally used in the horse and buggy type carriages due to its good strength and stability under load. The ladder frame comprises of two beams running the total length of the vehicle, with other members cross linking the frame rail so as to hold it firmly together. Cars from the ladder frame era make use of braces at the sides of the frame, which add large amount of strength to the ladder chassis frame. X-bracing or cross-bracing is an effective way of strengthening the ladder frame. Fig. 1 shows the FEA images of both the ladder frame and the X-bracing frame.

Reina [10] did a study on the analysis of a bicycle frame using of finite element method to verify the validity of finite element approach. The experiment showed great extent of the functionality, implementation and high accuracy of the finite element method when compared to the experimental values. It was then concluded that the finite element results are greatly dependent on authenticity of the modelling methods and decisions employed. Jurgens Caravans Company performed an analysis using finite element analysis to analyse stress on chassis. The work was used to identify locations of high stress concentration to ensure prerequisite strengthening measures are put in place. Fig. 2 below shows the FEA static result obtained, in analysing the effect of load on the frame chassis.

The above case studies have shown that using solid works FEA simulation help to: provide accurate, efficient solution to difficult analysis problems; accelerate time-to-market; minimize design uncertainty, reduce error in production; and helps to reduce returns/warranty claims, thereby enhances productivity and profitability.

The major challenge in today's ground vehicle industry is how to overcome the increasing demands for higher performance, lower weight, and longer life of components at a reasonable cost and in a shortest

period of time [12]. Several works have been done on the analyses of frame ladder chassis including modifications on reduction of weight using different FEA software [8, 13-15]. However, in this work, modeling of low cost combine harvester component parts precisely chassis and hitch was carried out for static and fatigue analyses using SolidWorks FEA.

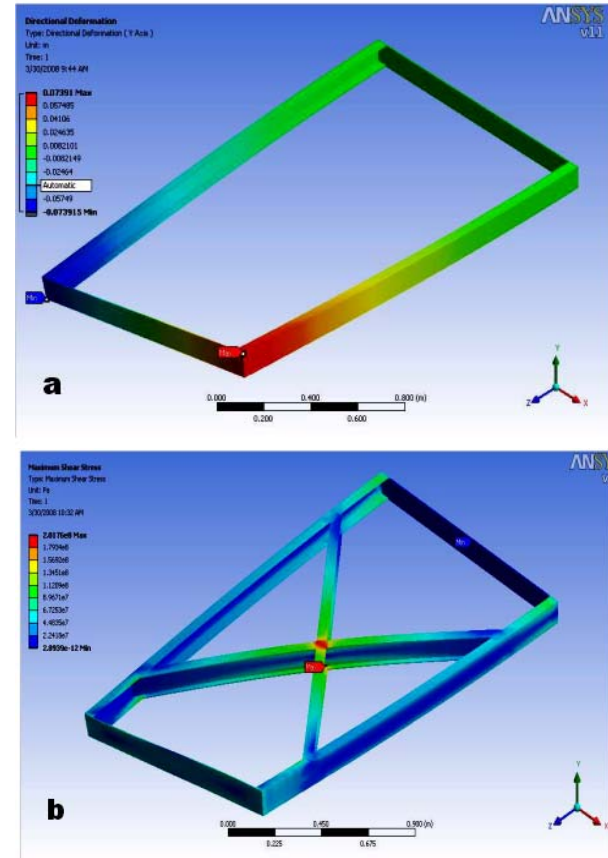


Fig. 1. Ansys simulation of: Steel ladder frame (a) and Large X-bracing frame (b) [8, 9]

