Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 06, June-2016

Structural Analysis of Crawler Excavator Chassis using FEM

Prashant I Bhandari[1], Mruthyunjaya S Alur[2],
[1,2] PG students,
Basaveshwar Engineering College,
Bagalkot, Karnataka.

Dr. S N Kurbet[3],
[3] HOD Mechanical Department.
Basaveshwar Engineering College,
Bagalkot, Karnataka.

Abstract- **Modern infrastructure needs earthmovers of different loading capacities .With the requirement from the infrastructure, the loading condition is defined. The need of the working load varies the deformation in the crawler excavator chassis. This project work illustrate the effect of different loads on chassis of the vehicle, the analysis of the stresses and deformations during the working .The motto of this work is to increase the factor of safety And to strengthen the chassis, structure change can help in this aspect. The main update in this project is to introduce the extra stiffener and to give extra strength to the structure. Project is to enhance the load carrying capacity of the structure. The loads considered in this work are the entire weight of the body which is on the chassis is assumed as the point load on sleeve ring (boom carrying platform ring) whose degree of freedom which allows it to rotate in X direction and other load is bending moment which can be defined as force multiplied by perpendicular distance. Primarily the model is generated in CAD tool CATIA V5 R19, then. It is imported in HYPER MESH 12.0 it is meshed in after applying the material and material properties .The next step is to export the model in solver deck the extension of the file is in .inp .the file then run into ABAQUS 12.0 for result like maximum stress according to von-mises failure theory, deformation and reaction forces.**

Keywords—Sleeve Ring; Crawler Excavator Chassis; CATIA; ABAQUS; HYPERMESH; Solver Deck.

I. INTRODUCTION

The present work essentially demonstrates the analysis of stability of chassis structure. The main loads carried by the chassis are considered overall weight of the excavator and load carried by the boom. Figure 1 shows the x type chassis for crawler excavator chassis [1]. B. Ramya, Sridhar Reddy of J B institute of engineering and technology worked on the design and structural analysis of automotive vehicles having loading capacity 1 ton to 40 tons of load. Here the project considered the maximum payload of 70 tons this load is considered as a point load on the sleeve ring (boom carrying platform ring) the load carried by boom and bucket is limited to the 70 tons. Here one more thing to consider is the moment force the moment it can define as the force multiplied by perpendicular distance. The perpendicular distance here we considered as per standard as 3.1 meters because when the boom at its maximum length its 3.1 meter for calculations purpose to get moment it should be multiplied by the force acting on the boom. The moment changes with respect to angle of the boom because as the angle of the boom changes the perpendicular distance also changes. The materials for the chassis is fixed that is steel having density 7850 kg and poisons ratios 0.3 the model of chassis is generated using any CAD tools available then it is imported in HYPER MESH. Hyper mesh is software globally used for meshing purpose. Meshing can be termed as discretization of the component in other words the component is divided into numerous small regular well known elements whose mechanical properties are known. The quality of the meshed components is done and warpage, aspect ratio, Jacobean and many quality parameters are checked is then solved using the ABAQUS solver. This step is called as processing and after the processing the post processing is done manually. Essential tables are prepared and conclusion is decided.

Figure 1 Typical x type chassis

METHODOLOGY

The method of structural analysis of crawler excavator chassis essentially consists of these methods.

- Gathering the information about the component either it may be in the form of information from the person or by the journal papers submitted upon the same work. We can refer the combined information to get essential information about the component [2].
- Once we get the required information about the component, the computer aided three dimensional model of the component. This modeling can be done in various CAD softwares like CATIA, SOLIDWORKS, PRO-E CREO, SOLIDEDGE, and AUTOCAD. After modeling the component of fully constrained structure, the model is converted natural file format i.e. IGES or IGS format.
- Next step for the process of analysis of structure of a crawler chassis analysis is to import the IGES file of CAD model of component into HYPERMESH. The software works for the step of pre-processing of the component, pre-processing involves specifying the material, material properties and mechanical properties of the material. The applied material here is structural steel with density 7850 kg/m^3, elastic property called Young's modulus of 2.15 MPa and Poisson's ratio of 0.3. After applying these properties the procedure follows creation of mid-surface in case of uniform thickness components i.e. shell meshing. In case of uneven thickness of the component solid meshing could be preferred. Shell meshing id done with 2d elements and solid meshing is done with 3d elements. In HYPERMESH these are known as C3D8K (HEXA), C3D6 (PENTA). After meshing of the component the quality check is done. Quality checking of meshed component involves