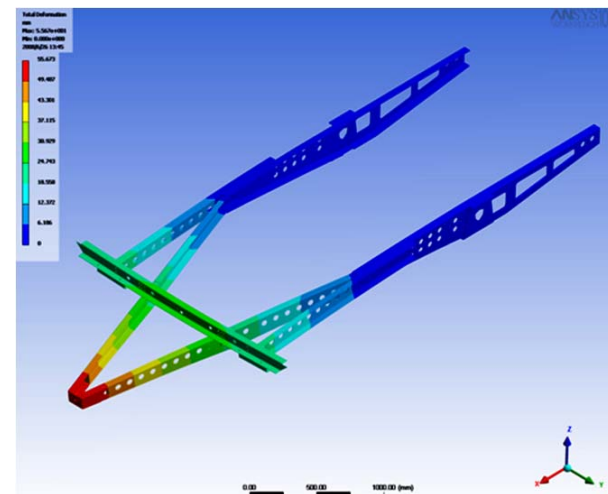


Fig. 2. FEA stress analysis on frame chassis [11]

2. METHODOLOGY

The flow chart in Fig. 3 illustrates the various steps followed towards static and fatigue analysis in this work. The detailed methodology for FEA analyses are detailed below.

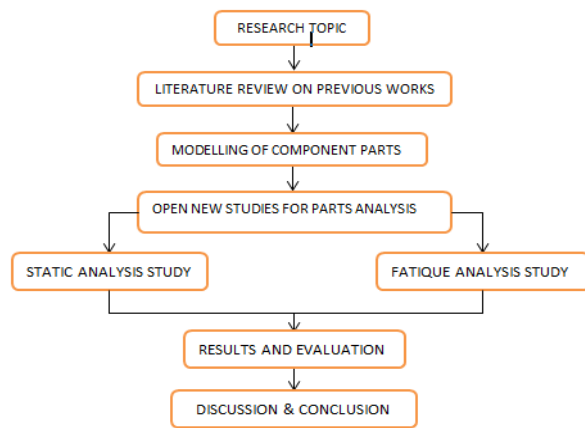


Fig. 3. Study Methodology Flow Chart

2.1 Methodology for the FEA analysis

The procedures below were followed during the Finite Element Analysis on the chassis and hitch:

1. Set the analysis units: Activation of simulation new study
2. Creating study for the analysis: Static study icon from the simulation menu was activated.
3. Assigning material: Material is applied to the model using custom icon from material window to specify the required material properties from design calculation.
4. Meshing: The model is meshed using default setting.
5. Applying fixtures: Fixed geometry icon is selected for restraining both ends.
6. Applying forces: External loads icon is active to apply load and direction to the model.
7. Run for analysis: After desire result from meshing the run icon is active to analyse the model
8. Visualizing and verifying the result: Checking on the results for stress and strain, displacement, and factor of safety by comparing to hand calculation.
9. Compile the report: After satisfactory result, report was compiled for recommendation.

2.2 FEA Analysis on Chassis and Hitch

The frame (chassis) being one of the most important components of the harvester, supports the entire load of the machine and this makes it an integral part that have to be analysed to determine how well it is able to carry the load being subjected to. Also the hitch which provides a link or attachment also needed to be analysed to ascertain that it does not fail during course of operation.

The design for the chassis was adopted from [14] with similar analysis carried out. Both statics and fatigue analysis were conducted on the chassis and hitch to determine how well the chassis had been designed. The ability of the chassis and hitch to withstand damage and deformation based on the designed load was also determined. The static analysis performed involved fixing the chassis in areas where it is to be supported by tyres and shafts.

The modeled chassis was assigned a material of AISI 1045 after thorough material consideration. The loads were applied onto two sections of the chassis considering the load uneven distribution. Load of 1994.92 N was applied to the front section while load of 3989.84 N was applied to rear section taking into consideration the gravitational force through the sections. The study was then simulated. Both the chassis and hitch were modelled using Solidworks (2014). New simulation studies were performed for static and fatigue analysis on the components. The stages and results obtained in the analysis are discussed below.

3. RESULTS AND DISCUSSION

To have a clear view of the various analyses, the different views of the combined harvester from the design images are contained in Fig. 4. The analyses will be restricted to the chassis frame and the hitch as detailed below.

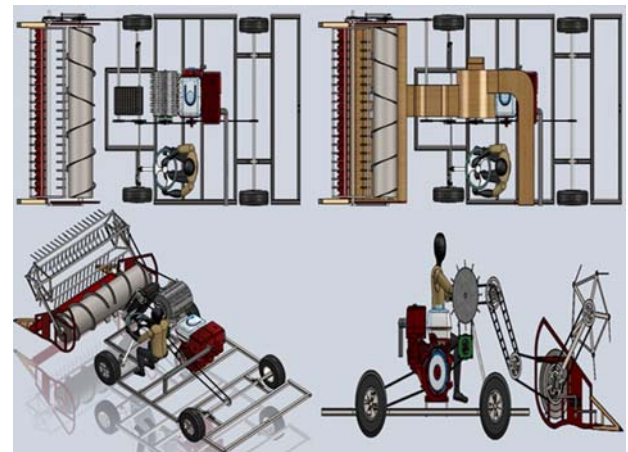


Fig. 4. Different Views of Combined Harvester Chassis and Components

3.1 Chassis Static Analysis Result

Fig. 5 showed the chassis frame, the chassis supporting fixtures (arrows pointing upwards) and the loading points (arrow pointing downwards) including the location of centre of gravity. The loading was not uniformly distributed. The load of about 2 kN applied to the front of the chassis was to account for the engine and the frontal loadarge chunk of 3989.8 N was applied to the rear portion of the chassis while about half of the rear load applied to the front portion.