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 06, June-2016

i. warpage should be kept at 20

ii. Aspect ratio maintained at 5

iii. Jacobean for 0.5

iv. Tapering for 0.5

v. Quadratic min angle > 40

vi. Quadratic max angle <140

vii. Triangle min angle > 30

viii. Triangle max angle < 150

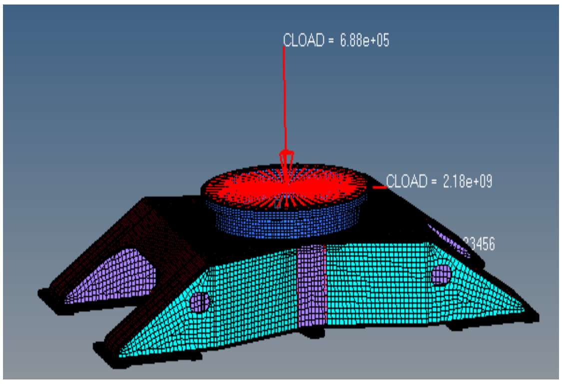

Figure 2 Meshed model of the component with loads

- After satisfying the quality parameters the model with boundary conditions, material properties, and mechanical properties is exported as a solver desk file in a separate folder. That file is run in ABAQUS 12.0 using command prompt. The procedure for solving the model in ABAQUS is as described. First it needs to copy the file location address then in command prompt change directory to the selected file location by typing [cd file location address] and move to check the directory by typing [dir]. We can get the results by commanding [ABAQUS job= file name without .inp extension].

- After satisfying the quality parameters the model with boundary conditions, material properties, and mechanical properties is exported as a solver desk file in a separate folder. That file is run in ABAQUS 12.0 using command prompt. The procedure for solving the model in ABAQUS is as described. First it needs to copy the file location address then in command prompt change directory to the selected file location by typing [cd file location address] and move to check the directory by typing [dir]. We can get the results by commanding [ABAQUS job= file name without .inp extension].

- After getting the raw data from the ABAQUS software i.e. the displacements, stresses and reaction forces one can compare those values with ultimate tensile strength of the given material. The stresses developed must be well within the optimum values. The comparison table is then prepared.

RESULTS AND DISCUSSIONS

From the results the displacements, stresses and reaction forces are as shown in the below images.

1) The analyzed result images are for loading condition of weight of the boom considered as the point load on the sleeve ring. Loading condition [LC1] depends on position of the boom. When the boom is along longitudinal direction with chassis axis.

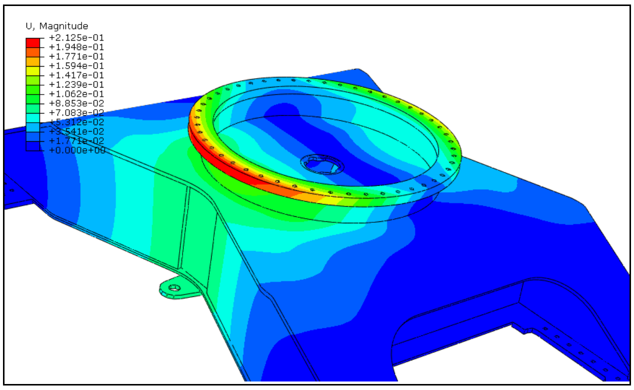

Figure 3 Displacements in x direction for point load.