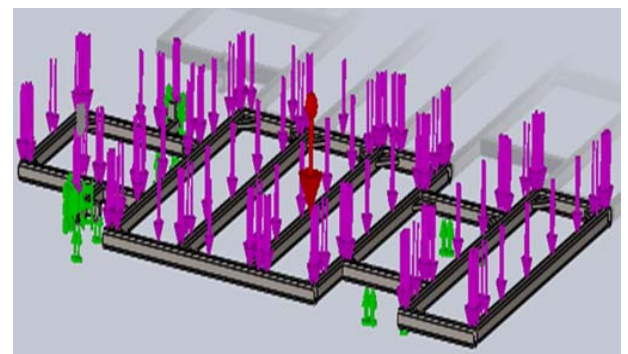


Fig. 5. Fixtures and load Application on the chassis for FEA Analysis

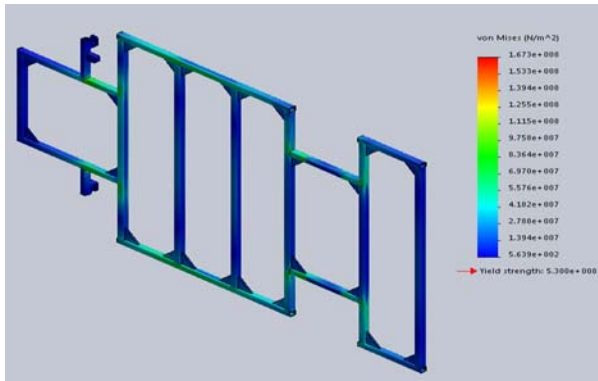


Fig. 6. Stress Distribution Result of the FEA Analysis

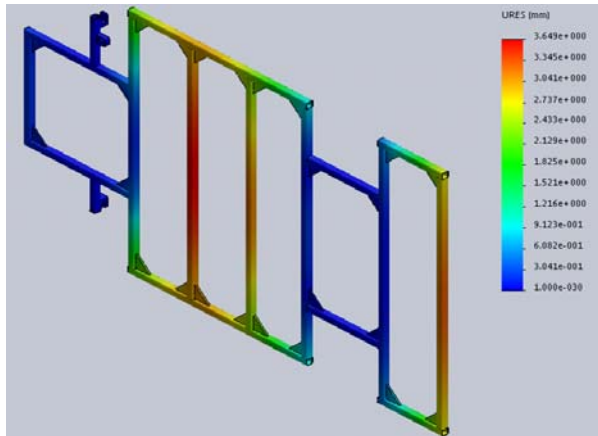


Fig. 7. Displacement Distribution Result of the FEA Analysis

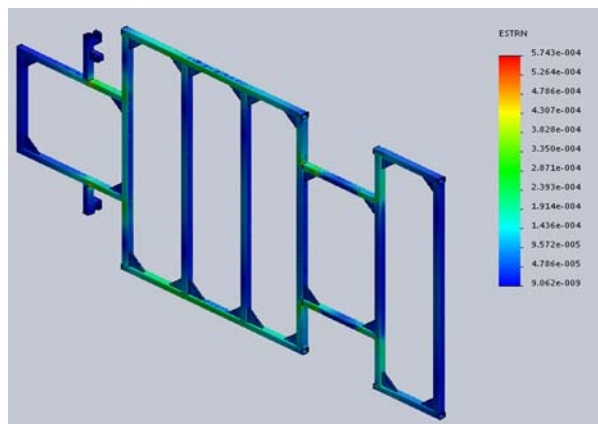


Fig. 8. Strain Distribution Result of the FEA Analysis

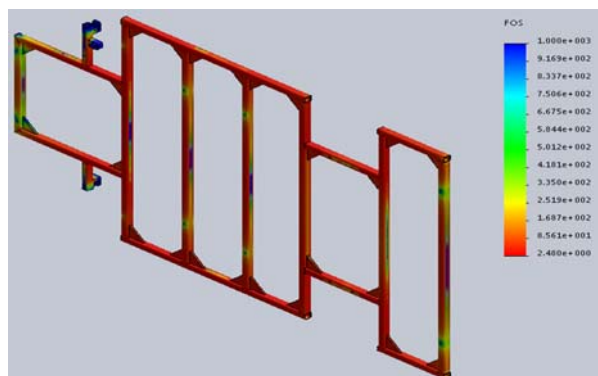


Fig. 9. Factor of Safety Result of the FEA Analysis

The greenish yellowish area in some part of the chassis as shown in Fig. 6 revealed the most stressed

region on the chassis. Red coloured regions in Fig. 7 are depicting region of high displacement. Fig. 8 also revealed greenish yellowish area representing highly strain region just similar to the one shown in Fig. 6. The last result on static analysis gave a minimum factor of safety of 2.48 (see Fig. 9).

3.2 Chassis Fatigue Analysis Results

The detailed results of fatigue analyses of the chassis are given below.

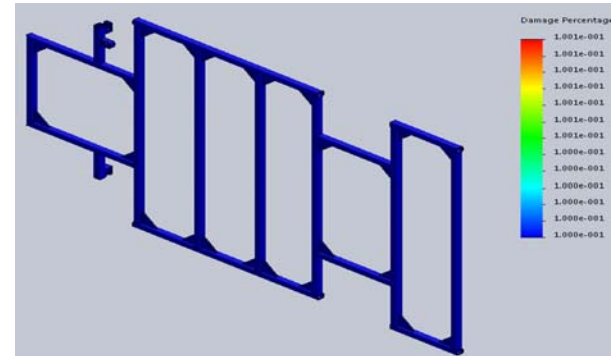


Fig. 10. Damage Percentages Result of the fatigue Analysis

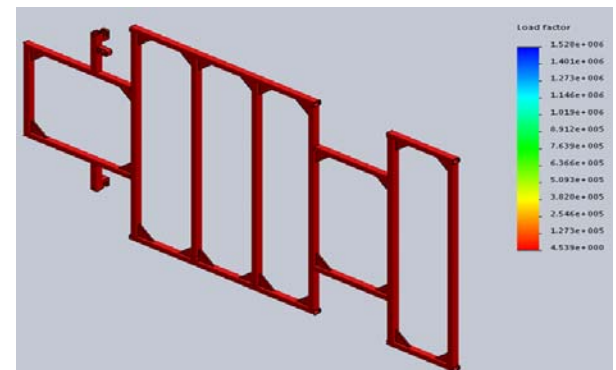


Fig. 11. Load Factor Result of the Fatigue Analysis

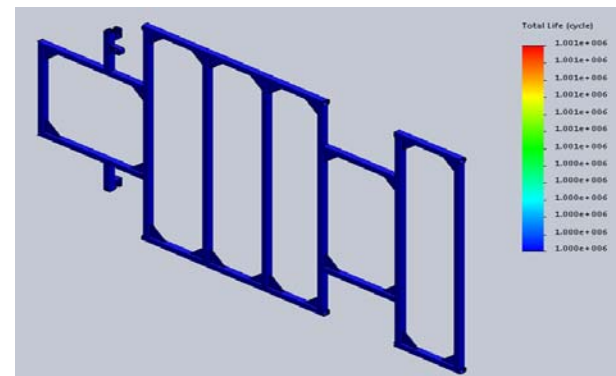


Fig. 12. Total Life Cycle Results for the Chassis

From Fig. 10, the chassis maximum damage percentage is 0.1. Fig. 11 shows a maximum load factor of 1.528×10^6 with the maximum total life of the chassis at 1.001×10^6 cycles as shown in Fig. 12.

Looking at the chassis insignificant damage percentage, load factor greater than one and the total life cycle, all clearly pointed out that the chassis have been well designed.

Table 1 and 2 showed the comparison of results done based on present work and adopted work of Abdulrahman [14], for static and fatigue analysis of the chassis respectively. From the Table 1, it will be seen that there was a considerable difference in the results obtained. These differences are attributed to type of material selected for the chassis and slight

improvement in the chassis design. The improvements led to an improved chassis of higher factor of safety. The analysis done by Kotari and Gopinath [17] on chassis frame, adding stiffener at region of maximum stress in other to improve it payload, revealed results of 7.3 mm deformation and factor of safety of 1.27.

FEA Analysis (Static)	Abdulrahman FEA		Present Work FEA	
	Min	Max	Min	Max
Stress	$1.43 \times 10^2 \text{ N/m}^2$	$2.43 \times 10^8 \text{ N/m}^2$	$5.6 \times 10^2 \text{ N/m}^2$	$1.67 \times 10^8 \text{ N/m}^2$
Strain	7.40×10^{-10}	8.70×10^{-4}	9.06×10^{-9}	5.743×10^{-4}
Displacement	$1.0 \times 10^{-30} \text{ mm}$	5.57mm	$1.0 \times 10^{-30} \text{ mm}$	3.649 mm
Factor of safety	1.66		2.48	

Table 1. Chassis FEA Static Analysis Results Comparison

Fea Analysis (Fatigue)	FEA Results	
	Abdulrahman (2014)	Present Work
Max. Damage Percentage (%)	0.705	0.1
Max. Load Factor	5.21×10^6	1.528×10^6
Total Life	1.0×10^6	1.0×10^6

Table 2. Chassis Fatigue Analysis Results

3.3 Hitch Static Analysis Results

There are various types of hitches currently available in the market differing with type of application. For this design, is a standard rear drawbar hitch that can carry load up to one tonne.it will allow rotation of two axes and will be bolted to the chassis. This type of hitch will not only allow all the attachments to be mounted or dismounted easily but also removable quickly and safely.

It can be evidenced from various researchers, design and analysis companies that several work has been done on different types of hitches. To mention few, Adnoor et al. [18] performed non-linear buckling analysis on tow bar. Alimardani et al. [19] and Khannade et al. [20] goes further from performing the FEA analysis to design dynamometer and transducers that can measure forces on hitches during operation using three point hitches of a tractor.

Since hitch is the critical component that allows the harvester and the trailer to move together. The great importance of hitch makes it justifiable to perform analysis on the component, to determine it strength during the operation. Like that of the chassis analysis, similar steps were followed in other to carry out statics analysis on the hitch. Cast iron material of yield strength of 526 Mpa was selected for the hitch.

Fig. 13a shows the fixed region (with green arrows) of the hitch while the pink arrow region depicts region of applied load. Fig. 13b shows the meshed image of the hitch.