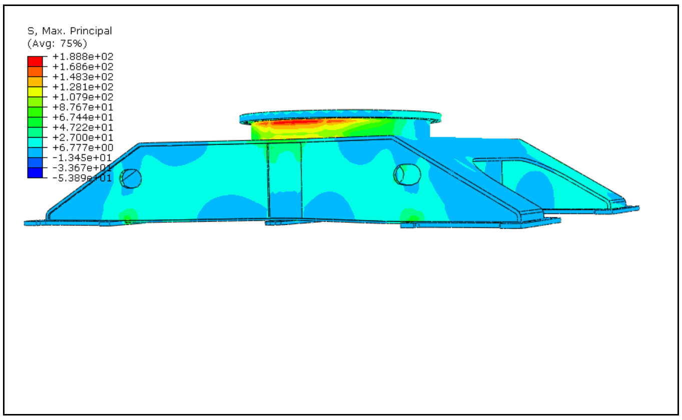

Figure4. Maximum principal stress according to von misses theory

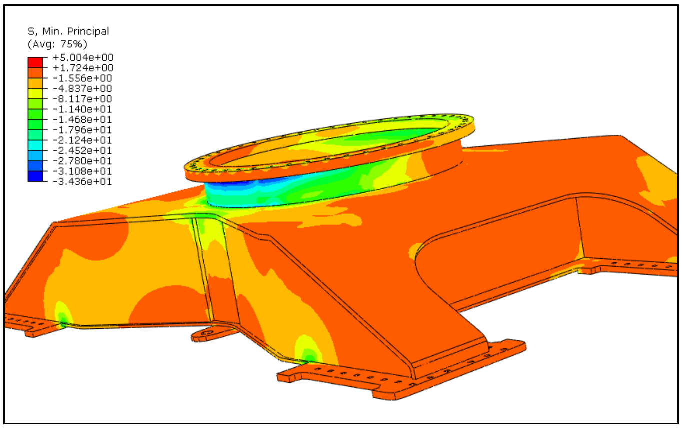

Figure 5 von mises minimum principal stress

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 06, June-2016

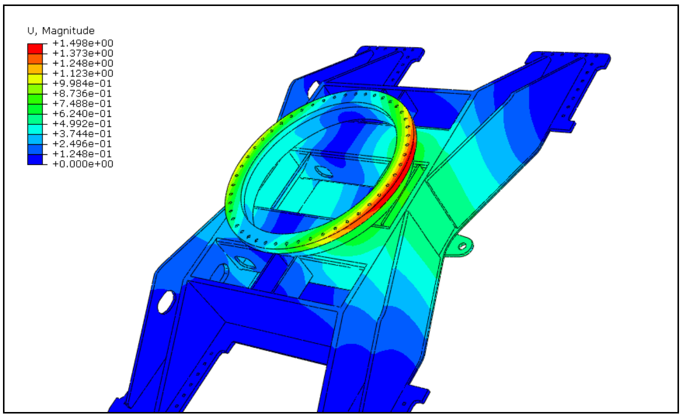

Figure 6 Displacements magnitude without top plate.

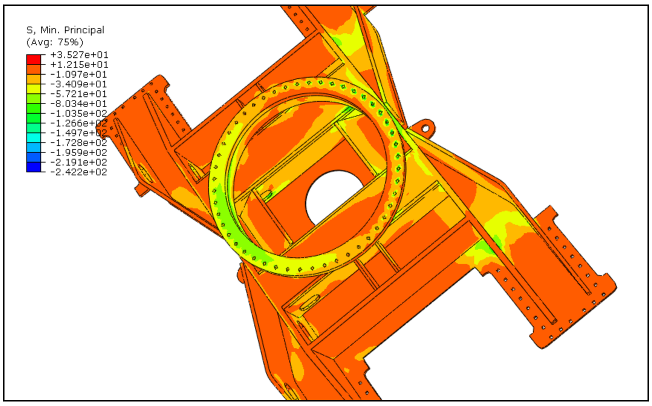

Figure 7 Von mises minimum principal stress without top plate

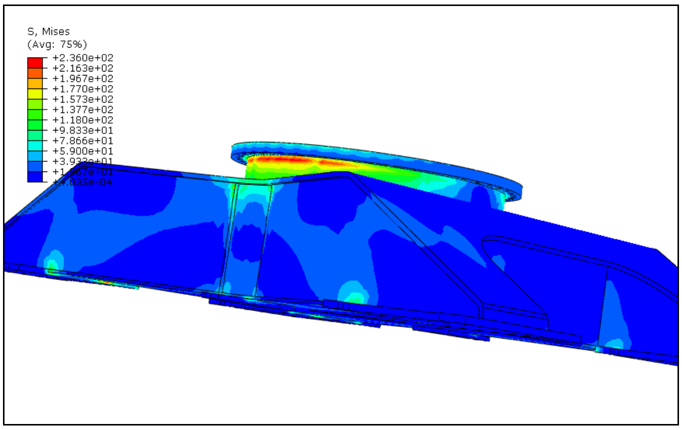

Figure 8 Von mises overall stress in longitudinal direction.