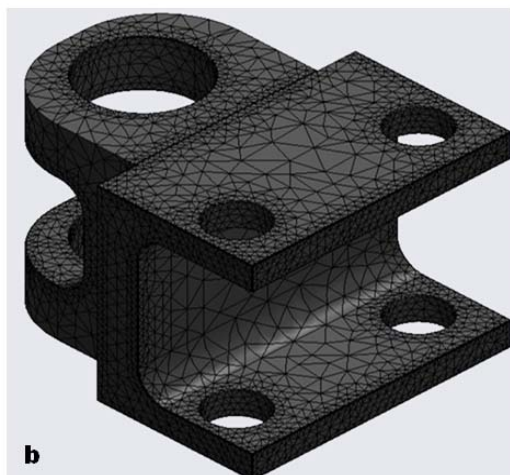
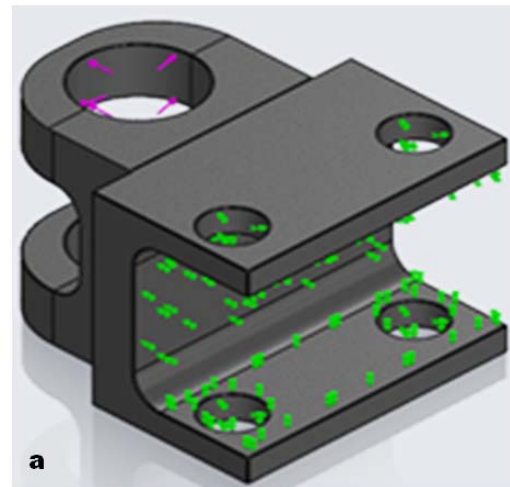


Fig. 13. Image of Hitch: Fixture and Load Application (a) and Meshed Image (b)

The results obtained from the simulation, revealed a maximum stress of $4.866 \times 10^7 \text{ N/m}^2$ with highly stressed region around the region of applied load as

shown in Fig. 14a. The same region also shows region of greatest displacement and strain, with maximum displacement of $1.532 \times 10^{-2} \text{ mm}$ and maximum strain of 5.669×10^{-4} as shown in Fig. 14b and Fig. 14c respectively. Fig. 14c revealed factor of safety of 2.

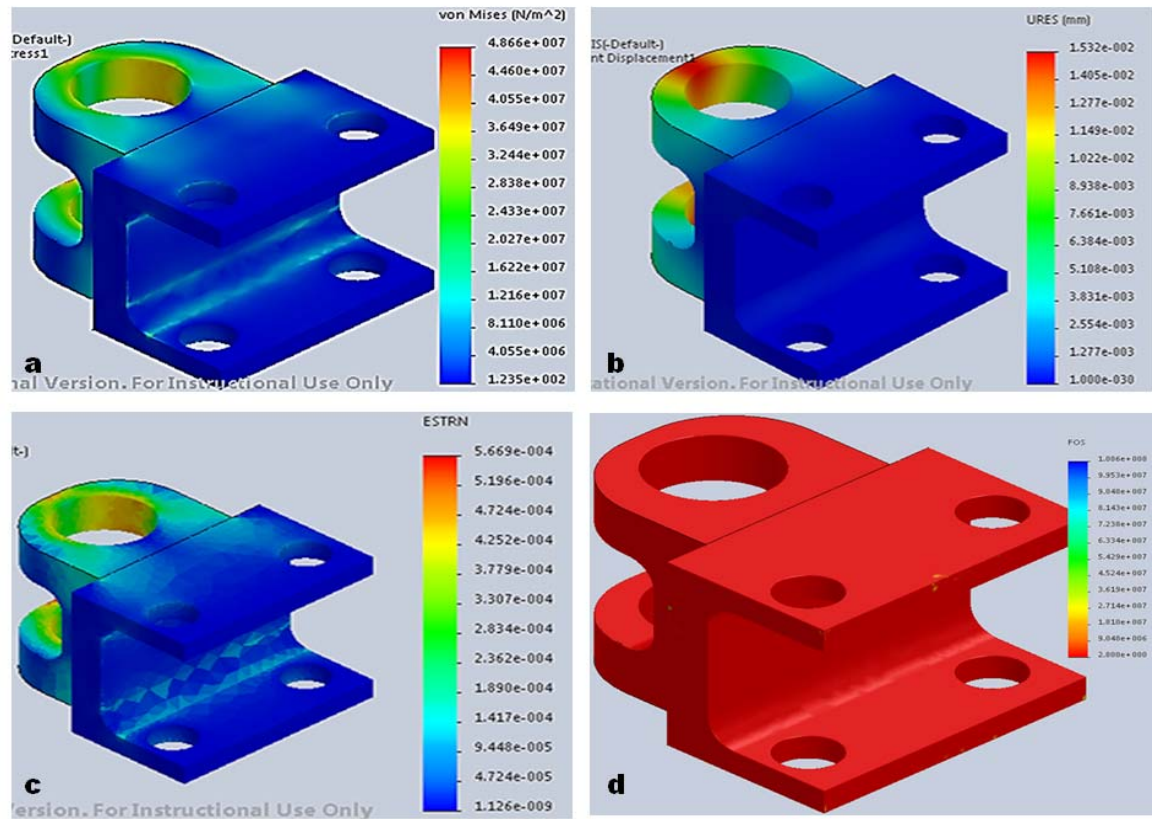


Fig. 14. FEA Analysis showing Distributions of: Stresses (a), Displacement (b), Strain (c), and Factor of Safety (d).

The fatigue analysis carried out for the hitch revealed the results shown in the Table 3. Tables 1-4 has clearly presented summary of the results obtained from the chassis and hitch FEA analysis. Looking at the

results showing negligible displacement with good factor of safety, it can therefore be said that both the chassis and hitch have been designed to withstand subjected loads and resist damage.

FEA Analysis (Fatigue)	FEA Results	
	Min	Max
Damage Percentage (%)	0.1	1.832
Load Factor	3.641	1.284×10^6
Total Life Cycle	5.457×10^4	1.0×10^6

Table 3. Hitch Fatigue Analysis Results

FEA Analysis (Static)	FEA Results	
	Min	Max
Stress (N/m^2)	1.235×10^2	4.865×10^7
Displacement(mm)	1.0×10^{-30}	1.53×10^{-2}
Strain	1.126×10^{-9}	5.669×10^{-4}
Factor of safety	2.8	

Table 4. Hitch FEA Static Analysis Results

4. CONCLUSIONS

Finite element analysis (FEA) results obtained for the chassis and hitch clearly revealed that the models have been adequately designed to withstand maximum

subjected load the components may be carrying provided high yield strength materials are used in the construction of such components.

The static analysis of the chassis revealed maximum displacement of 3.6mm and a factor of

safety of 2.48. The fatigue analysis gave a damage percentage of 0.1 and maximum load factor of 1.52×10^6 when subjected to maximum load. The hitch gave a maximum displacement of 1.53×10^{-2} mm and a factor of safety of 2.8 when subjected to maximum load. FEA simulation software (SolidWorks 2014) have clearly demonstrated that it is a viable tool in analyzing models as it showed areas of possible failures that needed additional strengthening mechanism to prevent imminent failure.

In conclusion, the analysis was successful on account of better results obtained when compared to previous work. The analysis also showed that it is possible to design for manufacture, chassis of a multi-purpose low-cost combine harvester that will make use of locally sourced materials like the square hollow pipe.