2) The analyzed result images are for loading condition of weight of the boom considered as the point load on the sleeve ring. Loading condition [LC2] depends on position of the boom. When the boom is along transverse axis with chassis axis.

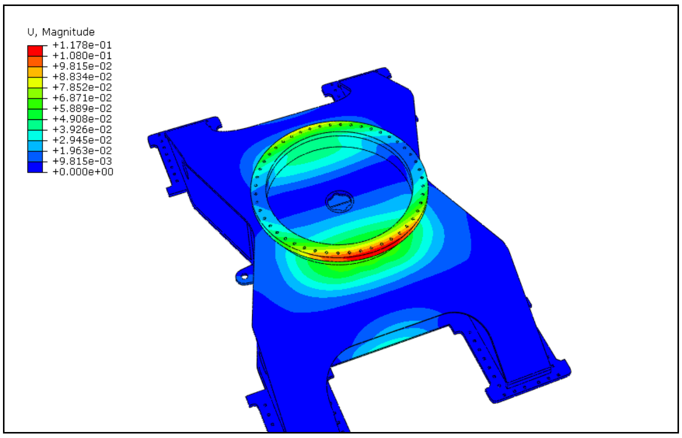

Figure 9 Displacements along transverse direction

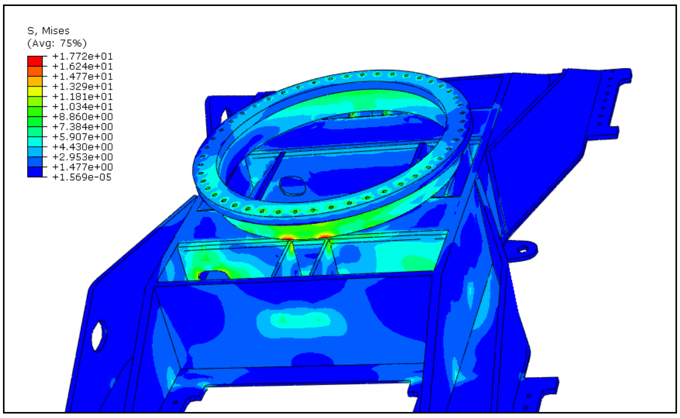

Figure 10 Von mises stress overall for LC2 without top plate

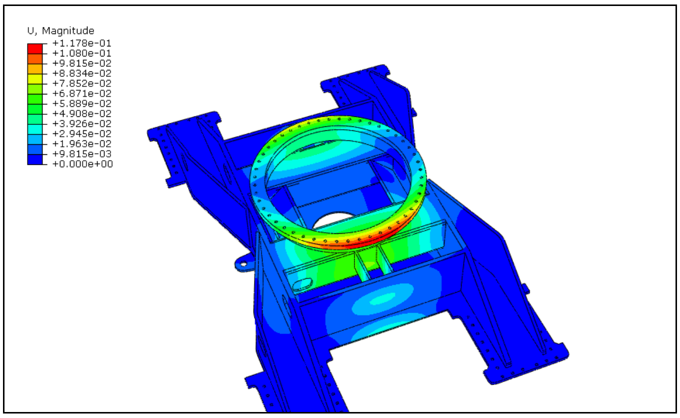

Figure 11 LC2 transverse displacement overall magnitude

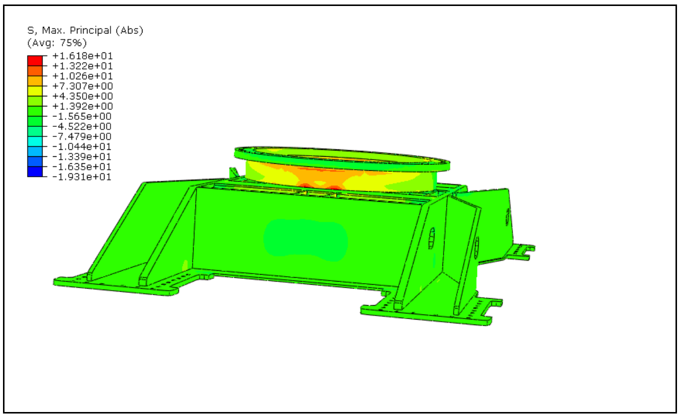

Figure12 Maximum principal stresses LC2

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 06, June-2016

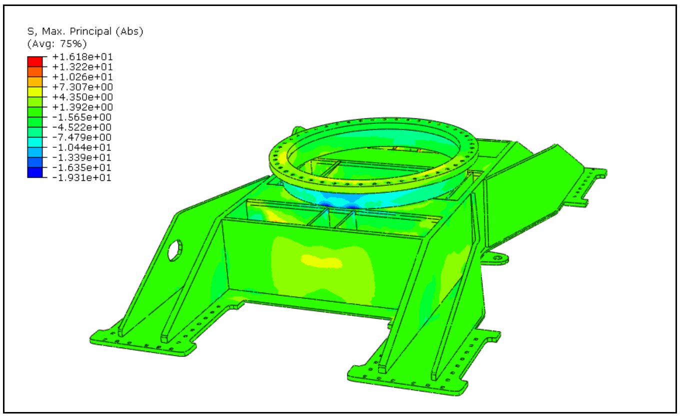

Figure 13 Maximum principal stress of LC2 without top plate

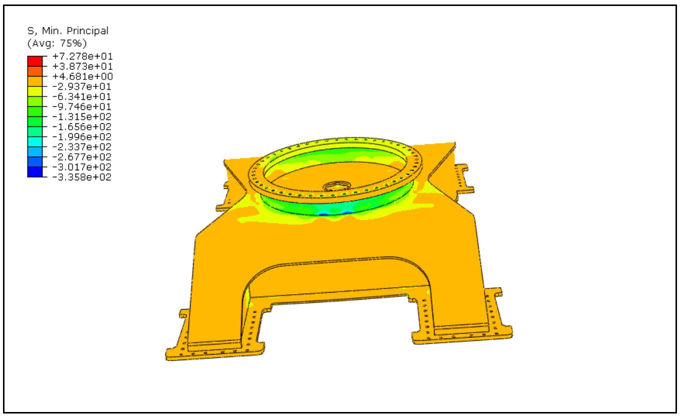

Figure 14 Min principal stress with top plate for LC2

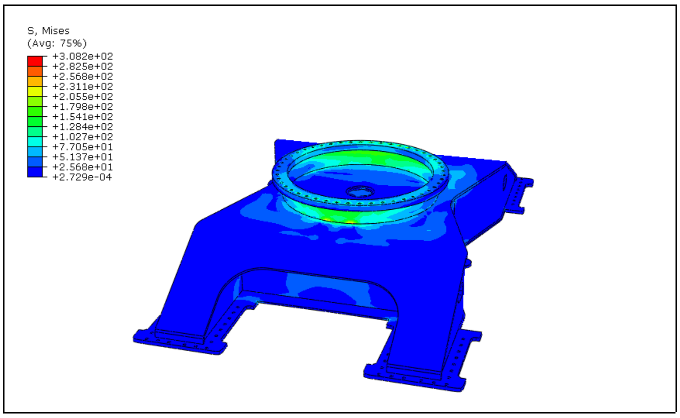

Figure 15 Von mises stresses for LC2 with top plate

By the above results the maximum stress reached is 308 MPa. The ultimate tensile stress for mild steel is 350 MPa. The stresses developed by the loading conditions in different positions of the boom are well within the ultimate stress of the material. Hence we get the required factor of safety.

CONCLUSION

From above results we can conclude that the stresses developed are well within the safe limit according to von-mises theory of ductile failure. For considered tonnage of the chassis the average maximum principal stress developed in loading condition 1 [LC1] is 236 MPa and in loading condition 2 [LC2] is 308 MPa. Both the stresses developed are well within the yield stress of the material. With this study, in future we can easily tackle and increase the loading capacity of the chassis.

REFERENCES

[1] B. Ramya, Sridhar Reddy, Dr. C. Udaya Kiran ‘Design and Structural Analysis of Chassis’ IJEDR | Volume 2, Issue 4 | ISSN: 2321-9939.

[2] Ketan Gajanan Nalawade, Ashish Sabu, Baskar P ‘Dynamic (Vibrational) and Static StructuralAnalysis of Ladder Frame’ (IJETT) – Volume 11 Number 2 - May 2014.

[3] ‘Introduction to finite elements in engineering’ by T R Chandrupatla and A D Belagundu.

[4] ‘Concepts and applications of finite element analysis’ by Robert D. Cook, David S.Malkus.