5. REFERENCES

- [1] Talabi, S. O. and Onasanya, O. *Agricultural Introduction*. 2013 22/09/2014]; Available from: <http://www.onlinenigeria.com/agriculture/?blurb=480>.
- [2] Aderohunmu, M., Is Nigeria's agricultural sector improving?, 2012.
- [3] Lamidi, W. A. and Akande, L. O., A Study of Status, Challenges and Prospects of Agricultural Mechanization in Osun State, Nigeria. 1(1): p. 001-008, 2013.
- [4] Takeshima, H., Pratt, A. N., and Diao, X., Agricultural Mechanization Patterns in Nigeria: Insights from Farm Household Typology and Agricultural Household Model Simulation., IFPRI, 2013.
- [5] Abdulkareem, Y. A., Indigenous Technology, Past, Present and Futures, in Workshop for the Processing Of Raw Materials: Raw Materials Research and Development Council, 1992.
- [6] Olukunle, O. T., *Developments in Grain Harvesting Mechanisation*. Journal of Agricultural Engineering and Technology (JAET). 18(1), 2010.
- [7] David, V. H., *Fundamentals of Finite Element Analysis*. First ed.: McGraw-Hill, 2004.
- [8] Wakeham, K. J. *Introduction To Chassis Design*. 2009 14/04/2014]; Available from: <http://www.keithwakeham.com/Files/ChassisDesign.pdf>
- [9] Wakeham, K. J. *Ansys simulation of Steel ladder frame and Large X-bracing frame* 14 April, 2014]; Available from: <http://www.keithwakeham.com/Files/ChassisDesign.pdf>.
- [10] Reina, J., Structural Analysis of a Bicycle Frame using Finite Element Techniques, 2007.
- [11] Jurgens. *Chassis Stress Analysis*. 2010 [cited 06 October, 2016]; Available from: <http://jurgens.com.au/Range/Smart-Technology.aspx>
- [12] Patel H, Pancha, K. C., and Jadav, C. S., *Structural Analysis of Truck Chassis Frame and Design Optimization for Weight Reduction*. International Journal of Engineering and Advanced Technology (IJEAT). 2(4), 2013.
- [13] Patil, H., Kachave, S. D., and Deore, E. R., *Stress Analysis of Automotive Chassis with various Thicknesses*. Journal of Mechanical and Civil Engineering. 6(1): p. 44-49, 2013.
- [14] Abdulrahman, K. O., Design of Low-Cost Vehicle for Nigerian Rural Farmers for Transportation of Farm Produce, 2014.
- [15] Singh, A., Soni, V., and Singh, A., *Structural Analysis of Ladder Chassis for Higher Strength*. International Journal of Emerging Technology and Advanced Engineering. 4(2), 2014.
- [16] SolidWorks, 2014.
- [17] Kotari, S. and Gopinath, V., *Static and Dynamic Analysis on Tatra Chassis*. International Journal of Modern Engineering Research (IJMER). 2(1): p. 086-094.
- [18] Adnoor, A., Guruprasad, H. L., and Maurthi, B. H., *Nonlinear Buckling Analysis of a Tow Bar*. Indian Journal Science. 9(1): p. 088-095, 2014.
- [19] Alimardani, R., Fazel, Z., Akram, A., Mahmoudi, A., and Varnamkhashi, M. G., *Design and Development of a three-point hitch dynamometer*. Journal of Agricultural Technology. 4(1): p. 37 - 52, 2008.
- [20] Khannade, P., Chitnis, A., and Jagdale, G., *Design and Stress Analysis of Tow Bar for Medium Sized Portable Compressors*. International Journal of Research in Engineering & Advanced Technology. 2(3): p. 1-6, 2014.

Authors: Researcher. Mr Kazim Olawale Abdulkarim, Department of Mechanical Engineering, University of Derby, Derby. DE22 1GB, UK. +447553081438. **Lecturer. Mr Kamardeen Olajide Abdulrahman**, Department of Mechanical Engineering, University of Ilorin, P.M.B. 1515, Nigeria, +2348061596252. **Lecturer. Dr Ismaila Idowu Ahmed**, Department of Materials and Metallurgical Engineering, University of Ilorin, P.M.B. 1515, Nigeria, +2347018271354. **Lecturer. Sulaiman Abdulkareem**, Department of Mechanical Engineering, University of Ilorin, P.M.B. 1515, Nigeria, +2348058219222. **Lecturer. Mr Jeel Adegunle Adebisi**, Department of Mechanical Engineering, University of Ilorin, P.M.B. 1515, Nigeria, +2348030422330. **Lecturer. Dani Harmanto**, Department of Mechanical Engineering, University of Derby, Derby. DE22 1GB, UK, +447773057610.

E-mail: zeem2002@yahoo.com
olajideabdulrahman@yahoo.com
ismaila.ahmed@yahoo.com
sulkarm@yahoo.com
adebisijeel@gmail.com
d.harmanto@derby.ac.uk